

NISA II
User's Manual



SPEED • ACCURACY • VALUE • TRUST

Disclaimer

Cranes Software, Inc. makes no warranty or representations in connection with NISA/HEAT, NISA/EMAG, NISA/ROTOR, NISA/CIVIL, NISA II, DISPLAY III/IV, NISA-COMPOSITE, ENDURE, NISAOPT, FEAP, NISA/3D-FLUID, DYMES, or any other software program of Cranes Software, Inc. (hereinafter referred to as NISA/DISPLAY SOFTWARE), express or implied, including any implied warranty of merchantability or fitness for a particular purpose. Cranes Software, Inc. makes no warranty and assumes no liability for the NISA/DISPLAY SOFTWARE. Cranes Software, Inc. assumes no responsibility for the use of the programs or for the accuracy or validity of any results obtained from the NISA/DISPLAY SOFTWARE. Cranes Software, Inc. shall not be liable for loss of profit, loss of business, or other financial loss which may be caused directly or indirectly by the NISA/DISPLAY SOFTWARE, when used for any purpose or use, or due to any defect or deficiency therein.

Any questions relating to the use or interpretation of the SOFTWARE or their operation should be directed to:

Cranes Software, Inc.
1133 E Maple Road, Suite 200
Troy, Michigan, 48083, USA
Tel: (248) 729-3010
Fax: (248) 689-7479

Notice for U.S. government users only: restricted rights legend.

Use, duplication or disclosure by the Government is subject to restrictions as set forth in subparagraph (c) (1) (ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013.

Copyright 2005-2014 by the Cranes Software, Inc. 1133 E Maple Road, Suite 200, Troy, Michigan 48083, U.S.A.

All rights reserved. No part of this manual may be reproduced, stored in a retrieval system, or transmitted in any way without the prior authorization of Cranes Software, Inc.

Contents

<i>Preface</i>	<i>xix</i>
-----------------------	-------------------

<i>Guidelines for Using this Manual</i>	<i>xxi</i>
--	-------------------

<i>1 Introduction</i>	<i>1.1</i>
------------------------------	-------------------

1.1 General Description	1.1
1.2 NISA Suite of FEA Software	1.9
1.2.1 NISA Software	1.9
1.2.2 NISA Documentations	1.14

<i>2 Theoretical Overview</i>	<i>2.1</i>
--------------------------------------	-------------------

2.1 Introduction	2.1
2.2 Linear Static Analysis	2.9
2.2.1 Body Forces	2.13
2.3 Nonlinear Static Analysis	2.16
2.3.1 Formulation Methods	2.16
2.3.2 Basic Equations	2.18
2.3.3 Total Lagrangian Formulation	2.24
2.3.4 Centrifugal Softening Effect.....	2.33
2.4 Linear Dynamic Analysis	2.34

2.4.1	Governing Equations of Motion	2.34
2.4.2	Eigenvalue Analysis	2.38
2.4.3	Normal Mode Method	2.41
2.4.4	Damping Characteristics.....	2.44
2.4.5	Ground Motion	2.46
2.4.6	Modal Transient Dynamic Analysis	2.53
2.4.7	Frequency Response Analysis	2.58
2.4.8	Random Vibration Analysis.....	2.62
2.4.9	Shock Spectrum Analysis	2.67
2.4.10	Direct Transient Analysis	2.72
2.4.11	Direct Frequency Response Analysis	2.77
2.4.12	Ground Motion	2.78
2.5	Buckling Analysis	2.80
2.6	Heat Transfer Analysis	2.82
2.6.1	Introduction and Basic Equations	2.82
2.6.2	Finite Element Equations:.....	2.89
2.6.3	Surface Radiation Heat Transfer.....	2.91
2.7	Constitutive Equations	2.98
2.7.1	Introduction.....	2.98
2.7.2	Ideal Elastic Material Model.....	2.98
2.7.3	Elastoplastic Material Model.....	2.106
2.7.4	Hyperelastic and Rubber-Like Material Model.....	2.124
2.7.5	Constitutive Relations for Thermo-Elastic-Plastic Creep.....	2.133
2.7.6	Constitutive Relations for Elasto-Plastic-Cracking concrete material model.....	2.136
2.8	NISA Elements	2.144
2.8.1	Concentrated Mass Elements (NKTP = 25-30).....	2.144
2.8.2	Spring, Spar and Cable Elements (NKTP = 14-22, 38 and 45).....	2.145
2.8.3	2-D Solid Elements (NKTP = 1, 2, and 3).....	2.152
2.8.4	3-D Solid Element (NKTP = 4).....	2.172
2.8.5	3-D General Shell Element (NKTP = 20).....	2.178
2.8.6	2-D Gap and Friction Element (NKTP = 49)	2.189

3 NISA Capabilities

3.1

3.1 Introduction	3.1
3.2 Coordinate Systems, Degrees of Freedom and Sign Convention	3.3
3.2.1 Types of Coordinate Systems.....	3.3
3.2.2 Degrees of Freedom	3.7
3.2.3 Sign Convention	3.8
3.3 Linear Static Analysis (STATIC)	3.10
3.3.1 Moving Loads in Static Analysis	3.13
3.4 Nonlinear Static Analysis (NLSTATIC)	3.14
3.4.1 Types of Nonlinearities and Formulation	3.14
3.4.2 Load Stepping	3.17
3.4.3 Solution Schemes	3.17
3.4.4 Basic Analysis Output	3.27
3.5 Linear and Nonlinear Dynamic Analysis	3.28
3.5.1 Overview of Analysis Types and Capabilities:	3.28
3.5.2 Eigenvalue Analysis (EIGENVALUE)	3.30
3.5.3 Modal Dynamic Analysis	3.37
3.5.4 Modal Transient Dynamic Analysis (TRANSIENT)	3.41
3.5.5 Random Vibration Analysis (RANDOM)	3.44
3.5.6 Frequency Response Analysis (FREQUENCY)	3.51
3.5.7 Shock Spectrum Analysis (SHOCK)	3.53
3.5.8 Linear Direct Transient Analysis (LTRANSIENT)	3.62
3.5.9 Direct Frequency Response Analysis (DFREQ).....	3.66
3.5.10 Nonlinear Direct Transient Analysis (NLTRANSIENT) ..	3.67
3.6 Buckling Analysis (BUCKLING)	3.70
3.7 Stress Stiffening	3.72
3.8 Heat Transfer Analysis	3.74
3.8.1 Introduction and Analysis Types	3.74
3.8.2 Heat Transfer Material Models	3.76
3.8.3 Boundary Conditions and Loading	3.77
3.8.4 Analogous Field Problems	3.80
3.9 Material Models	3.82
3.9.1 Linear Elastic Material Model (*MATERIAL)	3.82

3.9.2 Isotropic Elastoplastic Material Model (*PLASTIC)	3.83
3.9.3 Hyperelastic Material Model (*HYPEREL)	3.86
3.9.4 Anisotropic Elastoplastic Material Model (*APLASTIC)	3.87
3.9.5 Creep Material Model (*CREEP)	3.88
3.10 Kinematic Constraints	3.89
3.10.1 Definitions	3.89
3.10.2 Kinematic Constraint Rules	3.92
3.11 Static Loads	3.95
3.11.1 Types of Static Loads	3.95
3.11.2 Multiple Load Cases (*LDCASE) and Load Combination (*LDCOMB)	3.100
3.12 Data Checking and Error Messages	3.101
3.12.1 Data Checking	3.101
3.12.2 Warning and Fatal Error Messages	3.102
3.12.3 Solution Errors	3.102
3.12.4 Causes of Singularity and High Round-off Errors	3.104
3.13 High Performance Solvers	3.107
3.13.1 Wavefront Solver	3.107
3.13.2 Wavefront Optimizer	3.110
3.13.3 Sparse Matrix Solver	3.112
3.13.4 Iterative Solver	3.112
3.13.5 Parallel Direct Sparse Solver (PARDISO)	3.114
3.14 Output Features and Postprocessing	3.118
3.14.1 General Description of the Output	3.118
3.14.2 Postprocessing of the Results	3.119
3.14.3 Saving Element Results for Postprocessing	3.135
3.15 NISA Files	3.137
3.16 Restart Capabilities	3.139
3.17 System of Units	3.143

4 Element Library

4.1

4.1 Introduction	4.1
4.1.1 Element Types	4.1

4.1.2 Element Input	4.9
4.1.3 Element Output	4.11
4.2 2-D Plane Stress Element (NKTP = 1)	4.14
4.3 2-D Plane Strain Element (NKTP = 2)	4.18
4.4 Axisymmetric Solid Element (NKTP = 3)	4.22
4.5 3-D Solid Element (NKTP = 4)	4.26
4.6 3-D Thick Shell Element (NKTP = 5)	4.31
4.7 3-D Solid Piezoelectric Element (NKTP = 6)	4.35
4.8 3-D Laminated Composite Solid Element (NKTP = 7)	4.38
4.9 3-D Membrane Piezoelectric Element (NKTP = 8)	4.45
4.10 3-D Hybrid Solid Element (NKTP = 9)	4.48
4.11 3-D Membrane Element (NKTP = 10)	4.51
4.12 3-D Tapered Beam Element (NKTP = 11)	4.55
4.13 3-D Beam Element (NKTP = 12)	4.64
4.14 2-D Beam Element (NKTP = 13)	4.77
4.15 3-D Spar Element (NKTP = 14)	4.81
4.16 2-D Spar Element (NKTP = 15)	4.85
4.17 3-D Translational Spring Element (NKTP = 17)	4.88
4.18 2-D Translational Spring Element (NKTP = 18)	4.92
4.19 3-D General Shell Element (NKTP = 20)	4.95
4.20 3-D Torsional Spring Element (NKTP = 21)	4.101
4.21 2-D Torsional Spring Element (NKTP = 22)	4.105
4.22 Concentrated Mass Elements (NKTP = 25-30)	4.108
4.23 3-D Laminated Composite General Shell Element (NKTP = 32)	4.110
4.24 3-D Laminated Sandwich General Shell Element (NKTP = 33)	4.120
4.25 Axisymmetric Solid with Non-Axisymmetric Loading (NKTP = 34)	4.129
4.25.1 Description	4.129

4.25.2 Fourier Series Representation for Non-Axisymmetric Loads	4.133
4.26 Axisymmetric Shell Element (NKTP = 36)	4.139
4.27 Axisymmetric Shell with Non-Axisymmetric Loading (NKTP = 37)	4.142
4.27.1 Description	4.142
4.27.2 Fourier Series Representation for Non-Axisymmetric Loads	4.147
4.28 3-D General Spring Element (NKTP = 38)	4.153
4.29 3-D General Beam Element (NKTP = 39)	4.156
4.30 3-D Thin Shell Element (NKTP = 40)	4.161
4.31 2-D or Axisymmetric Gap Element (NKTP = 42)	4.166
4.32 3D Gap Element (NKTP = 43)	4.170
4.33 3-D Cable Element (NKTP = 45)	4.174
4.34 3-D Straight Pipe Element (NKTP = 46)	4.177
4.35 3-D Elbow Element (NKTP = 47)	4.183
4.36 3-D Translational Damper Element (NKTP = 48)	4.190
4.37 2-D Gap/Friction Element (NKTP = 49)	4.193
4.38 3-D Gap/Friction Element (NKTP = 50)	4.200
4.39 2-D Planar Element for Heat Transfer (NKTP = 101)	4.208
4.40 2-D Planar Element of Unit Thickness for Heat Transfer (NKTP = 102)	4.211
4.41 Axisymmetric Solid Element for Heat Transfer (NKTP = 103)	4.214
4.42 3-D Solid Element for Heat Transfer (NKTP = 104)	4.217
4.43 3-D Thick Shell Element for Heat Transfer (NKTP = 105)	4.221
4.44 3-D Bar Element for Heat Transfer (NKTP = 112)	4.225
4.45 3-D Curved Thin Shell Element For Heat Transfer (NKTP = 120)	4.228
4.46 3-D Convection Link Element (NKTP = 149)	4.233
4.47 3-D Radiation Link Element (NKTP = 150)	4.235

5 Input Setup and Executive Commands

5.1

5.1 Introduction and Input Data Setup	5.1
5.2 Free Format Input Features	5.5
5.3 Executive Commands	5.21
5.3.1 General Description	5.21
5.3.2 Executive Commands Common to all Analyses	5.22
5.3.3 Executive Commands Valid for Specific Analyses	5.35
5.4 Input Data Setup for Various Analysis Types	5.46
5.4.1 Summary of Applicable Data Groups for Various Analyses	5.46
5.4.2 Input Data Setup for Static Analysis	5.53
5.4.3 Input Data Setup for Eigenvalue Analysis	5.54
5.4.4 Input Data Setup for Nonlinear Static Analysis	5.56
5.4.5 Input Data Setup for Buckling Analysis	5.57
5.4.6 Input Data Setup for Heat Transfer Analysis	5.58
5.4.7 Procedure for Thermal Stress Analysis Following Heat Transfer Analysis.....	5.59
5.4.8 Procedure for Modal Dynamic Analysis.....	5.60
5.4.9 Input Data Setup for Linear and Nonlinear Direct Transient Dynamic Analysis	5.63
5.4.10 Input Data Setup for Direct Frequency Response Analysis	5.64
5.5 Restart Procedures	5.66
5.5.1 Introduction	5.66
5.5.2 Data Deck Setup for Restart Runs	5.66
5.5.3 General Requirements and Guidelines for Restart Runs	5.68
5.5.4 Special Features and Requirements in Static Analysis Restarts	5.71
5.5.5 Special Features and Requirements in Eigenvalue Analysis Restarts	5.74
5.5.6 Special Features and Requirements in Buckling Analysis Restarts	5.75
5.5.7 Special Features and Requirements in Nonlinear Static Analysis Restarts	5.76
5.5.8 Special Features and Requirements in Direct Transient Dynamic Restarts	5.76

5.5.9 Special Features and Requirements in Heat Transfer	
Restarts	5.78

6 Model Data

6.1

6.1 Introduction	6.1
6.2 TITLE	6.6
6.2.1 *TITLE Data Group - Problem Title	6.6
6.3 Element Data	6.7
6.3.1 *ELTYPE Data Group - Element Type Selection	
Table	6.7
6.3.2 *ELEMENTS Data Group - Element Definition	6.9
6.3.3 *E1 Data Group - Alternate Form for Element	
Definition	6.18
6.3.4 *RCTABLE Data Group - Real Constant Table	6.23
6.3.5 *LAMANGLE Data Group - Rotation Angles for	
Laminated Composite	
Elements	6.25
6.3.6 *LAMSQ2 Data Group - Composite Lamination	
Sequence Data	6.27
6.3.7 *BMDATA Data Group - Specification of	
Miscellaneous Data Required for	
General Beam Element (NKTP = 39)	6.37
6.3.8 *BMSECT Data Group - Specification of Cross	
Section Data for General Beam Element	
(NKTP = 39)	6.41
6.3.9 *NLSPRING Data Group - Specification of	
Force-Deflection Data for Nonlinear Springs	6.47
6.3.10 *WELEMENTS Data Group - Weld Elements	
Definition	6.52
6.4 Nodal Data	6.53
6.4.1 *LCSYSTEM Data Group - Definition of Local	
Coordinate Systems	6.53
6.4.2 *NODES Data Group - Nodal Coordinates	6.57
6.4.3 *G2 Data Group - Alternate Form for Defining	
Local Displacement Coordinate Systems at Nodes ..	6.64
6.5 Material Data	6.66
6.5.1 *MATERIAL Data Group - Material Property Data ...	6.66

6.5.2	*MATDIR1 Data Group - Definition of Orthotropic Material Axes with Respect to Nodes	6.69
6.5.3	*MATDIR2 Data Group - Definition of Orthotropic Material Axes with Respect to Elements	6.71
6.5.4	*MATHEAT Data Group - Material Properties Data for Heat Transfer Analyses	6.74
6.5.5	*HYPEREL Data Group - Hyperelastic and Rubber-like Material Properties.....	6.77
6.5.6	*PLASTIC Data Group - Plasticity Properties for Elastoplastic Material Models	6.82
6.5.7	*APLASTIC Data Group - Anisotropic Plastic Properties for Elastoplastic Material Models.	6.90
6.5.8	*CREEP Data Group - Creep Material Law	6.97
6.5.9	*PDAMPING Data Group - Proportional Damping Coefficients.....	6.102
6.5.10	*MATPIEZO data group – Material properties for piezoelectric analyses	6.104
6.6	Kinematic Constraint Data	6.107
6.6.1	*RIGLINK Data Group - Rigid Element Data	6.107
6.6.2	*MPCEQN Data Group - Definition of Linear Multi-Point Constraint (MPC) Equations	6.111
6.6.3	*CPDISP Data Group - Coupled Displacement Data.....	6.114
6.6.4	*CPTEMP Data Group - Coupled Temperature Data for Heat Transfer Analyses	6.117
6.7	Miscellaneous Data	6.119
6.7.1	*SETS Data Group - Definition of Sets	6.119
6.7.2	*VECTORS Data Group - Definition of Vectors	6.122
6.7.3	*TEMPFN Group Data - Temperature Dependency Curves for Heat Transfer Analysis	6.123
6.7.4	*TIMEAMP Data Group - Definition of Time Amplitude Curves	6.125
6.7.5	*PCHANGE1 - Enthalpy vs. Temperature Curve for Phase Change Analysis.....	6.130
6.7.6	*CRACKNODES Data Group – Crack Nodes Definition Data	6.132
6.7.7	*LDFILE data group – load history file data.....	6.136
6.7.8	*STFUNCTION data group – Definition of standard periodic time function	6.140
6.7.9	*PRESTN data group – definition of pre-stresses/strains	6.142
6.7.10	*FREQFUNCTION Data Group.....	6.144

6.8 Surface Contact Data	6.147
6.8.1 *CSURFACES Data Group - Definition of Contact Surfaces	6.147
6.8.2 *CELEMENTS Data Group - Definition of Contact Elements	6.157
6.9 Moving Load Data	6.163
6.9.1 *MLOAD Data Group - Moving Load Definition ..	6.163
6.9.2 *MPATH Data Group - Moving Load Path Definition	6.166

7 Analysis Data

7.1

7.1 Analysis Control Data	7.7
7.1.1 *LDCASE Data Group - Static Load Case Control	7.7
7.1.2 *EIGCNTL Data Group - Eigenvalue Analysis Control	7.11
7.1.3 *EVENT Data Group - Event Control	7.14
7.1.4 *FREQCNTL Data Group - Direct frequency analysis control	7.26
7.1.5 *HEATCNTL Data Group - Heat Transfer Analysis Control	7.27
7.1.6 *LDCOMB Data Group - Load Combination	7.28
7.1.7 *LCTITLE Data Group - Load Case Title	7.31
7.1.8 *MODEOUT Data Group - Mode Selection for Output Options	7.32
7.1.9 *TIMEINTEG Data Group - Time Integration Parameters for Transient Heat Transfer	7.34
7.1.10 *STEPSIZE Data Group - User Specified Time Steps	7.36
7.1.11 *CONVERGENCE Data Group - Convergence Criteria	7.37
7.2 Fourier Analysis Data	7.48
7.2.1 *FRCNTL Data Group - Fourier Analysis Control	7.48
7.2.2 *FRCOEF Data Group - Fourier Coefficients	7.49
7.2.3 *ANGSEC Data Group - Angular Sections for Response Calculation	7.52
7.3 Boundary Conditions and Loading	7.53

7.3.1	*SPDISP Data Group - Specified Displacement Boundary Conditions	7.53
7.3.2	*CFORCE Data Group - Concentrated Nodal Forces	7.56
7.3.3	*CFOLLOWER Data Group - Follower Concentrated Forces	7.58
7.3.4	*PRESSURE Data Group - Pressure Loads	7.60
7.3.5	*BEAMLOAD Data Group - Beam Loads	7.63
7.3.6	*BODYFORCE Data Group - Body Force Data	7.66
7.3.7	*NDTEMPER Data Group - Nodal Temperature Data	7.69
7.3.8	*NDTEMPDIF Data Group - Nodal Temperature Differences	7.71
7.3.9	*INITIAL Data Group - Initial Condition Data.....	7.73
7.3.10	*TBODYFORCE -Time Dependent Body Force Data Group.....	7.75
7.3.11	*L1 data group -Alternate Form for Specifying Pressure Loads	7.77
7.3.12	*MASTER - Master Degrees of Freedom for Guyan Reduction	7.79
7.3.13	*EDGFORCE - Edge Force for Shell Element Data Group.....	7.81
7.3.14	*ECDENSITY data group – Material properties for piezoelectric analyses.....	7.84
7.4	Boundary Condition and Loading for Heat Transfer	7.86
7.4.1	*NITEMP Data Group - Initial Temperatures	7.86
7.4.2	*SPTEMP Data Group - Specified Nodal Temperature Boundary Conditions.....	7.87
7.4.3	*CFLUX Data Group - Specified Concentrated Nodal Fluxes.....	7.88
7.4.4	*DFLUX Data Group - Specified Distributed Heat Fluxes.....	7.90
7.4.5	*ELHEATGEN Data Group - Specified Internal Heat Generation at Elements	7.92
7.4.6	*NDHEATGEN Data Group - Specified Internal Heat Generation at Nodes.....	7.94
7.4.7	*CONVBC Data Group - Convection Boundary Conditions.....	7.96
7.4.8	*RADBC Data Group - Radiation Boundary Conditions.....	7.99
7.4.9	*RADSURFACE Data Group - Surface Radiation Data.....	7.101
7.5	Output Control Data	7.103

7.5.1 *EIGOUT Data Group - Eigenvalue Output Control	7.103
7.5.2 *NLOUT Data Group - Selecting Output Parameters	7.107
7.5.3 *PRINTCNTL Data Group - Selective Printout Control	7.111
7.5.4 *REGIONS Data Group - Regions for Averaged Nodal Stress Printout	7.116
7.5.5 *STRSFILTER Data Group - Nodal Stress Filtering Data	7.117
7.5.6 *SFDCOMP Data Group - Composite Element Stress Calculation and Filtering Data	7.119
7.5.7 *TEMPHISTORY Data Group - Temperature History Output for Transient Heat Transfer.....	7.121
7.5.8 *TEMPOUT Data Group - Time Steps for Transient Heat Transfer Output	7.122
7.5.9 *HISTOUT -History Output Request Data Group.....	7.123
7.5.10 *I5 Data Group - Alternate Form for Printout Control	7.125
7.5.11 *N5 Data Group - Alternate Form for Nodal Stress Printout Control	7.127
7.5.12 *POSTCNTL Data Group - Selective saving of element output of quantities on the post-data file for Post Processing	7.128
7.6 Data Terminator	7.132
7.6.1 *ENDDATA Data Group - Input Data Terminator ...	7.132

8 Modal Dynamic Analysis Data

8.1

8.1 Introduction	8.1
8.2 Classification of Data by Analysis Types	8.7
8.2.1 Overview	8.7
8.2.2 Transient Dynamic Analysis (TRANSIENT)	8.7
8.2.3 Random Vibration Analysis (RANDOM)	8.10
8.2.4 Frequency Response Analysis (FREQUENCY)	8.11
8.2.5 Shock Spectrum Analysis (SHOCK)	8.13
8.2.6 Frequency Point Selection (for frequency response and random vibration analyses)	8.16

8.3	Executive Commands	8.18
8.3.1	General Description	8.18
8.3.2	Executive Commands Applicable to All Modal Dynamic Analyses	8.18
8.3.3	Executive Commands Applicable to Specific Modal Dynamic Analysis Types	8.22
8.4	Title	8.31
8.4.1	*TITLE Data Group - Problem Title	8.31
8.5	Analysis Control Data	8.32
8.5.1	*MODESELECTION Data Group - Mode Selection Data	8.32
8.5.2	*DAMPING Data Group - Damping Values.....	8.34
8.5.3	*TSTEP Data Group - Time Step Data	8.36
8.5.4	*ADDFREQUENCY Data Group - Additional Exciting Frequencies	8.37
8.5.5	*INITIAL Data Group - Initial Condition Data.....	8.38
8.5.6	*MATDAMPING Data Group - Material Damping Data.....	8.40
8.5.7	*FLOORRESPONSE Data Group – Floor Response Spectra Data	8.42
8.6	Load Function Description Data	8.44
8.6.1	*ARRIVALTIME Data Group - Arrival Time Data.....	8.44
8.6.2	*TIMEFUNCTION Data Group - Time Function Definition	8.45
8.6.3	*PSDFUNCTION Data Group - Power Spectral Density (PSD) Function Definition	8.48
8.6.4	*CORRELATION Data Group - Correlation Ensemble Data	8.52
8.6.5	*SPECTRUM Data Group - Spectrum Definition Data	8.56
8.6.6	*MDSPECTRUM Data Group - Multiple Damping Spectrum Data.....	8.59
8.6.7	*NONSTATIONARY Data Group - PSD and the Corresponding Envelope Function Definition	8.61
8.6.8	*ENVELOPE Data Group - Envelope Function Definition.....	8.62
8.6.9	*PRSPECTRA data group -Prestored Response Spectra definition data	8.66
8.7	Load Specification Data	8.77
8.7.1	Load Specification Parameters	8.77

8.7.2	*GROUND Data Group - Ground Motion Data	8.80
8.7.3	*DCFORCE Data Group - Concentrated Nodal Force Data.....	8.83
8.7.4	*DPRESSURE Data Group - Pressure Loading Data..	8.85
8.7.5	*DRIVER Data Group - Driver Node Data.....	8.87
8.7.6	*MVELOAD Data Group - Moving Load Definition	8.89
8.7.7	*MVEPATH Data Group - Moving Load Path Definition.....	8.92
8.7.8	*MSEXCITATION data group – Multiple support excitation data group	8.94
8.8	Output Control Data	8.97
8.8.1	*RSET Data Group - Response Set Data	8.97
8.8.2	*HISTORY Data Group - Response Time History Request Data.....	8.102
8.8.3	*SNAPSHOT Data Group - Snap Shot.....	8.104
8.8.4	*PSDOUT Data Group - Response PSD Request Data.....	8.106
8.8.5	*RMSOUT Data Group - Root Mean Square of Response Request Data	8.109
8.8.6	*SPOUT Data Group - Output Spectra Request Data	8.110
8.8.7	*RESPONSE Data Group - Maximum Response Request Data.....	8.112
8.8.8	*FRSOUT Data Group – Floor Response Spectra Output Request Data Group	8.114
8.9	Data Deck Terminator	8.118
8.9.1	*ENDDATA Data Group - Input Data Deck Terminator.....	8.118

A Modeling Hints ***A.1***

B NISA Static Analysis Example ***B.1***

C Quick Reference List ***C.1***

C.1 Alphabetical Quick Reference List of all Executive Commands .	C.1
C.2 Alphabetical Quick Reference List of All Data Groups	C.8

D User's Subroutines ***D.1***

D.1 Introduction	D.1
D.2 User's Subroutine USRCP	D.2
D.3 User's Subroutine USRPLS	D.4
D.3.1 Introduction	D.4
D.3.2 Modeling Tips	D.10
D.4 User's Subroutine USRFAL	D.12
D.4.1 Introduction	D.12

E References ***E.1***

I.1 Index ***I.1.1***

Preface

You have with you the latest version of NISA 19.0; a proven Finite Element Analysis software that has enjoyed a long-standing presence in the arena of Engineering Analysis and Design. Generations of scientists, engineers and researchers have come to depend on NISA to solve their most complex engineering problems.

With the acquisition of NISA by Cranes Software, Inc. in 2005 and subsequent investment in fresh talent and resources had elevated NISA[®] into one of the market leaders. Many past users are delighted with NISA's new capabilities and have come back to the software they loved and were at ease with.

With Version 19.0, we have carried forward the significant improvements that were initiated with Version 16.0 with respect to interoperability and performance. In addition to enhancements, the CAD translators; Sat2Neu, CATIA[™] V4/V5, Pro/E[™], Inventor[™], UG[™], STEP, Adv. IGES[™] to DISPLAY Neutral are upgraded with ACIS R19. With features such as Advanced Tetrahedral Mesher, meshing individual Trimmed Parametric Surface (TPS) data and Applying Boundary Conditions on TPS the meshing capabilities are enhanced. With the introduction of Shifted block-Lanczos eigen solver and Lagrangian algorithm for surface to surface contact element, the performance of NISA II[®] is vastly improved for structural problems. A new tree menu structure and many more user friendly navigating facilities have been introduced in DISPLAY IV[®]. Parametric programming using MACRO has been further extended.

Significant updates have been made to individual modules of NISA Special care has been taken to maintain the quality of the manuals which the industry has grown to appreciate. New additions to the various modules have been consolidated and presented in an integrated manner. We hope the Version 19 experience is a great one for you. Your feedback on the software and documentation is very valuable to us; please let us know how we can improve to serve you better. Please send your feedback and technical support questions to support@nisasoftware.com.

Guidelines for Using this Manual

NISA is a large scale general purpose finite element program with diversified capabilities for the analysis of a wide range of engineering applications. It is important to note, however, that the user is not expected to read the entire manual before using NISA for solving problems. At the same time, it is assumed throughout this manual that the user has a basic knowledge of engineering mechanics and the finite element method.

Guidelines for effective use of this manual follows, as well as references to other pertinent NISA documentations and support.

□ **How to Access Information in this Manual**

Various cross-references, listings, quick reference tables and figures are provided throughout this manual. To aid the user in accessing a particular topic or an input item, the following global tables maybe consulted:

- [Table 4.1](#) to [Table 4.5](#) summarize element types, applicability, features and required element input data.
- [Table 5.3](#) and [Table 5.4](#) provide an alphabetical listing of all executive commands and data group identification names.
- [Table 3.3](#) to [Table 3.5](#) provide descriptions of output quantities and how to request them.

□ **Getting Familiar with NISA Capabilities**

It is recommended that the user start the manual by reading Introduction [Chapter 1](#) and Theoretical Overview [Chapter 2](#), summarizes NISA capabilities in an easy-to-read point format. A detailed description of the capabilities is given in [Chapter 3](#). To get acquainted with the element library, the user should review [Section 4.1](#) before proceeding to the detailed element description. Quick element reference guides are provided in the description of each element.

Recommended modeling practices, given in [Appendix A](#), should be considered in preparing the model. Users interested in theoretical aspects should review [Chapter 2](#). It should be noted that the capabilities not applicable to the particular problem being solved may be entirely skipped.

□ **NISA Input Data Structure and Preprocessing**

It should please the user to know that the entire NISA data deck (with few exceptions) may be generated interactively using DISPLAY program. DISPLAY is a 3-D color graphics program with extensive modeling capabilities, CAD/CAM interface (directly or through IGES format), and both command and menu driven modes with on-line help.

The user should review [Chapter 5](#) to learn about free format rules and the input data setup for various analysis types.

Finally, [Chapter 6](#) through [Chapter 8](#) should be used as a reference to provide detailed description of each data group. The data groups are arranged by function and presented in a sequence that closely follows the procedure of finite element modeling.

□ **Output Request and Postprocessing**

[Table 3.3](#) to [Table 3.5](#) should be consulted for quick reference of output quantities from each analysis type and how to obtain them. A sample static analysis problem is given in [Appendix B](#) to aid beginning users in understanding input and output specifications.

It is recommended to use the DISPLAY program to obtain graphical representation of the results. [Section 3.14](#) should be reviewed for postprocessing procedure and features.

□ **Related Documentations and NISA Seminars**

In addition to this user manual, provides other documentations as listed in [Section 1.2](#). Among these, the NISA training manual provides detailed description of model generation, analysis and postprocessing for sample problems using DISPLAY and NISA programs. The NISA verification problems

manual contains a large number of problems covering a wide range of program capabilities and may be used as a source of sample problems.

Also, offers periodic introductory and advanced NISA training seminars. Contact for details and availability.

Introduction

1.1 General Description

NISA (Numerically Integrated elements for System Analysis) is a general purpose finite element program developed and maintained to analyze a wide spectrum of problems encountered in engineering mechanics. A brief overview of NISA capabilities is given below. Detailed description of the capabilities is given in [Chapter 3](#). Highlights of the main features of the pre- and post-processor DISPLAY program are also included. Refer to *DISPLAY III User's Manual* for details.

□ Preprocessing

NISA directly interfaces with the preprocessing module of the DISPLAY program; a 3-D interactive color graphic program with extensive modeling capabilities for finite element model generation and problem definition. Highlights of the capabilities are:

- CAD/CAM interface, directly from geometry data base or through IGES format.
- Both command and menu driven modes, with on-line help.
- 3-D geometric modeling including points, lines, arcs, curves, surfaces and solids as well as surface intersections.
- Geometric transformations including translation, rotation, scaling, mirror imaging and dragging a curve along an arbitrary 3-D path.
- 3-D interactive finite element mesh generation including automatic node and element generation.
- Mesh grading with uniform or non-uniform spacing.

Introduction

General Description

- Merging separate models into a larger one.
 - Definition of element attributes including material and geometric properties.
 - Specification of loading and boundary conditions.
 - Extensive model editing capabilities.
 - Extensive plotting options including boundary line, hidden line removal and shrink element plots for selected elements or regions.
 - Color shading and light effects.
 - Model checking including calculation of element areas, volumes, normals, and distortion index.
 - Complete NISA data deck generation.
- **Input Data Structure**
- Simple modular input data structure and easy to use free format.
 - Descriptive data group identification names (headers) reflecting the function of each data group. For example, material properties are entered in *MATERIAL data group.
 - The data deck consists of three data blocks: The executive commands specifying the overall control parameters in simple alphanumeric format, the model data block describing the model characteristics and finally the analysis data block specifying the loads, boundary conditions and the output options.
 - Data groups (modules) may appear in any order within each data block, with very few exceptions.
 - Annotated echo of the input data.
 - Extensive data checking with self-explanatory diagnostic messages.
 - Ease of modeling for different analysis types: The input data used in one analysis type can be used for another analysis type with minor modification.
- **Element Library**
- Linear and higher order isoparametric elements.
 - 2-D and 3-D point mass, spring and spar elements.
 - 2-D and 3-D prismatic and tapered beam elements.
 - 2-D plane stress, plane strain and axisymmetric solid elements.

- 3-D thick shell, solid and hybrid solid elements.
 - 3-D general shell, thin shell and axisymmetric shell elements.
 - 3-D laminated composite shell, sandwich shell and laminated composite solid elements.
 - 2-D axisymmetric and 3-D gap elements, tension-only and compression-only members.
 - Axisymmetric shell and solid elements with non-axisymmetric loading.
 - 2-D planar elements, axisymmetric and 3-D solid elements, 3-D shell and 3-D bar elements for heat transfer.
- ☐ Material Models
- Linear elastic material models (isotropic, orthotropic and laminated layered composite).
 - Elastoplastic material models with various yield criteria and flow rules.
 - Hyperelastic (rubber-like) material models.
 - Heat transfer material models (isotropic and orthotropic).
 - Temperature dependent material properties.
 - Creep material models.
- ☐ Kinematic Constraints
- Coupled displacements, coupled temperatures.
 - Rigid links.
 - General linear, nonhomogenous multi-point constraint (MPC) equations.
 - Automatic resequencing of MPC equations (including rigid link equations).
 - Specified displacement boundary conditions may be changed with loading conditions.
- ☐ Analysis Types and Loading
- Loads may also include non-zero support displacements. Relative response is obtained.
 - Linear Static Analysis:

Loads may consist of concentrated nodal forces, non-zero specified displacement, thermal loading and body forces due to gravity, angular velocity and angular acceleration. Multiple load cases with different boundary conditions may be analyzed. Results may be combined using load combination.

- **Nonlinear Static Analysis:**

Geometric and material nonlinearities including large displacements and rotations and finite strains as well as elastoplastic, creep, and hyperelastic material models can be analyzed. Both total and updated Lagrangian formulations are available. Loads include follower (deformation dependent) nodal forces and pressure in addition to all loads available for linear static analysis. Automatic load stepping is available.

- **Linear Direct Transient Analysis:**

Linear direct transient analysis may be performed with loads including prescribed forces, gravity, Coriolis and centrifugal forces. Kinematic conditions may include specified and coupled displacements, multi-point constraints and rigid links. Proportional damping and discrete damper may be used.

- **Eigen Value Analysis:**

Natural frequencies and mode shapes are obtained, including rigid body mode computation and Sturm sequence check. Consistent or lumped mass formulation may be used.

- **Transient Dynamic Analysis:**

Loads may be in the form of time dependent concentrated nodal forces, pressure, ground motion and multiple support excitation. Modal viscous or proportional viscous damping may be used. Absolute or relative response may be obtained.

- **Direct Frequency Response Analysis:**

Loading may be in the form of harmonic concentrated nodal forces and pressure with frequency dependent amplitudes and phase angles. Proportional damping and discrete damper may be used.

- **Random Vibration Analysis:**

Loading may be in the form of fully or partially correlated stationary concentrated nodal forces, pressure, ground motion and multiple support excitation defined in terms of their auto-and cross-spectral density functions (PSDs). Acoustic load input may be specified in terms of percentage octave band

decibel levels. Modal viscous, proportional viscous or modal structural damping may be used. Absolute or relative response may be obtained.

- Frequency Response Analysis:

Loading may be in the form of harmonic concentrated nodal forces, pressure, ground motion and multiple support excitation with frequency dependent amplitudes and phase angles. Damping may be specified as modal viscous, proportional viscous or modal structural. A procedure for point-to-point transfer function evaluation is available. Absolute or relative response may be obtained.

- Shock Spectrum Analysis:

Input may be in the form of displacement, velocity and acceleration spectra. The spectra may be specified independently for each direction and for various damping values. Analysis can be performed for either uniform or non-uniform excitation at the supports. Response can be calculated using the different combination rules available in the literature. Response may also be combined across directions using absolute or RMS combination rules. A correction for the missing mass of the neglected modes can be made by specifying the cutoff frequency.

- Buckling Analysis:

Bifurcation buckling is determined in terms of load factors and mode shapes.

- Steady State and Transient Heat Transfer Analysis:

Linear and nonlinear analysis may be performed. Material properties may be temperature and/or time dependent. Loading and boundary conditions include concentrated and distributed flux, internal heat generation, convection and radiation boundary conditions and specified temperature. Loading may be temperature and/or time dependent. Solidification and melting processes (phase change) may be analyzed.

□ Solution Techniques

- Built-in element resequencing algorithm for wavefront minimization. All input specifications and output requests are in terms of the user's element identification numbers.
- Efficient wavefront solver with relatively small central memory and disk space requirement.

- Conventional and accelerated subspace iteration, inverse iteration, and Lanczos methods for eigen value extraction.
- Conventional and modified Newton-Raphson methods and pure incremental procedure for nonlinear static analysis.
- Conventional and modified Newton-Raphson methods for nonlinear heat transfer in conjunction with four common integration techniques for transient analysis: forward difference, Crank-Nicholson, Galerkin and backward difference schemes.
- Closed form solution for modal equations in transient dynamics; exact for piecewise linear forcing functions.
- Numerical integration using Simpson and trapezoidal rules in random vibration analysis in addition to closed form solution for white noise.
- Parallel Direct Sparse of INTEL MKL library.
- Conventional and accelerated subspace iteration, inverse iteration, Lanczos and shifted block Lanczos methods for eigen value extraction.

□ Output Features

Various output options are offered to meet individual requirement, with comprehensive output control for all analysis types. Highlights of printed output for various analyses are given below.

Static, Nonlinear Static, Buckling, Eigenvalue, Direct Transient Dynamic and Direct Frequency Response Analyses

- Displacements in global or local directions and largest magnitudes of the nodal displacement vector.
- Reactions and summation of reactions in global directions
- Element internal forces and strain energy.
- Element stresses at centroid, Gauss and nodal points as well as principal stresses and stress intensities associated with various theories of failure.
- Element stress resultants for springs, spars and beams.
- Averaged nodal stresses, nodal principal stresses and stress intensities (von Mises equivalent stress, maximum shear and octahedral shear stresses).
- Various options for stress filtering and stress sorting.

- Second Piola-Kirchhoff or Cauchy stress measures for nonlinear static analysis.
- Natural frequencies/load factors and mode shapes for eigenvalue/buckling analysis.
- Amplitudes and phase angles in direct frequency response analysis.
- Assembled global stiffness matrix in case of static analysis and stiffness, damping and mass matrices in case of direct frequency analysis

Heat Transfer Analysis

- Nodal temperatures (and nodal temperature differences for shell elements as applicable).
- Nodal point heat flow at the nodes with specified temperatures.
- Heat flow on element boundary faces with convection and radiation boundary conditions.

Modal Dynamic Analysis

Displacements, velocities, accelerations, reactions, beam-end forces, base shear and stresses are provided in the form of:

- Response time history or snapshot (freeze), floor response spectra in transient dynamic analysis.
- RMS values, response PSD, cross PSD and extreme value characteristics in addition to modal covariance matrices in random vibration analysis.
- Amplitudes and phase angles in frequency response analysis.
- Maximum response using various modal and directional combination rules in shock spectrum analysis.

Postprocessing

Graphical representation of the results may be obtained interactively using the post-processing module of the 3-D color graphics DISPLAY program following a successful NISA run. A brief account of the postprocessing features are given below.

- Various geometry plotting options including hidden line removal, boundary and feature line plots and view manipulation including rotation, scaling and zooming.
- Deformed geometry plots, separate or superimposed on undeformed geometry.
- Animated deformed shapes (on certain graphic devices).

Introduction

General Description

- Deformed history plots for nonlinear static analysis.
- Contour plots for displacements, mode shapes, stresses and temperatures.
- Contour plots for cut sections of 3-D models.
- XY history plots for various output quantities.
- Unaveraged element stress contours including error estimation.

1.2 NISA Suite of FEA Software

1.2.1 NISA Software

NISA provides an integrated and comprehensive family of general and special purpose programs for Computer Aided Engineering (CAE). The NISA suite of FEA software covers a wide spectrum of engineering applications, e.g., linear and nonlinear structural and heat transfer analysis, structural and shape optimization, fatigue and fracture analysis, fluid flow analysis and printed circuit board stress and heat transfer analysis. All analysis programs are directly interfaced with the DISPLAY program for pre- and post-processing. A brief description of the NISA suite of FEA software is given below. Complete description of these programs is available in their pertinent user's manuals listed in [Section 1.2.2](#).

□ NISA II

This is the parent program of the NISA family. It is a general purpose finite element analysis program to which all other programs in the NISA family are interfaced. NISA capabilities and input specification are given in this manual. Other NISA related documentations are listed in [Section 1.2.2](#).

□ DISPLAY

A 3-D interactive color graphics finite element pre- and post-processing program. Highlights of the main capabilities are given in *Display III/IV User's Manual*.

□ COMSYN

A free interface component mode synthesis program to compute natural frequencies and mode shapes of a structure from NISA eigenvalue analyses of its components. As a special feature, this program can also compute natural frequencies and mode shapes of cyclic symmetric structures, given the results of NISA eigenvalue analysis of a single sector.

□ STROPT

A general purpose finite element structural optimization program employing the state-of-the-art optimization techniques including design sensitivity computation. The optimization process involves appropriately re-sizing the cross-sectional parameters (design variables) while simultaneously satisfying prescribed limits (constraints) on the structural response to minimize material volume, mass or

weight. Constraints include displacements, stresses and natural frequencies in addition to limits on the design variables.

The design variables may be fixed (constant valued) or free (variable valued), and they may be grouped and linked in many possible ways. This facility is especially useful for uniformity of design in certain regions of the structure, for creating a symmetrical design under unsymmetric loading conditions, and for satisfying certain design specifications. Optimization may be performed for multiple loading conditions.

□ **SHAPE**

A finite element program for shape optimum design of structures. SHAPE adopts a novel approach for the shape optimization of 2-D and 3-D continuum structures. The novelty lies in that the design variation is not limited to the original boundaries. Instead, new boundaries may be created by removal of material from inside the structure. There is no need to model the boundaries by parametric curves. In addition, symmetry conditions may be fully exploited in modeling, and major design changes without the need for mesh refinement can be accomplished. Shapes can be derived for new applications by starting from a very general model.

Various displacement and stress constraints may be specified for optimization under multiple loading conditions. Furthermore, parts of the original design may be frozen by the user in order to satisfy specific manufacturing requirements.

SHAPE uses NISA as the analysis module and accepts the same NISA input file. Specific optimization information, e.g., constraints, number of iterations, frozen regions, are read from a separate file. For each new design, a new NISA input file and an updated optimization input file are generated. In addition, SHAPE also provides a NISA input file with smoothed boundaries. Furthermore, a separate file for design history is output, to enable tracing of the changes in design. Each improvement in the design may be plotted and postprocessed in DISPLAY.

□ **SECOPT**

A program for optimizing the area of beam cross-section under a set of constraints on section properties as well as other constraints imposed by code requirements. Additional constraints may be imposed to control local buckling of webs or flanges and to satisfy other practical considerations. The section properties of the optimum section may be written to a file in NISA input format.

□ ENDURE

A program for analyzing the fatigue performance of engineering structures. Durability studies including crack initiation as well as crack propagation can be performed. Both stress-life and strain-life approaches are available for crack initiation studies. Damage tolerance analysis can be carried out conveniently with the availability of built-in crack configurations and crack growth models. Crack growth models include: Paris, Foreman, Collipriest, Walker and Elber. Fracture parameters include stress intensity factor and J-integral.

Single and multi-channel loading may be used. Both deterministic and probabilistic description of loading are available. The rainflow method of cycle-counting is used for the general case of sequential load history. Stress input may be specified manually during preliminary design or directly interfaced from NISA static or dynamic analysis.

Fatigue life contours may be plotted using the postprocessing module of the DISPLAY program.

□ FEAP

A program for the analysis of Printed Circuit Boards (PCBs) and electronic systems. FEAP is completely interfaced with NISA for various types of analysis including stress and thermal analysis, heat transfer with convective heat flow and random vibration analysis.

The finite element model of the PCB is created interactively using a large built-in library of commercial components such as: Dual Inline Packages (DIPs), hybrid packages, flat packs, Leadless Ceramic Clip Carriers (LCCCs), etc. Direct interface with several CAD geometry data bases is also available.

The Integrated Circuits (ICs) are modeled according to the method by which they are mounted on the PCBs, either the traditional poke-through attachment or the more recent surface-mounted technology. Once all components on a PCB are defined, a finite element mesh is automatically generated for each IC and the entire board. Appropriate elements, including isotropic or composite elements, are chosen from the NISA element library.

Other capabilities include: Analysis of several PCBs or an entire Avionics box, special sub-modeling and automated mesh refinement for an area of interest, detailed solid model of end pins in addition to prediction of reliability and fatigue life of any PCB component.

□ NISA-3D/FLUID

A finite element program for analyzing a wide range of 2-D, axisymmetric and 3-D fluid flow problems. Analysis capabilities include steady state and transient, laminar and turbulent, incompressible and compressible and Newtonian and Non-Newtonian fluid flow. In addition, conductive and convective heat transfer problems may be analyzed.

NISA-3D/FLUID interfaces with DISPLAY for graphical representation of results, e.g., color contour plots of pressure, velocity, temperature, strain rates, in addition to stream lines.

□ DYMES

A general purpose program for mechanical system dynamics to analyze constrained, multi-body and spatial mechanical systems, in which body elements are connected through mechanical joints such as spherical, revolute and translational joints and force elements such as nonlinear springs, dampers and actuators.

Major areas of applications of the code include machine mechanisms, robotics, vehicle dynamics, controlled systems and space craft dynamics.

□ NISA/CIVIL

A finite element program that offers CAD based solutions to a wide variety of problems encountered in the Analysis and Design of Reinforced Concrete and Steel Structures. Backed by powerful NISA II Analysis and DISPLAY III/IV - the graphical Pre and Post processor of NISA family of programs, NISA/CIVIL provides excellent tools for modeling, associating design information and carry out design process in Limit state and working stress methodologies of design. Design results are processed to produce structural engineering drawings in AutoCAD environment. Equipped with an extremely user friendly GUI and graphic displays, NISA/CIVIL, presents an elegant platform for analysis and design of different types of structures encountered in practice.

□ NISA/EMAG

A general purpose finite element program used to analyze a wide spectrum of problems in electrical engineering. In particular NISA/EMAG can be used to perform 2D and 3D electric and magnetic field analysis in electromagnetic devices.

□ NISA/HEAT

A finite element program for analyzing a wide range of 2D, axisymmetric and 3D thermal problems. Analysis capabilities include steady and transient analysis, 2D, 3D and axisymmetric domains, phase change, orthotropic thermal conductivity, surface radiation with view factor computation, volumetric/nodal heat generation. In addition, conductive and convective heat transfer problems may be analyzed. Also, heat transfer in laminated composite shells can be modeled.

NISA-3D/Heat interfaces with DISPLAY for graphical representation of results, e.g., color contour temperature plots.

□ NISA/DYSPAN

Modern structural systems have demands of stringent performance and integrity under various loading conditions. This necessitates an accurate analysis and design. This in turn, demands precise modeling of structures and forces acting on them. Many sophisticated tools are available for design engineers. This is particularly true for random forces such as earthquake excitation, wind forces and ocean waves. To help the design and analysis engineers for properly modeling the dynamical forces particularly the stochastic forces, a new module called Dynamic Spectral Analyzer (DYSPAN) has been developed.

DYSPAN is a general purpose program for characterization of dynamic loading/forces. This program is particularly useful for modeling of stochastic excitations, such as earthquake forces, wind loads and ocean waves.

□ NISA/AERO

A program for aeroelastic analysis. In the present version of NISA/AERO, divergence, flutter and aeroelastic response solution techniques (for both lifting surfaces and panels) at subsonic and supersonic speeds are available.

1.2.2 NISA Documentations

The NISA family of programs are documented in the following manuals:

User's and Verification Manuals

1. NISA II User's Manual
2. NISA II Verification Manual
3. DISPLAY III User's Manual
4. DISPLAY IV User's Manual
5. NISA/HEAT User's Manual
6. NISA/3D-FLUID (Incompressible) User's Manual
7. NISA/3D-FLUID (Compressible) User's Manual
8. NISA/EMAG User's Manual
9. NISA/OPT (Sectional Optimization) User's Manual
10. NISA/OPT (Shape Optimization) User's Manual
11. NISA/OPT (Structural Optimization) User's Manual
12. FEAP User's Manual
13. ENDURE User's Manual
14. ENDURE Examples & Verification Manual
15. NISA/ROTOR User's Manual
16. NISA/ROTOR Verification Manual
17. NISA/DYSPAN User's Manual
18. DYMES User's Manual
19. NISA/AERO User's Manual
20. NISA/CIVIL User's Manual
21. NISA/CIVIL Examples & Verification Manual

Training Manuals

1. DISPLAY III Training Manual
2. DISPLAY IV Training Manual
3. NISA/HEAT Training Manual
4. NISA/3D-FLUID Training Manual
5. NISA/EMAG Training Manual
6. FEAP Training Manual
7. NISA/ROTOR Training Manual
8. ENDURE Training Manual
9. DISPLAY IV Plugin

Theoretical Overview

2.1 Introduction

This chapter gives an overview of the theoretical basis for each analysis type as well as the basic element relations. For extensive description of the finite element method, the interested reader should consult standard texts on the finite element method (e.g., [2.5, 2.19, 2.52]).

Briefly, the finite element method is a powerful numerical technique for approximate solution of continuum mechanics problems, and can be regarded as an extension of the Rayleigh-Ritz procedure, since piecewise continuous trial functions are used. The method is approximate, since a continuum with an infinite number of degrees of freedom is replaced with a discrete system with a finite number of degrees of freedom. The method involves subdividing a continuum into a finite number of regions called elements connected at a finite number of points called nodes to which loads are applied. An approximate admissible solution is constructed over the assemblage of elements, and the solution continuity is maintained at the inter-element boundaries.

The NISA analysis types are discussed beginning with linear static analysis in [Section 2.2](#). General description of nonlinear analysis including basic equations and formulation methods is given in [Section 2.3](#). Dynamic analyses including eigenvalue, normal mode method and direct time integration are described in [Section 2.4](#). The normal mode method includes transient dynamic, frequency response, random vibration and shock spectrum analyses. [Section 2.5](#) deals with the Buckling Analysis. [Section 2.6](#) describes the basic equations and boundary conditions for heat transfer analysis. Material models including ideal elastic, elastoplastic and hyperelastic are described in [Section 2.7](#). Special numerical

treatments associated with a particular material model are also discussed in the same section.

The element characteristics are discussed in [Section 2.8](#). Unless noted to the contrary, the elements in NISA are isoparametric, which means that the same shape functions are used to represent both the element geometry and the displacement (temperature) variation within an element. To aid in understanding of the element development, a complete derivation is given for 2-D and 3-D solid elements in [Section 2.8](#).

Notation

Matrix notations are used throughout. An upper case bold face letter denotes a matrix, e.g., \mathbf{K} . A lower case bold face letter denotes a vector (or column matrix), e.g., \mathbf{u} . Superscripts T and -1 denote matrix transposition and inverse, respectively, e.g. \mathbf{K}^T and \mathbf{K}^{-1} . Variables are defined when they first appear. Right subscripts may refer to nodal points, coordinate direction or components of the quantity. Left superscripts denote the configuration at which the quantity is measured whereas left subscripts indicate the configuration to which the quantity is referred.

Whenever convenient, the indicial notation or symbolic tensor notations of continuum mechanics are sometimes used. Unless indicated to the contrary, repeated indices imply summation over the range, e.g.,

$$\sigma_{ij}^{\varepsilon_{ij}} = \sum_{i=1}^3 \sum_{j=1}^3 \sigma_{ij}^{\varepsilon_{ij}}$$

The quantities needed to present the theory in this chapter are scalars, vectors, second order tensors, matrices and, occasionally, fourth order tensors, (e.g., constitutive stress-strain relations and their transformations). These quantities may be expressed in direct matrix notation or in component form as follows:

- scalars a
- vector \mathbf{a} or \mathbf{a}_i or $\mathbf{a}_i \mathbf{e}_i$ or $\{\mathbf{a}\}$

second order tensors	\mathbf{A} or \mathbf{A}_{ij} or $\mathbf{A}_{ij}\mathbf{e}_i\mathbf{e}_j$ or $[\mathbf{A}]$
matrix	\mathbf{A} or \mathbf{A}_{ij} or $[\mathbf{A}]$
fourth order tensors	\mathbf{A} or \mathbf{A}_{ijkl} or $\mathbf{A}_{ijkl}\mathbf{e}_i\mathbf{e}_j\mathbf{e}_k\mathbf{e}_l$

The base vectors, \mathbf{e}_i are defined by,

$$\mathbf{e}_i = \frac{\partial \mathbf{x}}{\partial \xi_i} / \left\| \frac{\partial \mathbf{x}}{\partial \xi_i} \right\|$$

where \mathbf{x} is the position vector of a point in the coordinate system and ξ_i is the coordinate (i.e., ξ_i may take the value of $\mathbf{x}, \mathbf{y}, \mathbf{z}, \xi, \eta, \dots$) and ' $\| - \|$ ' denotes the Euclidean norm of the vector. In addition to the basic matrix operations, the following vector and tensor operations are used in this chapter:

Dot product of two vectors

$$\alpha = \mathbf{a} \cdot \mathbf{b} = a_i b_i \equiv \sum_{i=1}^3 a_i b_i$$

Cross product of two vectors

$$\mathbf{a} = \mathbf{b} \times \mathbf{c} = \det \begin{bmatrix} \mathbf{i} & \mathbf{j} & \mathbf{k} \\ b_x & b_y & b_z \\ c_x & c_y & c_z \end{bmatrix}$$

Dot product of two second order tensors

Theoretical Overview

Introduction

$$\mathbf{A} = \mathbf{B} \cdot \mathbf{C} \quad \text{or} \quad A_{ij} = B_{ik} C_{kj} \equiv \sum_{k=1}^5 B_{ik} C_{kj}$$

Double dot product of two second order tensors

$$\alpha = \mathbf{A} : \mathbf{B} \quad \text{or} \quad \alpha = A_{ij} B_{ij} \equiv \sum_{i=1}^3 \sum_{j=1}^3 A_{ij} B_{ij}$$

A vector derivative is given by,

$$\begin{aligned} \mathbf{a} &= \frac{\partial \mathbf{v}}{\partial t} \\ &\equiv \frac{\partial v_i}{\partial t} \mathbf{e}_i \quad \text{in Cartesian coordinate system, and} \end{aligned}$$

$$\begin{aligned} \mathbf{A} &= \frac{\partial \mathbf{v}}{\partial \mathbf{x}} \\ &\equiv \frac{\partial v_j}{\partial x_i} e_i e_j \quad \text{in Cartesian coordinate system} \end{aligned}$$

which means that,

$$d\mathbf{v} = \frac{\partial \mathbf{v}}{\partial \mathbf{x}} \cdot d\mathbf{x}$$

A second order tensor derivative is given by,

$$\begin{aligned} \mathbf{B} &= \frac{\partial \mathbf{A}}{\partial t} \\ &\equiv \frac{\partial A_{ij}}{\partial t} \mathbf{e}_i \mathbf{e}_j \quad \text{in Cartesian coordinate system, and} \end{aligned}$$

$$\mathbf{C} = \frac{\partial \mathbf{S}}{\partial \mathbf{E}}$$

$$\equiv \frac{\partial S_{kl}}{\partial E_{ij}} \mathbf{e}_i \mathbf{e}_j \mathbf{e}_k \mathbf{e}_l \text{ in Cartesian coordinate system}$$

The following list provides a nomenclature for the most commonly used quantities throughout the chapter:

- c : damping coefficient
- E : modulus of elasticity
- H : frequency response function
- I : invariant of a second order tensor
- N : shape or interpolation function
- R : cross correlation function
- S : surface area, power spectral density (PSD) function
- t : time
- T : kinetic energy, temperature
- U : strain energy of the body
- V : potential of applied loads, volume
- \mathbf{b} : body force vector
- \mathbf{B} : strain-displacement matrix
- \mathbf{C} : Cauchy-Green deformation tensor, damping matrix
- \mathbf{C}, \mathbf{D} : elasticity or constitutive matrix
- \mathbf{D} : rate of deformation tensor
- \mathbf{E} : Green-Lagrange strain tensor
- \mathbf{F} : deformation gradient tensor

- I : identity matrix
- K : stiffness matrix, conductivity matrix
- L : velocity gradient tensor, Influence Matrix
- M : mass matrix
- p : vector of applied forces
- q : vector of generalized or normal coordinates
- q, Q : heat flux vector
- r : position vector
- R : transformation matrix
- S : second Piola-Kirchhoff stress tensor
- t : surface traction vector
- T : transformation matrix
- u : displacement vector, nodal displacement vector
- \bar{u} : nodal displacement vector
- \dot{u} : velocity vector
- \ddot{u}, a : acceleration vector
- x : position vector
- γ, Γ : engineering shear strain, participation factor
- δ : first variation
- δ_{ij} : Kronecker delta
- Δ : increment
- ε : strain tensor, strain tensor arranged in a vector form, or Euler-Almansi strain tensor
- ν : Poisson's ratio

- ξ : damping ratio, isoparametric coordinate
 Π : total potential energy
 ρ : density
 σ : stress tensor
 φ, Φ : eigenvector, mode shape
 ω, Ω : angular velocity, natural frequency, driving frequency

The following provides a nomenclature for the most commonly used subscripts and superscripts:

- ∂ : derivative with respect to a coordinate direction
 $^{-1}$: inverse of a matrix or a second order tensor
 $1, 2, 3$: components of a vector, coordinate directions
 $1, 2, 3, \dots, n$: nodal numbers, mode shapes
 b : body forces, internal heat generation
 c : concentrated quantity, cosine component of a load, conductivity quantity
 d : damping
 e : element quantity, elastic quantity
 G : initial stress, geometric
 h : convection B.C. quantity
 i, j, k : components of a vector or tensor, elements of a matrix, dummy indices, model quantity
 $i, (i+1)$: iteration number
 k : nodal quantity
 lin : linear component or part of a matrix or tensor
 l, m, n, p, q : coordinate directions, dummy indices

nc : non conservative

nl : nonlinear component or part of a matrix or tensor

$0, t, t + \Delta t$: time (or configuration) at $\mathbf{t} = 0, \mathbf{t}, \mathbf{t} + \Delta \mathbf{t}$, respectively

p : plastic

r : modal quality, radiation

s : surface quantity, sine component of a load

t_0, t_f : time points, $\mathbf{t} = \mathbf{t}_0$, and $\mathbf{t} = \mathbf{t}_f$

tot : total value of a matrix or tensor

T : transpose of a matrix or a tensor

x, y, z : coordinate directions

$\dot{\bar{\epsilon}}$: strain rate dependent quantity

τ : time (or configuration) at $\mathbf{t} = \tau$

2.2 Linear Static Analysis

The total potential of an elastic body under general loading may be defined by [2.1],

$$\Pi = U + V \quad (2.1)$$

where, U is the strain energy of the body and V is the potential or the potential energy of the applied loads

The principle of minimum potential energy states that among all the admissible displacement fields \mathbf{u} which satisfies the prescribed kinematic boundary conditions, the actual displacement is the one that makes the total potential energy stationary. Therefore Equation (2.1) gives,

$$\delta\Pi = \delta U + \delta V = 0 \quad (2.2)$$

where,

$$\delta U = \int_V \boldsymbol{\sigma}^T \delta \boldsymbol{\varepsilon} dV \quad (2.3)$$

$$-\delta V = \int_V \delta \mathbf{u}^T \mathbf{b} dV + \int_A \delta \mathbf{u}^T \mathbf{t} dA + (\delta \bar{\mathbf{u}}^c)^T \mathbf{p}_c \quad (2.4)$$

where,

$\boldsymbol{\sigma}$ and $\boldsymbol{\varepsilon}$ list the components of the stress and strain tensors in a vector form,

$$\boldsymbol{\sigma}^T = [\sigma_{xx}, \sigma_{yy}, \sigma_{zz}, \sigma_{xy}, \sigma_{yz}, \sigma_{zx}] \quad (2.5)$$

$$\begin{aligned} \boldsymbol{\varepsilon}^T &= [\varepsilon_{xx}, \varepsilon_{yy}, \varepsilon_{zz}, 2\varepsilon_{xy}, 2\varepsilon_{yz}, 2\varepsilon_{zx}] \\ &= [\varepsilon_{xx}, \varepsilon_{yy}, \varepsilon_{zz}, \gamma_{xy}, \gamma_{yz}, \gamma_{zx}] \end{aligned} \quad (2.6)$$

\mathbf{b} and \mathbf{t} are the body force and surface traction vectors,

$$\mathbf{b}^T = [b_x, b_y, b_z] \quad (2.7)$$

$$\mathbf{t}^T = [t_x, t_y, t_z] \quad (2.8)$$

\mathbf{u} is the displacement vector given by,

$$\mathbf{u}^T = [u_x, u_y, u_z] \equiv [u_1, u_2, u_3] \quad (2.9)$$

and, finally $\bar{\mathbf{u}}^c$ is a vector listing the displacement components corresponding to the concentrated forces \mathbf{p}_c . Equation (2.2) through Equation (2.4) indicate that for an arbitrary virtual displacement $\delta\mathbf{u}$ satisfying the prescribed kinematic boundary conditions, the virtual work done by the internal forces is equal to the virtual work done by the external forces. This is exactly the principle of virtual work [2.1]. It is important to note that the principle of virtual work holds for any kinematically admissible virtual displacements and it is independent of the material stress-strain relationship.

In finite element analysis, a continuum is discretized by a number of suitable finite elements which are interconnected through nodal points on the boundaries of the elements. The total potential of the continuum, Π , may be considered as the sum of the individual element contributions Π^e , so that,

$$\Pi = \sum_{e=1}^m \Pi^e \quad (2.10)$$

where, m is the number of elements. The integrals in Equation (2.3) and Equation (2.4) are thus performed for each element individually and summed over all the elements.

For finite element formulations based on the principle of minimum potential energy, the displacements within the element \mathbf{u}^e , are assumed to be function of the element nodal displacements $\bar{\mathbf{u}}^k$,

$$\mathbf{u}_i^{(e)}(x, y, z) = \sum_{k=1}^n N_k(x, y, z) \bar{\mathbf{u}}_i^k \quad \text{or} \quad (2.11)$$

$$\mathbf{u}^{(e)} = N \bar{\mathbf{u}}^{(e)}$$

where, $N_k(x, y, z)$ is the displacement shape or interpolation function, and $\bar{\mathbf{u}}_i^k$ is the vector of nodal displacement components of a typical element,

$$\{\bar{\mathbf{u}}_i^k\}^T = [\bar{u}_i^1, \bar{u}_i^2, \bar{u}_i^3, \bar{u}_i^4, \dots, \bar{u}_i^n] \quad (2.12)$$

where, n is the number of nodes for a typical element.

Element strains $\boldsymbol{\varepsilon}^{(e)}(x, y, z)$ are now calculated by differentiating the appropriate displacement components in [Equation \(2.11\)](#),

$$\boldsymbol{\varepsilon}^{(e)}(x, y, z) = \mathbf{B}(x, y, z) \bar{\mathbf{u}}^{(e)} \quad (2.13)$$

where, \mathbf{B} is the appropriate strain-displacement matrix, and $\bar{\mathbf{u}}^{(e)}$ is the nodal displacement vector.

$$\{\bar{\mathbf{u}}^{(e)}\}^T = [\bar{u}_1^1, \bar{u}_2^1, \bar{u}_3^1, \bar{u}_1^2, \bar{u}_2^2, \bar{u}_3^2, \dots, \bar{u}_1^n, \bar{u}_2^n, \bar{u}_3^n] \quad (2.14)$$

For a linear elastic material, the stress-strain relations may be expressed in the form:

$$\boldsymbol{\sigma} = \mathbf{C}(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}^0) \quad (2.15)$$

where, \mathbf{C} is the elasticity or material matrix and $\boldsymbol{\varepsilon}^0$ is the initial strain or most commonly the thermal strain vector. Various forms of the material constitutive matrix \mathbf{C} for isotropic and orthotropic materials as well as forms for specific element types are given in [Section 2.7](#).

Substituting [Equation \(2.3\)](#) through [Equation \(2.43\)](#) into [Equation \(2.2\)](#) and considering arbitrary virtual displacements, we obtain the equilibrium equations of element assemblage in the form:

$$\begin{aligned} \mathbf{K}\mathbf{u} &= \mathbf{p} \\ &= \mathbf{p}_b + \mathbf{p}_s + \mathbf{p}_o + \mathbf{p}_c \end{aligned} \quad (2.16)$$

where, \mathbf{u} is the global nodal displacement vector (the bar is dropped for convenience, i.e. $\mathbf{u} \equiv \bar{\mathbf{u}}$), and the matrix \mathbf{K} is the global stiffness matrix of element assemblage.

$$\mathbf{K} = \sum_{e=1}^m \mathbf{K}^{(e)} = \sum_{e=1}^m \int_V \mathbf{B}^T \mathbf{C} \mathbf{B} dV \quad (2.17)$$

The total load vector \mathbf{p} includes the effect of the body forces, \mathbf{p}_b , the effect of the element surface tractions \mathbf{p}_s , the effect of the initial strains, \mathbf{p}_o , and finally the concentrated forces \mathbf{p}_c , where,

$$\mathbf{p}_b = \sum_{e=1}^m \int_V \mathbf{N}^T \mathbf{b} dV \quad (2.18)$$

$$\mathbf{p}_s = \sum_{e=1}^m \int_A \mathbf{N}^T \mathbf{t} dA \quad (2.19)$$

$$\mathbf{p}_o = \sum_{e=1}^m \int_V \mathbf{B}^T \mathbf{C} \boldsymbol{\varepsilon}^0 dV \quad (2.20)$$

Note that in [Equation \(2.17\)](#) through [Equation \(2.20\)](#), the summation sign is symbolic and should be interpreted as an assembly operator, for example,

$$\mathbf{K} = \sum_{e=1}^m \mathbf{J}_{(e)}^T \mathbf{K}^{(e)} \mathbf{J}_{(e)} \quad (2.21)$$

$$\mathbf{p}_b = \sum_{e=1}^m \mathbf{J}_{(e)}^T \mathbf{p}_b^{(e)} \quad (2.22)$$

where, $\mathbf{J}_{(e)}$ is a binary matrix that relates the element displacement vector to the global displacement vector of the entire system, i.e.,

$$\bar{\mathbf{u}}^{(e)} = \mathbf{J}_{(e)} \bar{\mathbf{u}} \quad (2.23)$$

2.2.1 Body Forces

Gravity loads and inertia loads (due to angular velocity and linear and angular acceleration) may be applied to any element which possesses a nonzero mass density. With reference to [Figure 2.1](#), the XYZ system is the rotating frame of reference which rotates with an angular velocity ω and angular acceleration α about the $X'Y'Z'$ system. In general, there is an offset between the global origin O and the reference point of rotation O' . At the instant shown, the two systems are parallel to each other. A linear acceleration field \mathbf{a}_L is also applied to the two systems.

The absolute acceleration of a generic point p can be written as [2.47],

$$\mathbf{a}_{\bar{p}} = \mathbf{a}_L + \ddot{\mathbf{u}} + \Lambda r + \Omega \Omega r + 2\Omega \dot{\mathbf{u}} \quad (2.24)$$

where,

$$\mathbf{r} = \mathbf{r}_{pO} - \mathbf{r}_{O'O} \quad (2.25)$$

and,

$$\Omega = \begin{bmatrix} 0 & -\omega_z & \omega_y \\ \omega_z & 0 & -\omega_x \\ -\omega_y & \omega_x & 0 \end{bmatrix}, \Lambda = \begin{bmatrix} 0 & -\alpha_z & \alpha_y \\ \alpha_z & 0 & -\alpha_x \\ -\alpha_y & \alpha_x & 0 \end{bmatrix} \quad (2.26)$$

\mathbf{a}_L = the absolute acceleration of the moving frame of reference $X'Y'Z'$ due to the linear acceleration field.

$\ddot{\mathbf{u}}$ = the stretching-term acceleration (i.e., relative) as viewed by an observer attached to the $X'Y'Z$ system.

$\Lambda \mathbf{r}$ = the tangential acceleration due to the angular acceleration α . For steady motion about a fixed axis, this term goes to zero.

$\Omega\Omega\mathbf{r}$ = the centripetal acceleration (due to the angular velocity of the moving axes)

$2\Omega\dot{\mathbf{u}}$ = the Coriolis acceleration (due to the interaction between the angular velocity and the stretching velocity).

For linear static analysis, the terms involving $\dot{\mathbf{u}}$ and $\ddot{\mathbf{u}}$ are not considered, and the generic point is considered on the undeformed configuration.

The body force vector to be used in [Equation \(2.18\)](#) is, by D'Alembert principle,

$$\mathbf{b} = -\rho \mathbf{a}_p \quad (2.27)$$

where, ρ is the mass density.

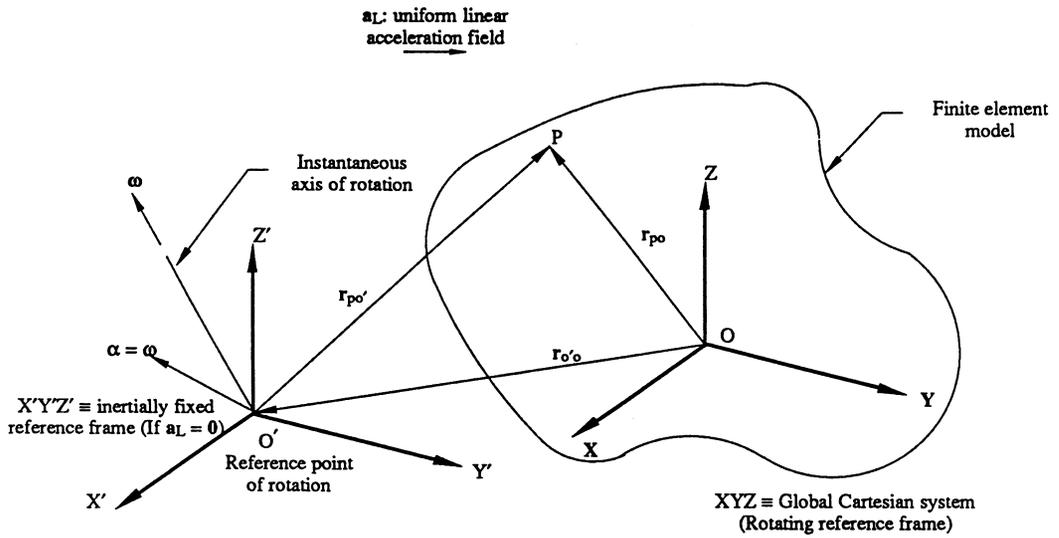


Figure 2.1: Acceleration at a point due to linear and angular motion

2.3 Nonlinear Static Analysis

2.3.1 Formulation Methods

In describing the motion of a body, there exist four methods of formulations, which Truesdell [2.2] called: the material, the referential, the spatial, and the relative formulation. All four methods of formulations are equivalent for any smooth motion of the body.

1. In the material description, the independent variables are the particle or the body-point \mathbf{P} , and the time t . This description is conceptually the most natural one and is the one exclusively used in analytical dynamics where the individual lumped masses are labeled in a discretized lumped mass system. It is rarely used, however, in continuum mechanics, or specifically in finite element applications.
2. In the referential description, the independent variables are the position \mathbf{x} of a particle \mathbf{P} in an arbitrarily chosen reference configuration (\mathcal{C}), and the time t . It is important to notice that the choice of the reference configuration is arbitrary and, essentially, this choice would not affect the results. A particular referential description was introduced

by Euler where the position ${}^o\mathbf{x}$ of the body-point \mathbf{P} at the particular time $t = 0$ is used to describe the motion. This particular description is often called the Lagrangian formulation in the literature. However, any other choice of the reference configuration at a specific time other than $t = 0$ would still be Lagrangian in nature, in the sense that the independent variable x is considered at a fixed time instant (See [Figure 2.2](#)).

The use of the Lagrangian description is particularly appealing because of the simplicity with which material rates of changes can be calculated. Another advantage of the Lagrangian description arises in the treatment of boundary conditions on the surface of the body which is changing during deformation. The boundary conditions on the deforming surface becomes nonlinear because the boundary surface depends on its unknown displacements. The use of a Lagrangian description which refers the motion to a fixed reference configuration enables one to treat most of the kinematic questions in a relatively simple manner. The same advantage actually arises for all integrated quantities since integration in the Lagrangian description will be taken over the original known configuration rather than the current unknown configuration. However, the use of the Lagrangian description also entails some disadvantages, which stem from the requirements that the stresses acting in the instantaneous current configuration have to be referred to the initial reference configuration in a way that is physically artificial though mathematically consistent. Special attention must, therefore, be given to the

development, and to the use of constitutive equations and Cauchy's axioms of motion. Another point that is difficult to handle in the Lagrangian description is the requirement for a continuous updating of boundary conditions as in contact problems and crack propagation problems. Excessive deformations of a body might cause a great deal of distortion in the original finite element mesh, and such distortions might greatly affect the accuracy and the reliability of the solution (See [Figure 2.2](#)).

3. In the spatial description, the independent variables are the current location ${}^t\mathbf{x}$ of the particle P , and the time t . This description was introduced by Daniel Bernoulli and D'Alembert but in the literature it is usually called the Eulerian description. In the spatial description, we are concerned with what is happening in a fixed region of space as a function of time, a description which seems to be perfectly suited for the study of fluids. However, for the same reason the spatial description is awkward for the expression of the principles of solid mechanics, since, in fact, the laws of continuum mechanics refer to what is happening to the body, and not to the region of space which the body momentarily occupies. In all such laws of mechanics, referring to the material of the body, the independent spatial variable ${}^t\mathbf{x}$ becomes a function of position ${}^0\mathbf{x}$ in the Lagrangian reference configuration and the time t (i.e. ${}^t\mathbf{x} = {}^t\mathbf{x}({}^0\mathbf{x}, t)$). Therefore, all material rates will be much more difficult to handle in the spatial description.

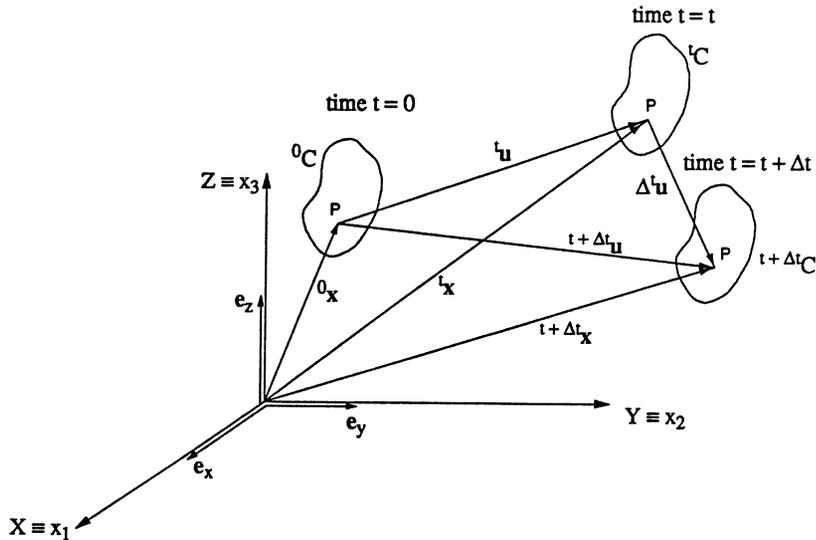


Figure 2.2: Geometry of motion

4. In the relative description, the independent variables are the position ${}^{\tau}\mathbf{x}$ in the current configuration and the variable time τ . The current configuration is taken as the reference configuration, and the variable time τ is the time relative to the present when the particle P occupied a place ξ , where $\xi = \xi({}^{\tau}\mathbf{x}, \tau)$. It is important to realize that the relative description is referential or Lagrangian in nature, in the sense that the reference position is now denoted by ${}^{\tau}\mathbf{x}$ at time $t = \tau$ rather than ${}^0\mathbf{x}$ at time $t = 0$. This justifies the classification of the updated Lagrangian formulation as a special case of the relative description. Owing to the nature of each formulation and its reference configuration, the relative description has some advantages over the referential description. As an example, the question of boundary condition updating and mesh distortion may be easier to handle in the updated Lagrangian description. It is also important to note that if a nonlinear discretized displacement field is assumed, the choice of an updated reference configuration may result in vanishing nonlinear contributions of the discretized field. On the other hand, the reference configuration is variable, more cumbersome to use, and may lead to some difficulties if analytical integration is used. However, numerical integration schemes, as in isoparametric finite element analysis, will avoid this difficulty. In the updated Lagrangian description, special attention should be given to the stress increment transformation and addition and also to the interpretation of constitutive equations and Cauchy's axioms of motion (See [Figure 2.2](#)).

In the finite element approach to continuum mechanics problems, only the last three descriptions are used [2.3-2.7]. Both the total and updated Lagrangian formulations are available in NISA.

2.3.2 Basic Equations

Comments on Notations

In all formulations to follow, we adopt the indicial tensor notations. All repeated indices are to be summed throughout their admissible range unless they are enclosed by parentheses. Left superscripts indicate the configuration at which the quantity is measured, whereas left subscripts indicate the relative configuration to which the quantity is referred. If the left subscript is omitted, it indicates that the quantity is referred to the original configuration (0C). Bold face upper case letters indicate matrices whereas bold face lower case letters indicate vector quantities or list of components. A matrix may or may not represent a tensor.

General Assumptions

A general body which occupies a finite region of Euclidean space is considered. When subjected to prescribed body forces and surface tractions, the body undergoes the motion ${}^t\mathbf{x} = {}^t\mathbf{x}({}^o\mathbf{x}, t)$. The particles of the body are identified by \mathbf{P} and the relation between the particle \mathbf{P} and the position vector \mathbf{x} does not change with time. Three configurations of the body are considered: configuration ${}^o\mathbf{C}$ at time $t = 0$, configuration ${}^t\mathbf{C}$ at time $t = t$ and configuration ${}^{t+\Delta t}\mathbf{C}$ at time $t = t + \Delta t$, see [Figure 2.2](#). For simplicity, without loss of generality, a fixed rectangular frame with Cartesian coordinate system in three-dimensional space is established to describe the motion. Only perfect mechanical systems which take no account of thermomechanical effects will be considered. Complete nonlinear kinematic relations within a linear increment will be used throughout the formulation. Further simplifying assumptions will be mentioned where introduced.

Deformation Gradients

Referring to [Figure 2.2](#), we may write the position vector \mathbf{x} as,

$${}^t\mathbf{x} = {}^o\mathbf{x} + {}^t\mathbf{u} \quad (2.28)$$

$${}^{t+\Delta t}\mathbf{x} = {}^o\mathbf{x} + {}^{t+\Delta t}\mathbf{u} = {}^t\mathbf{x} + {}^{t+\Delta t}{}_t\mathbf{u} \quad (2.29)$$

The deformation gradients may be expressed in the form,

$${}^t\mathbf{F} = ({}^o\nabla {}^t\mathbf{x}^T)^T = \mathbf{I} + ({}^o\nabla {}^t\mathbf{u}^T)^T \quad (2.30)$$

where,

$${}^o\nabla^T = \left\{ \frac{\partial}{\partial {}^o\mathbf{x}} \right\}^T = \left[\frac{\partial}{\partial {}^o x_1}, \frac{\partial}{\partial {}^o x_2}, \frac{\partial}{\partial {}^o x_3} \right] \quad (2.31)$$

and \mathbf{I} is the identity matrix of order 3. In a similar way, other deformation gradients may be defined as,

$${}^{t+\Delta t}{}_o\mathbf{F} = ({}^o\nabla^{t+\Delta t}\mathbf{x}^T)^T = \mathbf{I} + ({}^o\nabla^{t+\Delta t}\mathbf{u}^T)^T \quad (2.32)$$

Strain Tensors

In finite deformations, various kinds of finite-strain measures may be introduced to describe the motion of the body. The majority of these strain measures can be computed from the deformation gradients. In our particular case, however, we will be mainly dealing with the Green-Lagrange strain tensor \mathbf{E} , and the Eulerian or Almansi strain tensor $\boldsymbol{\varepsilon}$ which are defined from the relations:

$$(ds)^2 - (d\mathbf{S})^2 = 2(d^o\mathbf{x})^T {}^t\mathbf{E}(d^o\mathbf{x}) \quad (2.33)$$

$$(ds)^2 - (d\mathbf{S})^2 = 2(d^t\mathbf{x})^T {}^t\boldsymbol{\varepsilon}(d^t\mathbf{x}) \quad (2.34)$$

where, (ds) and $(d\mathbf{S})$ are the lengths of a material vector $d\mathbf{x}$ at time $t = t$ and $t = 0$, respectively, i.e., the deformed and the undeformed lengths. The above definitions yield the following expressions for ${}^t\mathbf{E}$ and ${}^t\boldsymbol{\varepsilon}$:

$${}^t\mathbf{E} = \frac{1}{2} \left[{}^o\nabla^t\mathbf{u}^T + ({}^o\nabla^t\mathbf{u}^T)^T + ({}^o\nabla^t\mathbf{u}^T)({}^o\nabla^t\mathbf{u}^T)^T \right] \quad (2.35)$$

$${}^t\boldsymbol{\varepsilon} = \frac{1}{2} \left[{}^t\nabla^t\mathbf{u}^T + ({}^t\nabla^t\mathbf{u}^T)^T - ({}^t\nabla^t\mathbf{u}^T)({}^t\nabla^t\mathbf{u}^T)^T \right] \quad (2.36)$$

It should be noted that for a general referential formulation, the reference configuration does not have to be the one at $t = 0$ (e.g., as in updated Lagrangian formulation) and the Green-Lagrange strain tensor may be defined with respect to a general configuration at time $t = \tau$ [2.6, 2.7],

$${}^t\mathbf{E} = \frac{1}{2} \left[{}^\tau\nabla^t\mathbf{u}^T + ({}^\tau\nabla^t\mathbf{u}^T)^T + ({}^\tau\nabla^t\mathbf{u}^T)({}^\tau\nabla^t\mathbf{u}^T)^T \right] \quad (2.37)$$

The incremental strain tensors may be obtained from [Equation \(2.35\)](#) and [Equation \(2.36\)](#) as,

$$\Delta({}_0^t \mathbf{E}) = \Delta({}_0^t \mathbf{E})^{Linear} + \Delta({}_0^t \mathbf{E})^{Nonlinear} \quad (2.38)$$

$$\Delta({}_0^t \boldsymbol{\varepsilon}) = \Delta({}_0^t \boldsymbol{\varepsilon})^{Linear} + \Delta({}_0^t \boldsymbol{\varepsilon})^{Nonlinear} \quad (2.39)$$

where,

$$\begin{aligned} \Delta({}_o^t \mathbf{E})^{Linear} &= \frac{1}{2} \left[{}^o \nabla (\Delta^t \mathbf{u})^T + \left(\left({}^o \nabla (\Delta^t \mathbf{u})^T \right)^T \right) \right. \\ &\quad \left. + \left({}^o \nabla^t \mathbf{u}^T \right) \left({}^o \nabla (\Delta^t \mathbf{u})^T \right)^T \right. \\ &\quad \left. + \left({}^o \nabla (\Delta^t \mathbf{u})^T \right) \left({}^o \nabla^t \mathbf{u}^T \right)^T \right] \end{aligned} \quad (2.40)$$

$$\Delta({}_o^t \mathbf{E})^{Nonlinear} = \frac{1}{2} \left[\left({}^o \nabla (\Delta^t \mathbf{u})^T \right) \cdot \left({}^o \nabla (\Delta^t \mathbf{u})^T \right)^T \right] \quad (2.41)$$

$$\begin{aligned} \Delta({}_t^t \boldsymbol{\varepsilon})^{Linear} &= \frac{1}{2} \left[{}^t \nabla (\Delta^t \mathbf{u})^T + \left(\left({}^t \nabla (\Delta^t \mathbf{u})^T \right)^T \right) \right. \\ &\quad \left. - \left(\left({}^t \nabla^t \mathbf{u}^T \right) \left({}^t \nabla (\Delta^t \mathbf{u})^T \right)^T \right) \right. \\ &\quad \left. - \left({}^t \nabla (\Delta^t \mathbf{u})^T \right) \left({}^t \nabla^t \mathbf{u}^T \right)^T \right] \end{aligned} \quad (2.42)$$

$$\Delta({}_t^t \boldsymbol{\varepsilon})^{Nonlinear} = \frac{1}{2} \left[\left({}^t \nabla (\Delta^t \mathbf{u})^T \right) \cdot \left({}^t \nabla (\Delta^t \mathbf{u})^T \right)^T \right] \quad (2.43)$$

It should be noted from [Figure 2.2](#) that,

$$\Delta^t \mathbf{u} = {}^{t+\Delta t} \mathbf{u} - {}^t \mathbf{u} = {}^{t+\Delta t} \mathbf{u} \quad (2.44)$$

Displacement Assumption:

For total Lagrangian formulation, we assume the displacement shape function in the form,

$${}^t \mathbf{u}_i({}^O \mathbf{x}) = \sum_{k=1}^n N_k({}^O \mathbf{x}) {}^t \bar{\mathbf{u}}_i^k \quad (2.45)$$

$$\Delta^t \mathbf{u}_i({}^O \mathbf{x}) = \sum_{k=1}^n N_k({}^O \mathbf{x}) \Delta^t \bar{\mathbf{u}}_i^k \quad (2.46)$$

where, n is the number of nodes for the element.

Whereas, for updated Lagrangian formulation, we have,

$${}^t \mathbf{u}_i({}^\tau \mathbf{x}) = \sum_{k=1}^n N_k({}^\tau \mathbf{x}) {}^t \bar{\mathbf{u}}_i^k \quad (2.47)$$

$$\Delta^t \mathbf{u}_i({}^\tau \mathbf{x}) = \sum_{k=1}^n N_k({}^\tau \mathbf{x}) \Delta^t \bar{\mathbf{u}}_i^k \quad (2.48)$$

where, $\bar{\mathbf{u}}$ (and $\Delta \bar{\mathbf{u}}$) indicate nodal displacements (and nodal incremental displacements) and the left superscript τ indicates the time of a typical reference configuration in updated Lagrangian formulation [2.6, 2.7].

Other Relations:

The following relations and/or definitions may be utilized in the derivation of the final equilibrium equations for both the Lagrangian and the updated Lagrangian formulations,

$$\mathbf{T} = |\mathbf{F}| \mathbf{F}^{-1} \boldsymbol{\sigma} \quad (2.49)$$

where, \mathbf{F}^{-1} is the inverse of \mathbf{F} , $|\mathbf{F}|$ is the determinant of \mathbf{F} , $\boldsymbol{\sigma}$ is the Cauchy stress tensor, and \mathbf{T} is the first Piola-Kirchhoff, the nominal, or the Piola-Lagrange stress tensor.

$$\mathbf{S} = |\mathbf{F}| \mathbf{F}^{-1} \boldsymbol{\sigma} (\mathbf{F}^{-1})^T = \mathbf{T} (\mathbf{F}^{-1})^T \quad (2.50)$$

is the symmetric second Piola-Kirchhoff stress tensor, or the Kirchhoff-Trefftz tensor.

$$\mathbf{F} = \mathbf{R} \mathbf{U} = \mathbf{V} \mathbf{R} \quad (2.51)$$

is the deformation gradient tensor, whereas \mathbf{R} is the rotation tensor. \mathbf{U} and \mathbf{V} are the right and left stretch tensors, respectively.

The velocity gradient tensor \mathbf{L} is given by,

$$\mathbf{L} = (\overset{t}{\nabla} \mathbf{v})^T = \mathbf{D} + \mathbf{W} \quad (2.52)$$

where, \mathbf{D} is the rate of deformation tensor (symmetric tensor), \mathbf{W} is the spin or vorticity tensor (anti-symmetric tensor) and \mathbf{v} is the velocity vector.

Based on the definition of the rate of work per unit final volume expressed by $\sigma_{ij} \dot{D}_{ij}$, Hill [2.8] defines particular conjugate pairs of stress and strain measures,

$$\{ \boldsymbol{\sigma}, \mathbf{L} \}, \{ \mathbf{S}, \dot{\mathbf{E}} \}, \{ |\mathbf{F}| \mathbf{F}^T \boldsymbol{\sigma} \mathbf{F}, \boldsymbol{\varepsilon} \}, \{ \mathbf{T}(\mathbf{R}, \dot{\mathbf{U}}) \} \text{ and } \{ \mathbf{T}, \dot{\mathbf{F}} \}$$

where, the “.” over any variable indicates $d(\)/dt$. Among the above pairs, we will be using $(\boldsymbol{\sigma}, \mathbf{L})$ and $(\mathbf{S}, \dot{\mathbf{E}})$ for our referential (total and updated Lagrangian) formulations.

Stress-Strain Relation:

The constitutive stress-strain law for total and updated Lagrangian formulations is expressed in the following incremental form:

$$\Delta({}^tS_{ij}) = {}^tC_{ijkl}\Delta({}^tE_{kl}) \quad (2.53)$$

$$\Delta({}^t\sigma_{ij}) = {}^tC_{ijkl}\Delta({}^t\varepsilon_{kl}) \quad (2.54)$$

where, S , σ are the second Piola-Kirchhoff, Cauchy stress tensor and E , ε are the Green-Lagrange, Euler-Almansi strain tensors, respectively. The following fourth order tensor transformation is used to convert the constitutive tensor in the two equations.

$${}^tC_{ijkl} = \frac{1}{|{}^tF|} \frac{\partial {}^t x_i}{\partial {}^o x_m} \frac{\partial {}^t x_j}{\partial {}^o x_n} {}^o C_{mnpq} \frac{\partial {}^t x_k}{\partial {}^o x_p} \frac{\partial {}^t x_l}{\partial {}^o x_q} \quad (2.55)$$

$${}^o C_{ijkl} = |{}^tF| \frac{\partial {}^o x_i}{\partial {}^t x_m} \frac{\partial {}^o x_j}{\partial {}^t x_n} {}^t C_{mnpq} \frac{\partial {}^o x_k}{\partial {}^t x_p} \frac{\partial {}^o x_l}{\partial {}^t x_q} \quad (2.56)$$

2.3.3 Total Lagrangian Formulation

Referring to [Figure 2.2](#), the virtual work principle in the current configuration ${}^{t+\Delta t}C$ may be expressed as,

$$\int_{{}^{t+\Delta t}V} {}^{t+\Delta t}\sigma_{ij} \Delta {}^{t+\Delta t}D_{ij} d {}^{t+\Delta t}V = \int_{{}^{t+\Delta t}V} {}^{t+\Delta t}\rho {}^{t+\Delta t}b_i \Delta {}^{t+\Delta t}u_i d {}^{t+\Delta t}V \quad (2.57)$$

$$+ \int_{{}^{t+\Delta t}A} {}^{t+\Delta t}\sigma_{ij} \Delta {}^{t+\Delta t}u_j d {}^{t+\Delta t}A_i$$

where σ , D are the Cauchy stress and the rate of deformation tensors, b is the body force vector *per unit mass*, ρ is the material density, V is the volume, and A is the surface area.

A consistent transformation of [Equation \(2.55\)](#) from the current deformed configuration to the undeformed reference configuration gives [2.7],

$$\int_{^oV} {}^{t+\Delta t} \mathbf{S}_{ij} \Delta {}^{t+\Delta t} \mathbf{E}_{ij} d^oV = \int_{^oV} {}^o\rho {}^{t+\Delta t} \mathbf{b}_i \Delta {}^{t+\Delta t} \mathbf{u}_i d^oV$$

$$+ \int_{^oA} d^oA_i {}^{t+\Delta t} \mathbf{S}_{ij} {}^{t+\Delta t} \mathbf{F}_{kj} \Delta {}^{t+\Delta t} \mathbf{u}_k$$
(2.58)

where \mathbf{S} , \mathbf{E} are the second Piola-Kirchhoff stress and the Green-Lagrange strain tensors, and \mathbf{F} is the deformation gradient tensor. It should be noted that the expression of the surface traction term presented above leads to a consistent development of the load-correction matrix or the initial load stiffness matrix [2.6, 2.7].

Introducing the displacement assumption as well as applicable relations from [Section 2.3.2](#) and following the usual finite element incrementation procedures, the final equilibrium equations for the total Lagrangian formulation may be expressed as,

$$({}^o\mathbf{K}^1 + {}^o\mathbf{K}^2 + {}^o\mathbf{K}^3 - {}^o\mathbf{K}^A) \Delta {}^t \mathbf{u}^i = {}^{t+\Delta t} \mathbf{p} - {}^{t+\Delta t} \mathbf{f}^{i-l}$$
(2.59)

where,

$${}^o\mathbf{K}^{(1)} = \int_{^oV} {}^t \mathbf{B}_{lin}^T {}^t \mathbf{C} {}^t \mathbf{B}_{lin} d^oV$$
(2.60)

is the usual small displacement stiffness matrix,

$${}^o\mathbf{K}^{(2)} = \int_{^oV} {}^t \mathbf{B}_{nl2}^T {}^t \hat{\mathbf{S}} {}^t \mathbf{B}_{nl2} d^oV$$
(2.61)

is the initial stress or the geometric stiffness matrix,

$${}^o\mathbf{K}^{(3)} = \int_{^oV} {}^t \mathbf{B}_{nl1}^T {}^t \mathbf{C} {}^t \mathbf{B}_{nl1} d^oV$$
(2.62)

is the initial displacement or the initial rotation stiffness matrix and

$${}^t\mathbf{K}^A = \int_{{}^oA} {}^t\hat{\mathbf{N}}^T {}^o\hat{\mathbf{S}} {}^t\bar{\mathbf{B}}_{nl2} d {}^oV \quad (2.63)$$

is an unsymmetric initial load or load correction matrix [2.3, 2.7].

Expressions for various matrices used in Equation (2.60) to Equation (2.63) are given below, for a typical node k , in three dimensional case with three degrees of freedom, as a reference.

$${}^t\mathbf{B}_{lin} = \left[\begin{array}{c|ccc|c} & {}^tN_{k,1} & 0 & 0 & \\ & 0 & {}^tN_{k,2} & 0 & \\ & 0 & 0 & {}^tN_{k,3} & \\ \dots & {}^tN_{k,2} & {}^tN_{k,1} & 0 & \dots \\ & 0 & {}^tN_{k,3} & {}^tN_{k,2} & \\ & {}^tN_{k,3} & 0 & {}^tN_{k,1} & \end{array} \right] \quad (2.64)$$

where, k is the typical node and

$${}^tN_{k,i} = \frac{\partial {}^tN_k}{\partial {}^o x_i} \quad (2.65)$$

The first part of the nonlinear strain-displacement matrix, ${}^t\mathbf{B}_{nl1}$, which gives rise to the initial displacement or the initial rotation stiffness matrix is given by [2.5],

$${}^t\mathbf{B}_{nl1} = \begin{bmatrix} \mathbf{B}_{11} & \mathbf{B}_{12} & \mathbf{B}_{13} \\ \mathbf{B}_{21} & \mathbf{B}_{22} & \mathbf{B}_{23} \\ \mathbf{B}_{31} & \mathbf{B}_{32} & \mathbf{B}_{33} \\ \mathbf{B}_{41} & \mathbf{B}_{42} & \mathbf{B}_{43} \\ \mathbf{B}_{51} & \mathbf{B}_{52} & \mathbf{B}_{53} \\ \mathbf{B}_{61} & \mathbf{B}_{62} & \mathbf{B}_{63} \end{bmatrix} \quad (2.66)$$

$$\mathbf{B}_{11} = l_{11o} {}^tN_{k,1}$$

$$\mathbf{B}_{12} = l_{21o} {}^tN_{k,1}$$

$$\mathbf{B}_{13} = l_{31o} {}^tN_{k,1}$$

$$\mathbf{B}_{21} = l_{12o} {}^tN_{k,2}$$

$$\mathbf{B}_{22} = l_{22o} {}^tN_{k,2}$$

$$\mathbf{B}_{23} = l_{32o} {}^tN_{k,2}$$

$$\mathbf{B}_{31} = l_{13o} {}^tN_{k,3}$$

$$\mathbf{B}_{32} = l_{23o}^t N_{k,3}$$

$$\mathbf{B}_{33} = l_{33o}^t N_{k,3}$$

$$\mathbf{B}_{41} = (l_{11o}^t N_{k,2} + l_{12o}^t N_{k,1})$$

$$\mathbf{B}_{42} = (l_{21o}^t N_{k,2} + l_{22o}^t N_{k,1})$$

$$\mathbf{B}_{43} = (l_{31o}^t N_{k,2} + l_{32o}^t N_{k,1})$$

$$\mathbf{B}_{51} = (l_{13o}^t N_{k,2} + l_{12o}^t N_{k,3})$$

$$\mathbf{B}_{52} = (l_{23o}^t N_{k,2} + l_{22o}^t N_{k,3})$$

$$\mathbf{B}_{53} = (l_{33o}^t N_{k,2} + l_{32o}^t N_{k,3})$$

$$\mathbf{B}_{61} = (l_{11o}^t N_{k,3} + l_{13o}^t N_{k,1})$$

$$\mathbf{B}_{62} = (l_{21o}^t N_{k,3} + l_{23o}^t N_{k,1})$$

$$\mathbf{B}_{63} = (l_{31o}^t N_{k,3} + l_{33o}^t N_{k,1})$$

where, k is a typical node and

$$l_{ij} = \sum_{k=1}^n {}^tN_{k,j} {}^t\mathbf{u}_i^k \quad (2.67)$$

The nonlinear strain-displacement transformation matrix, ${}^t\mathbf{B}_{nl2}$, which gives rise to the initial stress or geometric stiffness matrix is given by [2.5, 2.7],

$${}^t\mathbf{B}_{nl2} = \left[\begin{array}{c|ccc|c} & {}^tN_{k,1} & 0 & 0 & \\ & {}^tN_{k,2} & 0 & 0 & \\ & {}^tN_{k,3} & 0 & 0 & \\ & 0 & {}^tN_{k,1} & 0 & \\ \cdots & 0 & {}^tN_{k,2} & 0 & \cdots \\ & 0 & {}^tN_{k,3} & 0 & \\ & 0 & 0 & {}^tN_{k,1} & \\ & 0 & 0 & {}^tN_{k,2} & \\ & 0 & 0 & {}^tN_{k,3} & \end{array} \right] \quad (2.68)$$

where, k is a typical node.

The definition of the stress matrix $\hat{\mathbf{S}}$ is given by,

$${}^t_o\hat{\mathbf{S}} = \begin{bmatrix} {}^t_o\mathbf{S} & \mathbf{O} & \mathbf{O} \\ \mathbf{O} & {}^t_o\mathbf{S} & \mathbf{O} \\ \mathbf{O} & \mathbf{O} & {}^t_o\mathbf{S} \end{bmatrix} \quad (2.69)$$

where, ${}^t_o\mathbf{S}$ is the second Piola-Kirchhoff stress tensor,

$${}^t_o\mathbf{S} = \begin{bmatrix} {}^t_oS_{11} & {}^t_oS_{12} & {}^t_oS_{13} \\ {}^t_oS_{21} & {}^t_oS_{22} & {}^t_oS_{23} \\ {}^t_oS_{31} & {}^t_oS_{32} & {}^t_oS_{33} \end{bmatrix} \quad (2.70)$$

and \mathbf{O} is a 3×3 null or zero matrix.

In the expression for the initial load or the load correction matrix [Equation \(2.63\)](#) the $\bar{\mathbf{B}}_{nl2}$ has the same expression as given in [Equation \(2.68\)](#) except that the bar indicates that the quantity is measured at the surface where the traction is applied, i.e., ${}^o\mathbf{A}$. The matrix $\hat{\mathbf{N}}$ in [Equation \(2.63\)](#) is given by,

$${}^t\hat{\bar{N}}_{nl2} = \left[\begin{array}{c|ccc|c} & {}^t\bar{N}_k & 0 & 0 & \\ & {}^t\bar{N}_k & 0 & 0 & \\ & {}^t\bar{N}_k & 0 & 0 & \\ & 0 & {}^t\bar{N}_k & 0 & \\ \dots & 0 & {}^t\bar{N}_k & 0 & \dots \\ & 0 & {}^t\bar{N}_k & 0 & \\ & 0 & 0 & {}^t\bar{N}_k & \\ & 0 & 0 & {}^t\bar{N}_k & \\ & 0 & 0 & {}^t\bar{N}_k & \end{array} \right] \quad (2.71)$$

Finally, the matrix ${}^o\hat{\bar{S}}$ is given by,

$${}^o\hat{\bar{S}} = \begin{bmatrix} {}^o\bar{S}_{(1)} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & {}^o\bar{S}_{(2)} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & {}^o\bar{S}_{(3)} \end{bmatrix} \quad (2.72)$$

where, $\mathbf{0}$ is a 3×3 zero or null matrix, and

$${}^o\bar{\mathbf{S}}_{(1)} = \begin{bmatrix} {}^o\bar{\mathbf{S}}_{(1)} & {}^o\bar{\mathbf{S}}_{(2)} & {}^o\bar{\mathbf{S}}_{(3)} \\ \mathbf{0} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{0} \end{bmatrix} \quad (2.73)$$

$${}^o\bar{\mathbf{S}}_{(2)} = \begin{bmatrix} \mathbf{0} & \mathbf{0} & \mathbf{0} \\ {}^o\bar{\mathbf{S}}_{(1)} & {}^o\bar{\mathbf{S}}_{(2)} & {}^o\bar{\mathbf{S}}_{(3)} \\ \mathbf{0} & \mathbf{0} & \mathbf{0} \end{bmatrix} \quad (2.74)$$

$${}^o\bar{\mathbf{S}}_{(3)} = \begin{bmatrix} \mathbf{0} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{0} \\ {}^o\bar{\mathbf{S}}_{(1)} & {}^o\bar{\mathbf{S}}_{(2)} & {}^o\bar{\mathbf{S}}_{(3)} \end{bmatrix} \quad (2.75)$$

and $({}^o\bar{\mathbf{S}}_{(1)}, {}^o\bar{\mathbf{S}}_{(2)}, {}^o\bar{\mathbf{S}}_{(3)})$ are the components of the surface traction vector (applied in the original configuration) in the global cartesian coordinates, where,

$${}^o(d\bar{\mathbf{S}})_i = {}^t_{\sigma}\mathbf{S}_{ji} {}^o\nu_j (d {}^oA) \quad (2.76)$$

where, ${}^t_{\sigma}\mathbf{S}_{ji}$ are the components of the second Piola-Kirchhoff stress tensor and ${}^o\nu_j$ are the direction cosines of the normal to the original surface area $(d {}^oA)$.

It is important to note that good results are obtained by taking only the symmetric part of the load correction or the initial load stiffness matrix [2.10].

2.3.4 Centrifugal Softening Effect

With reference to [Section 2.2.1](#), an additional stiffness term arises due to the deformation - dependent nature of the centrifugal forces. It should be noted that for nonlinear analysis, the generic point p ([Figure 2.1](#)) is considered to be on the deformed configuration (i.e., $\mathbf{r}_{po} = \mathbf{x}_p + \mathbf{u}$ where \mathbf{x}_p is the position vector of point p on the undeformed configuration). The following expression can be derived (see [Equation \(2.24\)](#) through [Equation \(2.26\)](#)),

$$\mathbf{K}_\omega = \int_V \rho \mathbf{N}^T \boldsymbol{\Omega} \boldsymbol{\Omega} \mathbf{N} dV \quad (2.77)$$

and since $\boldsymbol{\Omega}$ is a skew symmetric matrix

$$\mathbf{K}_\omega = - \int_V \rho \mathbf{N}^T \boldsymbol{\Omega}^T \boldsymbol{\Omega} \mathbf{N} dV \quad (2.78)$$

where, \mathbf{N} is the matrix of shape functions.

As an illustration of the centrifugal (or spin) softening effect, consider a 3-D solid element, and the special case when $\boldsymbol{\omega}^T = [0, 0, \omega_z]$, the (i, j) partition of \mathbf{K}_ω is then given by,

$$\mathbf{K}_\omega^{ij} = -\omega_z^2 \int_V \rho \begin{bmatrix} \mathbf{N}_i \mathbf{N}_j & 0 & 0 \\ 0 & \mathbf{N}_i \mathbf{N}_j & 0 \\ 0 & 0 & 0 \end{bmatrix} dV \quad (2.79)$$

which shows a softening effect in the plane of the rotation (the plane normal to the spin axis) in the direction of \mathbf{u}_x and \mathbf{u}_y .

The spin softening effect tends to reduce the stiffening effect of the centrifugal forces and is automatically included in all the NISA elements. The stiffening and softening effects are of practical importance in rotating structures. Through the eigenvalue restart capability ([Section 3.16](#)), the vibration characteristics of rotating structures may be analyzed.

2.4 Linear Dynamic Analysis

2.4.1 Governing Equations of Motion

The equations of motion describing the dynamic behavior of a structural system may be obtained from the extended Hamilton's principle for elastodynamics [2.11],

$$\delta \int_{t_0}^{t_f} (U - T) dt - \int_{t_0}^{t_f} \delta W dt = 0 \quad (2.80)$$

where,

δ = first variation

t_0, t_f = two arbitrary time points at which the first variation vanishes

U = strain energy in terms of geometrically compatible displacements

T = kinetic energy in terms of geometrically compatible displacements

δW = virtual work of the external forces acting on the system during virtual displacements

The external forces may be conservative or non conservative. A force is said to be conservative when its components are derivable from a potential function. Non conservative forces, e.g. damping and follower forces, are deformation dependent, their magnitudes and/or directions depend on the deformations. Note that during the virtual displacements, the variations of the nonconservative forces are ignored. The virtual work δW , therefore, can be written as,

$$\delta W = -\delta V + \delta W_{nc} \quad (2.81)$$

where, V is the potential of the applied conservative forces and δW_{nc} is the virtual work of the nonconservative forces during virtual displacements. Substituting [Equation \(2.81\)](#) into [Equation \(2.80\)](#), one obtains,

$$\delta \int_{t_0}^{t_f} (\Pi - T) dt - \int_{t_0}^{t_f} \delta W_{nc} dt = 0 \quad (2.82)$$

where, Π is the total potential energy of the system (see Equation (2.1)). Note that the above equation reduces to the statics case when the kinetic energy vanishes and the remaining terms are invariant with time.

In this section, viscous damping forces will be the only nonconservative force considered. Introducing the discretization Equation (2.11) into the kinetic energy and the virtual work of nonconservative forces, and with Equation (2.1) through Equation (2.15) in mind, we have,

$$\Pi = \Pi(\bar{\mathbf{u}}) = \sum_{e=1}^m \Pi^e(\bar{\mathbf{u}}^{(e)}) \quad (2.83)$$

$$T = T(\dot{\bar{\mathbf{u}}}) = \sum_{e=1}^m T(\dot{\bar{\mathbf{u}}}^{(e)}) = \frac{1}{2} \int_V \rho \dot{\mathbf{u}}^T \dot{\mathbf{u}} dV \quad (2.84)$$

$$\begin{aligned} &= \sum_{e=1}^m \frac{1}{2} \dot{\bar{\mathbf{u}}}^{(e)T} \mathbf{M}^{(e)} \dot{\bar{\mathbf{u}}}^{(e)} \\ \delta W_{nc} &= \sum_{e=1}^m \int_V \delta \mathbf{u}^T (-c_s^{(e)} \dot{\mathbf{u}}) dV \\ &= - \sum_{e=1}^m \delta \bar{\mathbf{u}}^{(e)T} \mathbf{C}^{(e)} \dot{\bar{\mathbf{u}}}^{(e)} = - \sum \delta \bar{\mathbf{u}}^{(e)T} \mathbf{f}_d^{(e)} \end{aligned} \quad (2.85)$$

where, $\bar{\mathbf{u}}$ and $\dot{\bar{\mathbf{u}}}$ are the nodal displacement and velocity vectors, respectively, and $\mathbf{M}^{(e)}$ and $\mathbf{C}^{(e)}$ are respectively the consistent mass and damping matrices for element e , given by,

$$\mathbf{M}^{(e)} = \int_V \rho^{(e)} \mathbf{N}^T \mathbf{N} dV \quad (2.86)$$

$$\mathbf{C}^{(e)} = \int_V c_s^{(e)} \mathbf{N}^T \mathbf{N} dV \quad (2.87)$$

where, $\rho^{(e)}$ and $c_s^{(e)}$ are the mass density and the viscous damping coefficient for element e , and \mathbf{N} is the matrix of shape functions (see Equation (2.11)).

The minus sign is used in Equation (2.85) since the damping force (viscous damping which is proportional to the velocity is assumed) opposes the motion. It should be noted that the damping matrix is not explicitly generated in NISA (except for the dashpot element in direct transient dynamic analysis). This as well as other forms of damping will be discussed in Equation (2.84).

Using Equation (2.84) through Equation (2.85) in the variational principle Equation (2.82), one obtains,

$$\int_{t_o}^{t_f} \left[\delta \bar{\mathbf{u}}^T \left[\frac{\partial U}{\partial \bar{\mathbf{u}}} - \frac{\partial T}{\partial \bar{\mathbf{u}}} + \mathbf{f}_d - \mathbf{p} \right] - \delta \dot{\bar{\mathbf{u}}}^T \frac{\partial T}{\partial \dot{\bar{\mathbf{u}}}} \right] dt = 0 \quad (2.88)$$

where, \mathbf{p} is the vector of applied loads (see Equation (2.16)), and \mathbf{f}_d is the damping force vector (assembly of $\mathbf{f}_d^{(e)}$) of Equation (2.85) associated with the generalized coordinates $\bar{\mathbf{u}}$.

Integrating the second term (involving $\dot{\bar{\mathbf{u}}}$) of Equation (2.88) by parts and observing that $\delta \bar{\mathbf{u}}$ vanishes at $t = t_o$ and $t = t_f$ is arbitrary in the open interval (t_o, t_f) , one obtains,

$$\frac{d}{dt} \left(\frac{\partial T}{\partial \dot{\bar{\mathbf{u}}}} \right) + \frac{\partial U}{\partial \bar{\mathbf{u}}} - \frac{\partial T}{\partial \bar{\mathbf{u}}} = \mathbf{p} - \mathbf{f}_d \quad (2.89)$$

which is a statement of Lagrange's equations. Using Equation (2.3) through Equation (2.15) together with Equation (2.84) through Equation (2.87), the governing differential equations of motion (the dynamic equilibrium equations) are obtained as,

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{p}(t) \quad (2.90)$$

where, \mathbf{M} , \mathbf{C} , and \mathbf{K} are respectively the global mass, damping and stiffness matrices (made up by a proper assembly of the element matrices), $\mathbf{p}(t)$ is the time-dependent applied force vector, $\ddot{\mathbf{u}}$, $\dot{\mathbf{u}}$ and \mathbf{u} are the nodal acceleration, nodal velocity and nodal displacement vectors, respectively (the bar is dropped for convenience, i.e., $\mathbf{u} \equiv \bar{\mathbf{u}}$).

All the dynamic analysis types in NISA proceed from the matrix [Equation \(2.90\)](#) as the governing equations for the dynamic equilibrium of the system. These equations are coupled through the off-diagonal terms in the coefficient matrices, and could be looked at as a generalization of the familiar equation for a single degree of freedom oscillator (mass-spring-dashpot system).

The mass matrix generated using [Equation \(2.86\)](#) is referred to as the consistent mass matrix because it is formulated in a way which is consistent with that of the stiffness matrix. A lumped mass formulation, which results in a diagonal mass matrix, is provided in NISA as an option. In this formulation ([2.14, 2.15]), the lumped mass matrix is derived by distributing the translational mass of the element in proportion to the diagonal terms of the consistent mass matrix associated with translation, thus preserving the translational mass of the element. In general, this procedure yields more accurate results in comparison to the traditional lumped mass matrix approach, in which the distributed element mass is replaced by point masses at the nodes with each point mass representing part of the element mass. For line elements (spars, beams), the traditional lumped mass approach is used in NISA. It should be noted, that rotary inertia are not represented in the lumped mass formulation, thus the lumped mass matrix will have zero diagonal terms corresponding to rotational degrees of freedom.

The coefficient matrices \mathbf{K} , \mathbf{M} and \mathbf{C} in [Equation \(2.90\)](#) are real and symmetric matrices. The consistent mass matrix is positive definite since the kinetic energy, a quadratic form in terms of the mass matrix (see [Equation \(2.84\)](#)), is itself a positive definite quantity. (The lumped mass matrix is positive semi-definite if zeros appear on one or more of the diagonal entries). A real and symmetric matrix \mathbf{A} is said to be positive definite if the quadratic form $\mathbf{x}^T \mathbf{A} \mathbf{x}$ is greater than zero for any \mathbf{x} , and is zero only when \mathbf{x} is a null vector. A real, symmetric and positive definite matrix has all of its eigenvalues as real and positive. The stiffness matrix \mathbf{K} is positive definite, when the structure is constrained against all possible rigid body motions, and is positive semi-definite when some or all of the rigid body modes are unconstrained. In the latter case, the strain energy, a quadratic form in terms of the stiffness matrix, is zero for the rigid body modes (which do not strain the body).

The different dynamic analysis types in NISA correspond to special forms of the governing [Equation \(2.90\)](#). The applicability of each analysis type depends on the type of excitation $\mathbf{p}(t)$.

The available types of dynamic analysis in NISA are:

1. Eigenvalue analysis (undamped free vibration analysis)
2. Transient dynamic analysis (forced vibration due to prescribed time dependent loads)
3. Frequency response analysis (response in terms of amplitudes and phase angles due to harmonic loads defined in terms of amplitude and phase angle spectra)
4. Random vibration analysis (provides statistical measures of the response due to random excitations of known statistical properties)
5. Shock spectrum analysis (provides estimates of the maxima of the response due to base excitation shock spectra input defined in terms of the response of a series of single degree of freedom systems with various natural frequencies and damping ratios)

The eigenvalue analysis provides the basis for the normal mode method (also known as mode-superposition) used in the other analysis types. For transient dynamic analysis, a direct numerical integration approach is also available ([Section 2.4.10](#)).

2.4.2 Eigenvalue Analysis

Eigenvalue analysis is used to obtain the dynamic characteristics of a system in terms of its natural frequencies (eigenvalues) and associated undamped free vibration modes (eigenvectors). These eigen modes may then be used to decouple the general equations of motion in the other types of dynamic analysis. Specializing [Equation \(2.90\)](#) for undamped free vibration ($\mathbf{C} = \mathbf{O}$ and $\mathbf{p}(t) = \mathbf{O}$, where \mathbf{O} is a null matrix or vector), we have

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{O} \tag{2.91}$$

Assuming a solution of the form,

$$\mathbf{u} = \bar{\varphi} \mathbf{e}^{i\omega t + \psi} \tag{2.92}$$

where, i is the unit imaginary number, gives

$$(\mathbf{K} - \lambda \mathbf{M})\bar{\boldsymbol{\varphi}} = \mathbf{0} \quad (2.93)$$

which is a general eigenvalue problem, where $\lambda = \omega^2$. For a nontrivial solution ($\bar{\boldsymbol{\varphi}} \neq \mathbf{0}$), the coefficient matrix $(\mathbf{K} - \lambda \mathbf{M})$ must be singular, from which the characteristic equation of the system is obtained as:

$$\det(\mathbf{K} - \lambda_i \mathbf{M}) = 0 \quad (2.94)$$

and λ_i are the roots of this equation. In general, there are n such roots, where n is the number of degrees of freedom in \mathbf{u} . If the mass matrix is not positive definite, as may be the case for a lumped mass matrix with some zero diagonals for example, then the number of eigenpairs will be less than n .

For each λ_i , $i = 1, 2, \dots, n$, a corresponding $\bar{\boldsymbol{\varphi}}_i$ can be found from [Equation \(2.93\)](#). Thus the solution to the general eigenvalue problem may be obtained in terms of n eigenpairs $(\lambda_i, \bar{\boldsymbol{\varphi}}_i)$. Several eigenvalue extraction methods are available in NISA. These are discussed in [Section 3.5.2](#).

Since both \mathbf{K} and \mathbf{M} are real and symmetric, it can be shown that all the roots λ_i (consequently $\bar{\boldsymbol{\varphi}}_i$) are real. Furthermore, if $\bar{\boldsymbol{\varphi}}_i$ is an eigenvector, then $c\bar{\boldsymbol{\varphi}}_i$, where c is a constant, is also an eigenvector as can be seen from [Equation \(2.93\)](#). The following orthogonality properties can also be derived,

$$\begin{aligned} \boldsymbol{\varphi}_i^T \mathbf{M} \boldsymbol{\varphi}_j &= 0 & \text{for } i \neq j \\ &= 1 & \text{for } i = j \end{aligned} \quad (2.95)$$

$$\begin{aligned} \boldsymbol{\varphi}_i^T \mathbf{K} \boldsymbol{\varphi}_j &= 0 & \text{for } i \neq j \\ &= \omega_i^2 & \text{for } i = j \end{aligned} \quad (2.96)$$

where,

$$\boldsymbol{\varphi}_i = \bar{\boldsymbol{\varphi}}_i / \sqrt{\mathbf{m}_i} \quad (2.97)$$

$$m_i = \bar{\varphi}_i^T \mathbf{M} \bar{\varphi}_i \quad (2.98)$$

In NISA, the eigenvectors are scaled as shown in Equation (2.97) to yield a generalized modal mass ($\bar{\varphi}_i^T \mathbf{M} \bar{\varphi}_i$) of unity for all modes. With this method of normalization, the eigenvectors are said to be *M*-orthonormal, i.e., orthonormal relative to the mass matrix.

Examination of Equation (2.96) reveals that an eigenvalue (square of a natural frequency) is twice the modal strain energy. Thus, for a properly constrained structure, \mathbf{K} is positive definite and $\lambda_i, i = 1, 2, \dots, n$ are all positive quantities. On the other hand, for an unconstrained or partially constrained structure, \mathbf{K} is only positive (i.e., positive semi-definite) and therefore the eigenvalues are nonnegative ($\lambda_i \geq 0$), with the zero frequencies for the rigid body modes which do not cause any strain energy.

Equation (2.95) and Equation (2.96) may also be written in terms of the matrix of eigenvectors, to read,

$$\Phi^T \mathbf{M} \Phi = \mathbf{I} \quad (2.99)$$

$$\Phi^T \mathbf{K} \Phi = \text{diag}(\omega_i^2) \quad (2.100)$$

where,

$$\Phi = [\varphi_1, \varphi_2, \dots, \varphi_n] \quad (2.101)$$

is the matrix of eigenvectors and \mathbf{I} is an $n \times n$ identity matrix. The n eigenvectors constitute n independent displacement patterns and therefore form a complete basis in the n -dimensional vector space. Note, however, that Equation (2.99) and Equation (2.100) are still valid if a subset of the eigenvectors is used.

2.4.3 Normal Mode Method

In this method, the nodal displacement response is expressed in terms of the normal modes found in the eigenvalue analysis. Because of their orthogonality properties, the coupled equations of motion Equation (2.90) are transformed into a set of independent differential equations cast in modal generalized coordinates form. The dynamic response of the original system is then obtained by superposing the responses of the uncoupled modal equations [2.12]. The generalized coordinates (modal coordinates) are introduced by the following coordinate transformation,

$$\mathbf{u} = \Phi \mathbf{q} = \sum_{i=1} \varphi_i \mathbf{q}_i \quad (2.102)$$

where, Φ is an $n \times m$ eigenvector (mode shape) matrix (see Equation (2.101)) having as its columns m number of eigenvectors ($m \leq n$) and \mathbf{q} is a vector of m number of mode-amplitude generalized coordinates (normal coordinates). The transformation Equation (2.102) represents a change of basis from the nodal displacements to the modal generalized coordinates.

Introducing Equation (2.102) into Equation (2.90) and premultiplying by Φ^T and using the orthogonality relationships Equation (2.99) and Equation (2.100) gives,

$$\ddot{\mathbf{q}} + \Phi^T \mathbf{C} \Phi \dot{\mathbf{q}} + \text{diag}(\omega_i^2) \mathbf{q} = \Phi^T \mathbf{p}(t) \quad (2.103)$$

For the triple product $\Phi^T \mathbf{C} \Phi$ to be a diagonal matrix, the following orthogonality property is assumed for damping [2.13],

$$\Phi^T \mathbf{C} \Phi = \text{diag}(2\xi_i \omega_i) \quad (2.104)$$

where, ξ_i is the damping ratio for mode i . This form has been assumed by generalization of damping for a single degree of freedom system. It is much more convenient and physically reasonable to define the damping by the damping ratios of each mode than it is to evaluate the damping matrix explicitly [2.13].

From Equation (2.51) and Equation (2.104), the r-th (uncoupled) modal equation, $r = 1, 2, \dots, m$ becomes,

$$\ddot{\mathbf{q}}_r + 2\xi_r\omega_r\dot{\mathbf{q}}_r + \omega_r^2\mathbf{q}_r = \mathbf{f}_r(t) \quad (2.105)$$

in which $\mathbf{f}_r(t)$ is the forcing function for mode r , defined as,

$$\mathbf{f}_r(t) = \Phi_r^T \mathbf{p}(t) \quad (2.106)$$

If the components of the applied force vector have the same time function, such that $\mathbf{p}(t)$ can be expressed as,

$$\mathbf{p}(t) = \bar{\mathbf{p}}g(t) \quad (2.107)$$

where, $\bar{\mathbf{p}}$ is a constant vector, then the modal load can be written as,

$$\mathbf{f}_r(t) = \Gamma_r g(t) \quad (2.108)$$

$$\Gamma_r = \Phi_r^T \bar{\mathbf{p}} \quad (2.109)$$

The quantity Γ_r is referred to as the modal participation factor. A particular use of this quantity is in ground motion type of excitation (see Section 2.4.5).

The initial conditions at time $t = 0$, $\mathbf{u}(0) = \mathbf{u}_o$ and $\dot{\mathbf{u}}(0) = \dot{\mathbf{u}}_o$ are transformed to the normal coordinates space by premultiplying Equation (2.102) by $\Phi^T \mathbf{M}$ to give,

$$\mathbf{q}(0) = \Phi^T \mathbf{M} \mathbf{u}_o \quad (2.110)$$

$$\dot{\mathbf{q}}(0) = \Phi^T \mathbf{M} \dot{\mathbf{u}}_o \quad (2.111)$$

Once the generalized coordinates \mathbf{q} are evaluated, the physical response of the original system Equation (2.11) in terms of nodal displacements, velocities and accelerations and stresses are recovered from Equation (2.102). For the stress recovery, it is noted that (Equation (2.5), Equation (2.6), Equation (2.13) and Equation (2.15)),

$$\begin{aligned} \sigma(t) &= \mathbf{CB}\bar{\mathbf{u}}^{(e)}(t) = \mathbf{CB} \left[\sum_{r=1}^m \varphi_r^{(e)} \mathbf{q}_r(t) \right] \\ &= \sum_{r=1}^m [\mathbf{CB}\varphi_r^{(e)}] \mathbf{q}(t) = \sum_{r=1}^m \sigma_r \mathbf{q}_r(t) \end{aligned} \tag{2.112}$$

where, σ_r lists the stress components corresponding to the r-th mode at a typical point. That is, σ_r ($r = 1, 2, \dots, m$) are modal stresses which should be interpreted as stress shapes. It should be noted that matrix \mathbf{C} in Equation (2.112) refers to the material matrix and not to the damping matrix.

The above derivation of the normal mode method showed the decoupling advantage of the normal coordinates, whereby the change of basis from the nodal displacements to the normal modes yielded a set of uncoupled modal equations, with each equation cast in the form of a single degree of freedom oscillator. Another major advantage of the normal mode method is that a good approximation to the response may be obtained using a drastically reduced number of coordinates ($m \ll n$) in the series Equation (2.102). For most type of loadings, the contributions of the lowest few modes are generally more pronounced than the higher modes. Furthermore, practical finite element idealization approximates the lower modes better and tends to be less reliable for higher modes of vibration. In general, the number of modes to be used depends on the spatial distribution and the frequency content of the loading. Guidelines for mode selection are given in Section 3.5.3.

2.4.4 Damping Characteristics

Commonly, damping is described in viscous or structural form. For viscous damping, the damping force is proportional to and opposing the velocity. In case of structural damping, the damping force is proportional in magnitude to the internal elastic force (i.e., to the displacement) and is opposite in direction to the velocity. Considering a single degree of freedom system, the above relations are expressed as [2.12, 2.13],

$$f_d = -c\dot{u} \quad (\text{viscous damping}) \quad (2.113)$$

$$f_d = -gk|u|\frac{\dot{u}}{|\dot{u}|} \quad (\text{structural damping}) \quad (2.114)$$

Damping may be introduced in NISA by means of:

1. Discrete viscous damper elements (dashpots). These elements are damping counterparts of the spring elements and are discrete idealization of viscous damping in the structure. The damping matrix resulting from these elements, in general, cannot be decoupled as in [Equation \(2.104\)](#). Hence these elements may be used only in direct transient dynamic analysis.
2. Proportional viscous damping (Rayleigh damping), which is of the form,

$$C = c_1K + 2c_2M \quad (2.115)$$

where, c_1 and c_2 are constants to be determined from two damping ratios corresponding to two unequal natural frequencies. Proportional damping may be used in modal dynamic analysis and in direct transient analysis. Since C is proportional to K and/or M , it satisfies the orthogonality property in

That is,

$$c_1\omega_i^2 + 2c_2 = 2\xi_i\omega_i \quad (2.116)$$

which leads to,

$$\xi_i = \frac{1}{2}\left(c_1\omega_i + \frac{2c_2}{\omega_i}\right) \quad (2.117)$$

For the case when \mathbf{C} is proportional to \mathbf{K} only ($c_2 = 0$), the damping ratio is proportional to the natural frequency, thus the higher modes will be damped more than the lower modes [2.13]. Similarly, if \mathbf{C} is proportional to \mathbf{M} only ($c_1 = 0$), the damping ratio is inversely proportional to the natural frequency, and the higher modes will have less damping than the lower modes. The relationship of Equation (2.117) is illustrated in Figure 2.3.

Proportional damping with one set of values of c_1 and c_2 may be used in both direct transient and modal dynamic analyses. For direct transient dynamic analysis, different sets of constants c_1 and c_2 may be assigned to different parts of the model. This may result in a non-orthogonal damping matrix. The damping orthogonality condition, however, is not required in direct transient dynamic analysis since the governing Equation (2.90) are directly integrated.

3. Modal viscous damping, in which the damping ratios are directly specified for the participating modes. This type of damping can only be used in modal dynamic analyses.
4. Modal structural damping, in which the structural damping coefficients are specified directly for the participating modes. This type of damping can be used only in modal frequency response and random vibration analyses. The modal equations in this case have the form,

$$\ddot{\mathbf{q}}_r + \omega_r^2 (1 + i g_r) \mathbf{q}_r = \mathbf{f}_r(t) \quad (2.118)$$

where, i is the unit imaginary number and g_r , is the modal structural damping coefficient.

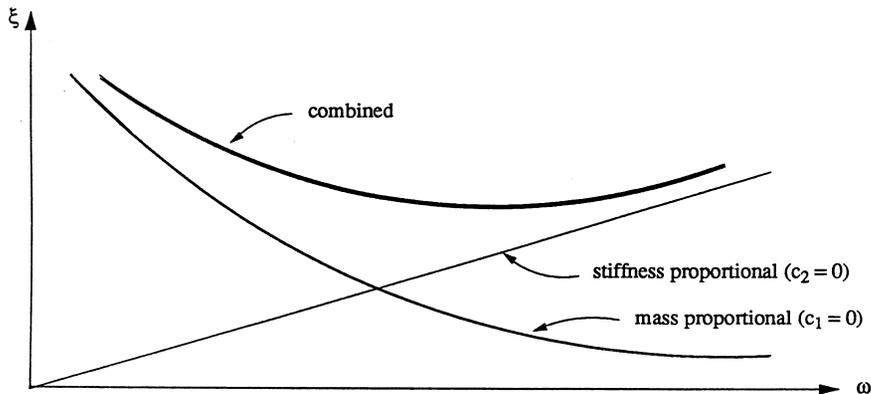


Figure 2.3: Proportional Damping

2.4.5 Ground Motion

Uniform Base Excitation

The equations of motion for a system subjected to ground accelerations in terms of translations (identical at all support points) and rotations (rocking motion about a specified point) have the form [2.13],

$$M\ddot{u}^t + C\dot{u}^t + Ku^t = O \quad (2.119)$$

where the superscript t denotes total motion, which can be decomposed into relative and rigid body motions in the form,

$$u^t = u + T\omega \quad (2.120)$$

with similar expressions for the total nodal velocity and acceleration vectors, where u and its time derivatives denote relative motion with respect to the base, and ω and its time derivatives denote the base motion. The base acceleration vector has six components and can be written as,

$$\ddot{\omega}^T = [\ddot{\omega}_T^T, \ddot{\omega}_R^T] \quad (2.121)$$

where, $\ddot{\omega}_T$ lists 3 components for the base translations and $\ddot{\omega}_R$ lists 3 rotational components. The transformation matrix T is an $n \times 6$ matrix and relates the rigid body motion of the nodes to the base motion, where n is the number of degrees of freedom in u . Matrix T can be written as,

$$T^T = [T_1^T, T_2^T, \dots, T_N^T] \quad (2.122)$$

$$T_i = \begin{bmatrix} I & R \\ O & I \end{bmatrix} \quad (2.123)$$

$$R = \begin{bmatrix} 0 & (z_i - z_o) & -(y_i - y_o) \\ -(z_i - z_o) & 0 & (x_i - x_o) \\ (y_i - y_o) & -(x_i - x_o) & 0 \end{bmatrix} \quad (2.124)$$

where, $T_i (i = 1, 2, \dots, N)$ is the transformation matrix for node i , N is the number of nodes, (x_i, y_i, z_i) are the coordinates of the i -th node and (x_o, y_o, z_o) are the coordinates of the point about which rocking motion, if any, takes place. It should be noted that if a node has less than six degrees of freedom, then only the appropriate rows in T_i are considered.

Substituting Equation (2.120) into Equation (2.119) and observing that the inertia forces depend on the total motion, whereas the elastic and damping forces depend on the relative motion, gives,

$$M\ddot{u} + C\dot{u} + Ku = p_{eff}(t) \quad (2.125)$$

which is of the same form as Equation (2.90) expressed in terms of relative motion and with an effective applied load,

$$P_{eff} = -MT\ddot{\omega} \quad (2.126)$$

For the normal load method, the forcing function for the r -th mode (see Equation (2.106)), becomes,

$$\mathbf{f}_r(t) = -\Gamma_r^T \ddot{\omega} \quad (2.127)$$

where, Γ_r lists the participation factors in 6 directions, and is defined by,

$$\Gamma_r^T = \Phi_r^T \mathbf{M} \mathbf{T} \quad (2.128)$$

The participation factors for all modes are listed in Γ , where,

$$\Gamma^T = \Phi^T \mathbf{M} \mathbf{T} \quad (2.129)$$

$$\Gamma = [\Gamma_1, \Gamma_2, \dots, \Gamma_n] \quad (2.130)$$

Let $\gamma_j(j = 1, 2, \dots, 6)$ denote the participation factors for all modes in direction j , we can write,

$$\gamma_j = \Phi^T \mathbf{M} \mathbf{t}_j \quad (2.131)$$

where, \mathbf{t}_j is the j -th column of \mathbf{T} . Now, when all modes are considered, Φ is a square matrix, and from the orthogonality property [Equation \(2.99\)](#), we can write,

$$\Phi^T \mathbf{M} = \Phi^{-1} \quad (\Phi \text{ is } n \times n) \quad (2.132)$$

substituting in [Equation \(2.131\)](#) gives,

$$\mathbf{t}_j = \Phi \gamma_j \quad (2.133)$$

Now consider the product $\mathbf{t}_j^T \mathbf{M} \mathbf{t}_j$. For a lumped mass matrix, this product can be interpreted as the total mass of the structure in the j -th direction (mass moment of inertia about global axes passing through the point of rotation, for rotational components). From [Equation \(2.133\)](#), we have,

$$\mathbf{t}_j^T \mathbf{M} \mathbf{t}_j = \gamma_j^T \Phi^T \mathbf{M} \Phi \gamma_j = \gamma_j^T \gamma_j = \sum_{r=1}^n \Gamma_{r,j}^2 \quad (2.134)$$

where, $\Gamma_{r,j}$ is the participation factor for mode r in the j -th direction. Thus, the summation of the squares of the participation factors of all modes in a typical direction may be interpreted as the total mass in that direction. The quantity $\Gamma_{r,j}^2$ is referred to as the effective modal mass.

Modal participation factors and effective masses are calculated and printed in the eigenvalue analysis. Since at this point the location of the point about which rocking motion occurs is not known, the participation factors for the rotational components are referred to the global origin. They are adjusted to refer to the specified point of rotation in the modal dynamic analysis run.

It should be noted that relative motion with respect to the ground has been considered in the above derivation. Therefore, the nodal responses, in terms of displacement, velocity and accelerations, are relative to the ground. Since stresses are unaffected by rigid body motion, they are properly calculated when relative response is considered. A procedure to obtain absolute response as well as to apply different enforced motions at different support points (multiple support excitation) is given in [Section 3.5.3](#).

Multiple Support Excitation

The governing equation of motion under N_g number of support excitations can be cast in the following form (Ref. [2.59]),

$$\bar{\mathbf{M}} \ddot{\bar{\mathbf{u}}} + \bar{\mathbf{C}} \dot{\bar{\mathbf{u}}} + \bar{\mathbf{K}} \bar{\mathbf{u}} = \bar{\mathbf{P}}(t) \quad (2.135)$$

where the global mass, damping and stiffness matrices in partition form are given by,

$$\bar{\mathbf{M}} = \begin{bmatrix} \mathbf{M} & \mathbf{M}_g \\ \mathbf{M}_g^T & \mathbf{M}_{gg} \end{bmatrix}, \bar{\mathbf{C}} = \begin{bmatrix} \mathbf{C} & \mathbf{C}_g \\ \mathbf{C}_g^T & \mathbf{C}_{gg} \end{bmatrix}, \bar{\mathbf{K}} = \begin{bmatrix} \mathbf{K} & \mathbf{K}_g \\ \mathbf{K}_g^T & \mathbf{K}_{gg} \end{bmatrix}, \quad (2.136)$$

$$\bar{\mathbf{P}}(t) = [\mathbf{P}(t)\mathbf{P}_g(t)]^T$$

Here the partition is made into superstructure part and the support part (suffix 'g') of the total structure. The structure is assumed to be viscously damped and the damping matrix is assumed such that the undamped normal modes of the system are orthogonal to the damping matrix.

The displacement vector is accordingly partitioned as,

$$\bar{\mathbf{u}} = \{\mathbf{u}_T \ \mathbf{u}_g\}^T \quad (2.137)$$

$\mathbf{u}_T(t)$ is the total displacement vector of superstructure of size N and $\mathbf{u}_g(t)$ is support displacement vector of size N_g .

The total displacement vector consists of two parts.

$$\mathbf{u}_T(t) = \mathbf{u}_{qs}(t) + \mathbf{u}(t) \quad (2.138)$$

Here $\mathbf{u}(t)$ is the dynamic displacement and $\mathbf{u}_{qs}(t)$ is the quasi static solution, which is due to static application of the prescribed support displacement $\mathbf{u}_g(t)$, given by,

$$\mathbf{u}_{qs}(t) = \mathbf{L}\mathbf{u}_g(t) \quad (2.139)$$

where, \mathbf{L} is the influence matrix of size $N \times N_g$. The influence vectors are obtained as the vectors of static displacements of the superstructure degrees of freedom due to application of unit displacement at each of the supports with one applied at a time. The matrix \mathbf{L} is obtained from a static analysis in the form,

$$\mathbf{L} = -\mathbf{K}^{-1} \mathbf{K}_g \quad (2.140)$$

The governing equation of motion for the dynamic displacement $\mathbf{u}(t)$ can be given by,

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{P}_{eff}(t) \quad (2.141)$$

Neglecting the inertial and damping forces, the effective force vector is given by,

$$\mathbf{P}_{eff}(t) = -(\mathbf{M}\ddot{\mathbf{u}} + \mathbf{M}_g\ddot{\mathbf{u}}_g) - (\mathbf{C}\mathbf{u} + \mathbf{C}_g\dot{\mathbf{u}}) \approx -(\mathbf{M}\mathbf{L} + \mathbf{M}_g)\ddot{\mathbf{u}}_g \quad (2.142)$$

Missing Mass Correction

Presently, missing mass correction is applicable only in the case of support excitation. In modal dynamics, accuracy of the response depends on the number of modes considered in the summation. However, a correction for the missing modes can be made by neglecting the inertial and damping forces of the neglected modes. The displacement solution $u_i(t)$ at any i^{th} degree of freedom is expressed as,

$$u_i(t) = \sum_{j=1}^m \phi_{ij} q_j(t) \quad (2.143)$$

Where, $q_j(t)$ the modal response in mode j and m is the total number of modes extracted from eigenvalue analysis.

Let p be the modal participation vector and it is given by [Equation \(2.144\)](#) and [Equation \(2.145\)](#) for uniform base excitation and multiple support excitations cases respectively.

$$p_j = \phi_j^T \mathbf{M} \mathbf{T} \quad (2.144)$$

$$p_j = \phi_j^T \{ \mathbf{M} \mathbf{L} + \mathbf{M}_g \} \quad (2.145)$$

Here, \mathbf{T} is a $N \times 6$ transformation matrix that relates the rigid body motion of the nodes to the base motion and \mathbf{L} is the influence matrix as defined in [Section 2.4.5](#).

The modal summation in [Equation \(2.143\)](#) is divided into two parts: (1) the first n modes with the natural periods such that dynamic effects are significant and (2) modes $n+1$ to m with time periods such that the dynamic load factors corresponding to these high frequency modes are close to unity. Here the dynamic load factor is defined in terms of the dynamic response amplification of a SDOF system relative to its static counterpart under a harmonic excitation.

Thus the [Equation \(2.143\)](#) takes the form,

$$u_i(t) = \sum_{j=1}^n \phi_{ij} q_j(t) + \sum_{j=n+1}^m \phi_{ij} q_j(t) \quad (2.146)$$

For the high frequency modes, a static solution of the modal response is given by,

$$\omega_j^2 q_j = -p_j \ddot{u}_g \quad (2.147)$$

Hence, the displacement solution is given by,

$$u_i(t) = \sum_{j=1}^n \phi_{ij} q_j(t) + \sum_{j=n+1}^m \left(-\phi_{ij} \frac{p_j \ddot{u}_g}{\omega_j^2} \right) \quad (2.148)$$

For the uniform base excitation case, the second sum in [Equation \(2.148\)](#) above is rewritten in the following form to avoid computation of the participation factors for the modes $n+1$ to m ,

$$\sum_{j=n+1}^m \left(-\phi_{ij} \frac{p_j \ddot{u}_g}{\omega_j^2} \right) = -\mathbf{K}^{-1} \mathbf{M} \mathbf{I} \ddot{u}_g - \sum_{j=1}^n \left(-\phi_{ij} \frac{p_j \ddot{u}_g}{\omega_j^2} \right) \quad (2.149)$$

The first term on the RHS of Equation (2.149) is the total static solution of the structure and the second is the static solution of the first m modes and this second term can be shown equal to $-K^{-1}M\phi p$ and response contribution for modes $n+1$ to m is finally given by,

$$\sum_{j=n+1}^m \left(-\phi_{ij} \frac{p_j \ddot{u}_g}{\omega_j^2} \right) = K^{-1} [-M(I - \phi p) \ddot{u}_g] \quad (2.150)$$

In Equation (2.150), $K^{-1} [-M(I - \phi p) \ddot{u}_g]$ is recognized as a pseudo static solution with respect to the ground acceleration \ddot{u}_g for the high frequency modes $n+1$ to m . The expression $-M(I - \phi p)$ is treated as a pseudo force vector per unit ground acceleration. Hence the dynamic response at any degree of freedom is now given by,

$$u_i(t) = \sum_{j=1}^n \phi_{ij} q_j(t) + K^{-1} [-M(I - \phi p)] \ddot{u}_g(t) \quad (2.151)$$

Similarly, dynamic response for the multiple support excitation case can be shown to be given by,

$$u_i(t) = \sum_{j=1}^n \phi_{ij} q_j(t) + K^{-1} [-M(L - \phi p) - M_g] \ddot{u}_g(t) \quad (2.152)$$

2.4.6 Modal Transient Dynamic Analysis

This analysis may be used to determine the response due to specified time-dependent loads, and is based on the mode superposition method (Section 2.4.3). Direct transient analysis, which integrates the coupled equations (Equation (2.90)) directly, is discussed in Section 2.4.10.

The uncoupled modal [Equation \(2.105\)](#), repeated here for convenience,

$$\ddot{\mathbf{q}} + \text{diag}(2\xi_r\omega_r)\dot{\mathbf{q}} + \text{diag}(\omega_r^2)\mathbf{q} = \{ \mathbf{f}(t) \} = \Phi^T \mathbf{p}(t) \quad (2.153)$$

consist of m independent second order differential equations with constant coefficients, where m is the number of the participating modes used in the mode superposition process. At time $t = 0$, each modal equation is subject to the initial conditions (see [Equation \(2.110\)](#) and [Equation \(2.111\)](#)),

$$q_r(0) = \Phi_r^T \mathbf{M} \mathbf{u}_o \quad (2.154)$$

$$\dot{q}_r(0) = \Phi_r^T \mathbf{M} \dot{\mathbf{u}}_o \quad (2.155)$$

For the underdamped case ($\xi_r < 1$), the solution to a typical modal equation (excluding rigid body modes, see below) is,

$$q_r(t) = e^{-\xi_r\omega_r t} [\alpha_r \sin \bar{\omega}_r t + \beta_r \cos \bar{\omega}_r t] + \int_0^t f_r(r) h_r(t-r) dr \quad (2.156)$$

where, $h_r(t - \tau)$ is the unit-impulse response function, defined by,

$$h_r(t - \tau) = \frac{1}{\omega_r} e^{-\xi_r\omega_r(t-\tau)} \sin \bar{\omega}_r(t - \tau) \quad (2.157)$$

The first term in [Equation \(2.156\)](#) represents the free vibration response (homogenous solution) and the second term represents the particular solution (Duhamel integral), α_r , and β_r , are constants evaluated from the initial conditions, and

$$\bar{\omega}_r = \omega_r \sqrt{1 - \xi_r^2} \quad (2.158)$$

The integral in [Equation \(2.156\)](#) and [Equation \(2.157\)](#) can be evaluated in closed form if the applied loading $\mathbf{p}(t)$, consequently $\mathbf{f}(t)$, is a piecewise linear function of time, such that,

$$f_r(\tau) = f_{r,n} + \frac{f_{r,n+1} - f_{r,n}}{h}(\tau - t_n) \quad (2.159)$$

where,

$$f_{r,n} = f_r(t_n), f_{r,n+1} = f_r(t_{n+1}) \text{ and } h = t_{n+1} - t_n, t_n \leq \tau \leq t_{n+1}$$

Using Equation (2.159) in Equation (2.156), Equation (2.157), the solution for the elastic modes is obtained as,

$$\begin{Bmatrix} q_r \\ \dot{q}_r \end{Bmatrix}_{n+1} = \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} \begin{Bmatrix} q_r \\ \dot{q}_r \end{Bmatrix}_n + \begin{bmatrix} b_{11} & b_{12} \\ b_{21} & b_{22} \end{bmatrix} \begin{Bmatrix} f_{r,n} \\ f_{r,n+1} \end{Bmatrix}, n = 0, 1, 2, \dots \quad (2.160)$$

$$\ddot{q}_n = f_{r,n} - 2\xi_r \omega_r \dot{q}_n - \omega_r^2 q_n, n = 0, 1, 2, \dots \quad (2.161)$$

where,

$$a_{11} = e_r \left(c_r + \frac{\xi_r \omega_r}{\bar{\omega}_r} S_r \right)$$

$$a_{12} = e_r \frac{S_r}{\bar{\omega}_r}$$

$$a_{21} = -e_r \frac{\omega_r^2}{\bar{\omega}_r} S_r$$

$$a_{22} = e_r \left(C_r - \frac{\xi_r \omega_r}{\bar{\omega}_r} S_r \right)$$

$$b_{11} = -e_r \left[\left(\frac{(2\xi_r^2 - 1)}{\omega_r^2 h} + \frac{\xi_r}{\bar{\omega}_r} \right) \frac{S_r}{\bar{\omega}_r} + \left(\frac{2\xi_r}{\omega_r^3 h} + \frac{1}{\omega_r^2} \right) C_r \right] + \frac{2\xi_r}{\omega_r^3 h}$$

$$b_{12} = e_r \left[\left(\frac{(2\xi_r^2 - 1)}{\omega_r^2 h} \right) \frac{S_r}{\bar{\omega}_r} + \frac{2\xi_r}{\omega_r^3 h} C_r \right] + \frac{1}{\omega_r^2} - \frac{2\xi_r}{\omega_r^3 h} \tag{2.162}$$

$$b_{21} = -e_r \left[\left(\frac{(2\xi_r^2 - 1)}{\omega_r^2 h} + \frac{\xi_r}{\bar{\omega}_r} \right) \left(C_r - \frac{\xi_r \omega_r}{\bar{\omega}_r} S_r \right) - \left(\frac{2\xi_r}{\omega_r^3 h} + \frac{1}{\omega_r^2} \right) \right.$$

$$\left. (\bar{\omega}_r S_r + \xi_r \omega_r C_r) \right] - \frac{1}{\omega_r^2 h}$$

$$b_{22} = e_r \left[\frac{(2\xi_r^2 - 1)}{\omega_r^2 h} \left(C_r - \frac{\xi_r \omega_r}{\bar{\omega}_r} S_r \right) - \frac{2\xi_r}{\omega_r^3 h} (\bar{\omega}_r S_r + \xi_r \omega_r C_r) \right] + \frac{1}{\omega_r^2 h}$$

in which

$$\begin{aligned} e_r &= e^{-\xi_r \omega_r h} \\ C_r &= \cos \bar{\omega}_r h \\ S_r &= \sin \bar{\omega}_r h \end{aligned} \tag{2.163}$$

Equation (2.160) is used recursively for $n = 0, 1, 2, \dots$ with the solution at time $t = t_n$ (i.e., $q_{r,n}$ and $\dot{q}_{r,n}$) as the initial conditions for time $t = t_{n+1}$, and with the starting initial conditions at $t = 0$ from Equation (2.154) and Equation (2.155). The acceleration at $t = t_n$ is computed from Equation (2.161).

For the rigid body modes ($\omega_r = 0$), the solution is,

$$\ddot{q}_{r,n} = f_{r,n} \text{ (rigid body modes)} \tag{2.164}$$

and Equation (2.160) is used for $q_{r,n+1}$, $\dot{q}_{r,n+1}$ with the following definition for a_{ij} , b_{ij} :

$$\left. \begin{aligned} a_{11} &= 1, a_{12} = h, a_{21} = 0, a_{22} = 1 \\ b_{11} &= \frac{h^2}{3}, b_{12} = \frac{h^2}{6}, b_{21} = \frac{h}{2}, b_{22} = \frac{h}{2} \end{aligned} \right\} \text{(rigid modes)} \tag{2.165}$$

It should be noted that the solution to the modal Equation (2.153) is exact when the actual loads in $\mathbf{p}(t)$ are piecewise linear functions of time provided that the changes in slopes occur exactly at the time intervals, t_n , $n = 0, 1, \dots$

2.4.7 Frequency Response Analysis

Frequency response analysis involves computing the steady state response of a structure to a set of harmonic loads [2.12]. In NISA, this is achieved by employing the normal mode method (Section 2.4.3). The harmonic loads may be defined in terms of different amplitude and phase spectra. In general, the load vector may be represented in the form,

$$\mathbf{p}(t) = \mathbf{p}_c \cos(\Omega t) + \mathbf{p}_s \sin(\Omega t) \quad (2.166)$$

in which \mathbf{p}_c and \mathbf{p}_s are the cosine and sine components of the loads and Ω is the frequency of the forcing function. The loads are independently specified in terms of their amplitudes and phase as functions of Ω . Thus, each component of the load vector is of the form,

$$P_i(t) = P_i(\Omega) \sin[\Omega t + \Psi_i(\Omega)] \quad (2.167)$$

where,

$$p_i(\Omega) = \sqrt{p_{ci}^2(\Omega) + p_{si}^2(\Omega)} \quad (2.168)$$

and,

$$\Psi_i(\Omega) = \text{atan} \left[\frac{P_{ci}(\Omega)}{P_{si}(\Omega)} \right] \quad (2.169)$$

Equation (2.166) may be written in a concise manner by the use of complex notation as,

$$\mathbf{p}(t) = \text{Real}[\bar{\mathbf{p}} e^{i\Omega t}] \quad (2.170)$$

where, ‘Real []’ refers to the real part of the quantity in brackets and the i-th entry is given by,

$$p_i(t) = \text{Real}(\bar{p}_i) \cos(\Omega t) - \text{Imag}(\bar{p}_i) \sin(\Omega t) \quad (2.171)$$

and,

$$\text{Real}(\bar{p}_i) = p_i(\Omega) \sin[\Psi_i(\Omega)] \quad (2.172)$$

$$\text{Imag}(\bar{p}_i) = -p_i(\Omega) \cos[\Psi_i(\Omega)] \quad (2.173)$$

Thus the real and imaginary parts of the components of vector $\bar{\mathbf{p}}$ represent respectively the cosine and sine components of the corresponding term in the load vector. The use of the complex form simplifies the calculations and the real and imaginary part of the solution yield the cosine and sine components of the desired response respectively. The term 'Real' will be omitted in the subsequent equations for brevity and the real and imaginary components of a complex variable will be interpreted as the cosine and the sine components, respectively. Substituting Equation (2.170) into Equation (2.90) and transforming to modal coordinates, we obtain for the r-th mode,

$$\ddot{q}_r + 2\xi_r \omega_r \dot{q}_r + \omega_r^2 q_r = \bar{f}_r e^{i\Omega t} \quad (2.174)$$

where \bar{f}_r is complex amplitude of modal load given by,

$$\bar{f}_r = \Phi_r^T \bar{\mathbf{P}} \quad (2.175)$$

The steady state modal response is of the form,

$$q_r(t) = \bar{q}_r e^{i\Omega t} \quad (2.176)$$

which when substituted into Equation (2.174) yields,

$$(-\Omega^2 + 2\xi_r \omega_r \Omega i + \omega_r^2) \bar{q}_r e^{i\Omega t} = \bar{f}_r e^{i\Omega t} \quad (2.177)$$

or,

$$\bar{q}_r(\Omega) = H_r(\Omega) \cdot \bar{f}_r(\Omega) \quad (2.178)$$

where, $\bar{q}_r(\Omega)$, and $\bar{f}_r(\Omega)$ are complex amplitudes of $q_r(t)$ and $f_r(t)$ in Equation (2.105), and $H_r(\Omega)$ is the complex frequency response function for mode r given by,

$$H_r(\Omega) = \frac{1}{(\omega_r^2 - \Omega^2) + 2\xi_r\omega_r\Omega i} \quad (2.179)$$

In case of structural damping, the complex frequency response function takes the form,

$$H_r(\Omega) = \frac{1}{(\omega_r^2 - \Omega^2) + g_r\omega_r^2 i} \quad (2.180)$$

Equation (2.178) in the frequency domain is equivalent to Equation (2.156) given in the time domain and $H_r(\Omega)$ is the Fourier transform of the unit impulse response function $h_r(t)$.

That is,

$$H_r(\Omega) = \frac{1}{2\pi} \int_{-\infty}^{\infty} h_r(t) e^{-i\Omega t} dt \quad (2.181)$$

The physical response is recovered from the generalized modal response by modal superposition as follows:

$$\bar{\mathbf{u}} = \Phi \bar{\mathbf{q}} \quad (2.182)$$

The response in the time domain $\mathbf{u}(t)$ can be written as,

$$\mathbf{u}(t) = \text{Real}[\bar{\mathbf{u}} e^{i\Omega t}] \quad (2.183)$$

Thus each component of the displacement vector has the form,

$$u_i(t) = \text{Real}(\bar{u}_i) \cos(\Omega t) - \text{Imag}(\bar{u}_i) \sin(\Omega t) \quad (2.184)$$

or,

$$u_i(t) = |\bar{u}_i| \sin(\Omega t + \gamma_i) \quad (2.185)$$

where,

$$\gamma_i = \text{atan} \left[\frac{-\text{Real}(\bar{u}_i)}{\text{Imag}(\bar{u}_i)} \right] \quad (2.186)$$

The velocity and acceleration components are obtained from the corresponding displacement component by multiplying by Ω and Ω^2 , and they lead the displacement component by 90 and 180 degrees respectively. The stress components are obtained by the use of modal stresses (Equation (2.112)) as,

$$\bar{\sigma} = [\sigma_1, \sigma_2, \dots, \sigma_m] \bar{q} \quad (2.187)$$

where, m is the number of modes considered. In NISA, the user can obtain the responses in either the amplitude-phase format (Equation (2.185)) or the real-imaginary format (Equation (2.184)). In addition, the actual value of all responses can also be obtained for a given value of Ωt .

In addition to the computation of responses to a set of harmonic concentrated loads, pressure loads, and ground motion, NISA frequency response analysis also has a feature for computation of transfer functions. The transfer functions are dynamic equivalents of the familiar flexibility coefficients in static analysis. The transfer functions are responses given as functions of the exciting frequencies for a unit amplitude input quantity (force, displacement, velocity or acceleration) at a node (called the driver node). These functions are evaluated as follows. First, a unit amplitude harmonic force is applied at the driver node and the frequency response analysis is carried out as above. Once the responses are evaluated, the response amplitudes are divided by the amplitude of the quantity (force, displacement, velocity or acceleration at the driver node) for which the transfer functions are desired and the phase values are offset by the phase of the input quantity.

2.4.8 Random Vibration Analysis

Random vibration analysis derives the statistical measures of the response from a knowledge of the statistical properties of input random excitations [Equation (2.122)]. A random process is described in terms of its statistical characteristics known as the moments of the process usually defined with respect to zero. A process is said to be stationary when these statistical measures are independent of time. The first moment of the process $u(t)$ about zero is the mean value of the process or the average value of $u(t)$ for an ensemble representing the process given by,

$$\mu_u = E[u(t)] \tag{2.188}$$

where, $E[]$ is the mean (expected value) operator. This ensemble mean can be obtained by taking the average of all values across a set of possible time functions (called an ensemble) that may represent the random process. Another property of the process, its autocorrelation function, is defined as the average value of the product $u(t)u(t + \tau)$ for the ensemble. Provided that the process is stationary, the autocorrelation will be independent of time and will depend only on the time delay τ given by,

$$R_u(\tau) = E[u(t)u(t + \tau)] \tag{2.189}$$

For a zero time delay ($\tau = 0$), the value of the autocorrelation function gives the second moment of the process about zero or the mean square value of the process. For a zero mean process, this value is also called the variance and the square root of variance is the standard deviation or the RMS value of the process. The second moment of the process defined in the frequency domain is the mean power spectral density (PSD) of the random process and is given by the Fourier transform of the autocorrelation function as follows,

$$S_u(\Omega) = \frac{1}{2\pi} \int_{-\infty}^{\infty} R_u(\tau) e^{-i\Omega\tau} d\tau \tag{2.190}$$

Random processes with flat or constant PSDs are often called 'white noise' processes. Equation (2.190) can also be expressed by defining the autocorrelation function as the inverse Fourier transform of the PSD given by,

$$R_u(\tau) = \int_{-\infty}^{\infty} S_u(\Omega) e^{i\Omega\tau} d\Omega \quad (2.191)$$

Thus the mean square value of the process, which is the autocorrelation with a zero delay is obtained by integrating the PSD of the process as,

$$\sigma_u^2 = \int_{-\infty}^{\infty} S_u(\Omega) d\Omega \quad (2.192)$$

Also, if $S_u(\Omega)$ is the PSD function of a process, the PSD function for the derivatives of the process, namely \dot{u} and \ddot{u} are given by,

$$S_{\dot{u}}(\Omega) = \Omega^2 S_u(\Omega) \quad (2.193)$$

and

$$S_{\ddot{u}}(\Omega) = \Omega^4 S_u(\Omega) \quad (2.194)$$

The PSD as defined above is called the two-sided power spectral density function with the frequency scale from $-\infty$ to ∞ and the units of frequency in radians/unit-time. The units for the two-sided PSD is given by (the unit of the process)²/(radians/unit-time). In NISA, a one-sided power spectral density function $W(\Omega)$ with the frequency scale from 0 to ∞ and the units of frequency in cycles/unit-time is used. The two-sided PSD $S(\Omega)$ and the one-sided power spectral density $W(\Omega)$ are related by,

$$W(\Omega) = 4\pi S(\Omega) \quad (2.195)$$

If there are more than one process, then the statistical correlation between them is expressed in the frequency domain in terms of their cross PSD functions. The cross PSD is zero if the two processes are statistically independent. For NP number of processes, a PSD matrix of size NP x NP can be given in terms of their auto PSDs forming the diagonals and the cross PSDs forming the off-diagonal terms. This matrix is Hermitian, whereby the (i, j) entry is the complex conjugate of the (j, i) entry. An off-diagonal entry (i, j) gives the cross PSD

between process i and process j . The inverse Fourier transform of the cross PSD gives the cross correlation functions as follows:

$$R_{rs}(\tau) = \int_{-\infty}^{\infty} S_{rs}(\Omega) e^{i\Omega\tau} d\Omega \quad (2.196)$$

It should be noted here that the cross PSDs should be described in the complex form for their characterization of the correlation between the two processes to be complete. The cross correlation function for a zero delay ($\tau = 0$) is called the covariance between process i and process j . The matrix of variances and covariances is often called the covariance matrix.

In NISA Random Vibration Analysis, we are interested in the computation of PSDs and RMS values of responses of a linear system due to random excitations having a zero mean and defined in terms of their auto and cross PSDs. This is done using the normal mode method (Section 2.4.3). The load vector $p(t)$ in Equation (2.90) is expressed in terms of several random processes as follows.

$$p(t) = AI(t) \quad (2.197)$$

where, $I(t)$ is a vector of random processes whose auto and cross spectral densities are given by their PSD matrix $S_L(\Omega)$. Transforming to modal coordinates, the modal loads can be expressed as,

$$f_r(t) = \Phi^T AI(t) \quad (2.198)$$

This results in the modal load PSD matrix,

$$S_f(\Omega) = \Gamma^T S_L(\Omega) \Gamma \quad (2.199)$$

where, Γ is the participation factor matrix given by,

$$\Gamma = \Phi^T A \quad (2.200)$$

Using the transfer function relation given in [Equation \(2.178\)](#) the PSD matrix for the stationary modal displacements may be related to PSD matrix of the modal loads as follows,

$$\mathbf{S}_q(\Omega) = \text{diag}[H_r(\Omega)] \cdot \mathbf{S}_f(\Omega) \cdot \text{diag}[H_r^*(\Omega)] \quad (2.201)$$

where, $H_r(\Omega)$ is the complex frequency response function for the r-th mode and $H_r^*(\Omega)$ is its complex conjugate. The PSD matrices for modal velocities and modal accelerations are obtained by multiplying the above matrix by Ω^2 and Ω^4 respectively. Once the modal displacement PSD matrix has been determined, the modal covariance matrix can be computed by integrating the terms of the modal displacement PSD matrix as,

$$\sum_q = \mathbf{R}_q(0) = \int_{-\infty}^{\infty} \mathbf{S}_q(\Omega) d\Omega \quad (2.202)$$

This integral is evaluated using numerical techniques such as the Simpson or trapezoidal rule. However, closed form response expressions are available for white noise input PSDs. Covariance matrices for modal velocities and accelerations can be obtained by integrating the terms of the PSD matrices of modal velocities and modal accelerations. Finally, the PSD and root mean square (RMS) of the physical responses are obtained by applying the appropriate modal transformations. This transformation is synonymous to the transformation given in [Equation \(2.182\)](#). For example, nodal displacement PSD matrix is given by,

$$\mathbf{S}_u(\Omega) = \Phi^T \mathbf{S}_q(\Omega) \Phi \quad (2.203)$$

and the covariance matrix of nodal displacements is obtained as,

$$\sum_u = \Phi^T \sum_q \Phi \quad (2.204)$$

The PSD and covariance matrices of nodal velocities and accelerations are obtained in a similar manner. The covariance matrix for stresses is obtained using the modal stresses (see [Equation \(2.112\)](#)) as,

$$\sum \sigma = [\sigma_1, \sigma_2, \dots, \sigma_m] \sum_q [\sigma_1, \sigma_2, \dots, \sigma_m]^T \quad (2.205)$$

where, m is the number of participating modes. It should be noted here that only the diagonal terms of the above matrices are computed in NISA. The user has the option to request for auto PSDs (the diagonal terms of the PSD matrix) and the RMS values (the square root of the variances) of all responses.

In addition, (i) zero mean crossing rate (also known as the apparent frequency), (ii) shape factor, (iii) extreme value of the response quantity α with user specified probability equal to p in the time interval T and (iv) expected value of the number of crossings of a level α with a positive slope in the time interval T can be obtained. Zero mean crossing rate is the expected mean rate at which the response is crossing its zero value and it may be determined from its PSD as,

$$\eta_{u_o}^+ = \frac{1}{2\pi} \sqrt{\frac{\lambda_2}{\lambda_0}} \quad (2.206)$$

where,

$$\lambda_j = \int_0^\infty \omega^j S_X(\Omega) d\Omega \quad (2.207)$$

The above integration is performed for each of the responses whose zero mean crossing rates are needed by using the trapezoidal rule. Shape factor of a random process is given by,

$$k = \sqrt{2\pi(1 - \lambda_1^2 / (\lambda_0 \lambda_2))} \quad (2.208)$$

Extreme value of the response quantity α with probability equal to p in the time interval T is given by,

$$\alpha = \sqrt{-\ln\left\{-\ln(p)\frac{2\pi\sigma_{\dot{x}}}{T\sigma_x}\right\}}2\sigma_x^2 \quad (2.209)$$

Expected value of the number of crossings of a level defined by α with a positive slope in the time interval T given by,

$$N(\alpha;T) = \frac{T}{2\pi} \frac{\sigma_{\dot{x}}}{\sigma_x} \exp\left\{-\frac{\alpha^2}{2\sigma_x^2}\right\} \quad (2.210)$$

2.4.9 Shock Spectrum Analysis

The shock spectrum analysis is also known as response spectrum analysis. This analysis is an efficient alternative to transient dynamic analysis for estimating the maximum response under support excitations. The solution of [Equation \(2.153\)](#) for viscous underdamped system under ground motion $w_j(t)$ in direction j is (see [Section 2.4.5](#) and [Equation \(2.156\)](#)),

$$q_{rj}(t) = \frac{\Gamma_{rj}}{\omega_r} \int_0^t \ddot{w}_j(\tau) e^{-\xi_r \omega_r (t-\tau)} \sin[\bar{\omega}_r (t-\tau)] d\tau \quad (2.211)$$

The peak or maximum response is given by,

$$(q_{rj})_{max} = \Gamma_{rj} S_{dj} \quad (2.212)$$

in which S_{dj} is the spectral displacement due to the support excitation given by,

$$S_{dj} = \max \text{ of all } t \left[\frac{1}{\omega_r} \int_0^t \ddot{w}_j(\tau) e^{-\xi_r \omega_r (t-\tau)} \sin[\bar{\omega}_r (t-\tau)] d\tau \right] \quad (2.213)$$

S_{dj} is the maximum response of a single degree of freedom (SDOF) system for the given support excitation. As it can be readily seen, this value is a function of the natural frequency and damping only. A shock (or response) spectrum curve for a certain value of damping may be defined as the maximum responses of all such SDOF systems with the given damping and plotted as a function of natural frequency. Similarly the maximum acceleration S_a , and maximum velocity S_v , may also be determined and are termed as spectral acceleration and spectral velocity respectively. The three spectral quantities are approximately related as follows,

$$S_{dj} \approx \frac{S_{vj}}{\omega_r} \approx \frac{S_{aj}}{\omega_r^2} \quad (2.214)$$

The spectral value of the physical response due to r mode according to Equation (2.102) is,

$$(u_{rj})_{max} = \phi_r(q_{rj})_{max} \quad (2.215)$$

Unlike in transient dynamic analysis, the contributions of the physical responses for each of the modes cannot be directly summed to obtain the total response. This is because of the fact that the maxima for each mode occur at different times, and the information on the time of maxima is not available in the shock spectra. Reasonable estimates of the maxima may be obtained by using one of the following modal combination methods available in NISA:

(a) ABS or PEAK

The absolute sum of the modal responses given by

$$R_{tot} = \sum_{r=1}^m |R_r| \quad (2.216)$$

in which R_r is the physical response due to mode r and m is the number of modes considered. This method is conservative and is used if the natural frequencies are closely spaced (within 10 percent of each other) and/or when damping is large.

(b) SRSS or RMS

The square root of the sum of the squares of the modal response and is given by,

$$R_{tot} = \sqrt{\sum_{r=1}^m R_r^2} \quad (2.217)$$

This method is applicable if the modes are statistically independent which is the case when the natural frequencies are far apart and/or when damping is small.

(c) CQC

The complete quadratic combination [2.45] is given by the following formula:

$$R_{tot} = \sqrt{\sum_{r=1}^m \sum_{s=1}^m R_r \cdot R_s \cdot P_{rs}} \quad (2.218)$$

in which

$$P_{rs} = \frac{8\sqrt{\xi_r \xi_s} (\xi_r + \gamma \xi_s) \gamma^{3/2}}{(1 - \gamma^2) + 4\xi_r \xi_s (1 + \gamma^2) + 4\gamma^2 (\xi_r^2 + \xi_s^2)} \quad (2.219)$$

where, ξ_r and ξ_s are modal damping ratios and $\gamma = \omega_s / \omega_r$. This method encompasses the SRSS and ABS procedures for $\xi_r = \xi_s$. When $\gamma = 0$, the CQC method reduces to the former and when $\gamma = 1$ to the latter.

(d) NRL or PRMS

The absolute maxima of the modal responses is added to the square root of the sum of the squares of the remaining modal responses as follows:

$$R_{max} = |R_j| + \sqrt{\sum_{r=1, r \neq j}^m R_r^2} \quad (2.220)$$

where,

$$R_j = \max \text{ of all } |R_r| \tag{2.221}$$

The above superposition rules are employed to get the response maxima due to ground excitation in one direction. Similarly the maximum responses due to ground motions in other directions are computed separately. These maxima are then superimposed by using the SRSS or the ABS procedures to get the total response.

(e) Grouping Method (GRP) (USNRC Regulatory guide 1.92)

The grouping method combines the modal responses by the following formula:

$$R_{tot} = \left[\sum_{r=1}^m R_r^2 + \sum_{q=1}^p \sum_{l=i}^j \sum_{s=i}^j |R_{lq} R_{sq}| \right]^{1/2} \quad l \neq s \tag{2.222}$$

where, R_{lq} , R_{sq} - Peak modal responses R_l , R_s of modes l and s within q^{th} group, respectively.

i - number of the mode where group starts.

j - number of the mode where group ends.

p - number of groups of closely spaced modes.

This method is used when modes are closely spaced. The closely spaced modes are divided into groups that include all modes having frequencies lying between lowest frequency in the group and a frequency of 10% higher. No one frequency is to be in more than one group.

(f) Ten Percent Method (TPM) (USNRC Regulatory guide 1.92)

The Ten Percent Method combines modal responses by the following formula:

$$R_{tot} = \left[\sum_{r=1}^m R_r^2 + 2 \sum |R_i R_j| \right]^{1/2} \quad i \neq j \tag{2.223}$$

The second summation in the equation is to be done on all i and j modes whose frequencies are closely spaced to each other. Let ω_i and ω_j be the frequencies of the i^{th} and j^{th} mode. In order to verify which of the modes are closely spaced, the following equation will apply:

$$\frac{\omega_j - \omega_i}{\omega_i} \leq 0.1, \quad 1 \leq i \leq j \leq m \quad (2.224)$$

(g) Double Sum Method (DSM) (USNRC Regulatory guide 1.92)

The Double Sum Method combines the modal responses by the following formula:

$$R_{tot} = \left[\sum_{k=1}^m \sum_{s=1}^m |R_k R_s| \varepsilon_{ks} \right]^{1/2} \quad (2.225)$$

R_k is the peak value of the response of the element due to the k^{th} mode, R_s is the peak value of the response due to s^{th} mode and m is the number of modes considered in the modal response combination,

$$\varepsilon_{ks} = \left[1 + \left\{ \frac{\omega'_k - \omega'_s}{\beta'_k \omega_k + \beta'_s \omega_s} \right\}^2 \right]^{-1} \quad (2.226)$$

in which

$$\omega'_k = \omega_k [1 - \beta_k^2]^{1/2} \quad (2.227)$$

$$\beta'_k = \beta_k + \frac{2}{t_D \omega_k} \quad (2.228)$$

where, ω_k and β_k are the modal frequency and the damping ratio in the k^{th} mode respectively, and t_D is the duration of the earthquake.

For the multiple support excitation case, the peak estimate of the “Quasi” static component is obtained from the [Equation \(2.139\)](#) as,

$$\left\{ \max_{0 < t < \infty} |u_{qs}(t)| \right\} = L \left\{ \lim_{\omega_n \rightarrow \infty} [\mathbf{S}_a(\omega_n, \xi_n) / \omega_n^2] \right\} = L \{ \mathbf{S}_d(0) \} \quad (2.229)$$

In the above form, $\mathbf{S}_d(0)$ represents the vector of zero frequency displacement (ZFD) of the support motions.

Peak estimates of $\mathbf{u}(t)$ is now obtained for different components of support excitation from NISA/Shock spectrum analysis using any of the modal combination rules. The total response is now obtained from Equation (2.142) by summing over the components according to the user's choice of ABS or SRSS combination rules.

Envelope Spectrum Method

Envelope Spectrum method is one method described in ASCE 4-98 [Ref. 2.62] for the case of multiple-support excitation. In this method, a single response spectrum that envelopes user specified input spectra of one direction is used as a uniform excitation. Thus the method is equivalent to the case of uniform base excitation. In this method, the influence of the phase characteristics of the response spectra representing non-uniform excitation is neglected.

2.4.10 Direct Transient Analysis

As shown in Section 2.4.1, the governing differential equations of motion can be written as,

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{p}(t) \quad (2.230)$$

where, \mathbf{M} , \mathbf{C} , and \mathbf{K} are respectively the global mass, damping and stiffness matrices; $\mathbf{p}(t)$ is the time-dependent applied force vector and $\ddot{\mathbf{u}}$, $\dot{\mathbf{u}}$ and \mathbf{u} are the nodal acceleration, nodal velocity and nodal displacement vectors, respectively. The corresponding initial conditions can be written as,

$$\mathbf{u}(0) = \mathbf{d}_o \quad (2.231)$$

and

$$\dot{\mathbf{u}}(0) = \mathbf{v}_0 \quad (2.232)$$

where, \mathbf{d}_0 and \mathbf{v}_0 are the given initial displacement and velocity vectors, respectively. To determine the response due to specified time dependent loads, the modal transient (mode-superposition) method outlined in [Section 2.4.6](#) may be utilized. Direct transient analysis is an alternative to modal superposition and is based on direct integration of the equations of motion, i.e., no prior transformation or eigenvalue analysis is required.

In NISA, Newmark method is used for direct transient analysis, which can be summarized by the following equations:

$$\mathbf{M}\ddot{\mathbf{u}}_{n+1} + \mathbf{C}\dot{\mathbf{u}}_{n+1} + \mathbf{K}\mathbf{u}_{n+1} = \mathbf{p}_{n+1} \quad (2.233)$$

$$\mathbf{u}_{n+1} = \mathbf{u}_n + \Delta t \dot{\mathbf{u}}_n + \left(\frac{1}{2} - \beta\right) \Delta t^2 \ddot{\mathbf{u}}_n + \beta \Delta t^2 \ddot{\mathbf{u}}_{n+1} \quad (2.234)$$

$$\dot{\mathbf{u}}_{n+1} = \dot{\mathbf{u}}_n + (1 - \gamma) \Delta t \ddot{\mathbf{u}}_n + \gamma \Delta t \ddot{\mathbf{u}}_{n+1} \quad (2.235)$$

where, \mathbf{u}_n , $\dot{\mathbf{u}}$ and $\ddot{\mathbf{u}}$ are the known solutions for \mathbf{u} , $\dot{\mathbf{u}}$ and $\ddot{\mathbf{u}}$ at time step n , respectively.

The parameters β and γ determine the stability and accuracy characteristics of the algorithm. In NISA, the Newmark method is implemented with displacement form. The predictors are defined as,

$$\hat{\mathbf{u}}_{n+1} = \dot{\mathbf{u}}_n + (1 - \gamma) \Delta t \ddot{\mathbf{u}}_n \quad (2.236)$$

$$\hat{\mathbf{u}}_{n+1} = \mathbf{u}_n + \Delta t \dot{\mathbf{u}}_n + \left(\frac{1}{2} - \beta\right) \Delta t^2 \ddot{\mathbf{u}}_n \quad (2.237)$$

As can be seen from the equation, $\hat{\mathbf{u}}_{n+1}$ and $\hat{\dot{\mathbf{u}}}_{n+1}$ are defined with displacement, velocity and acceleration from the previous step. With these and [Equation \(2.234\)](#), [Equation \(2.235\)](#), the acceleration and the velocity of the current time step can be written as,

$$\hat{\mathbf{w}}_{n+1} = \frac{1}{\beta\Delta t^2}\hat{\mathbf{u}}_{n+1} - \frac{1}{\beta\Delta t^2}\hat{\mathbf{u}}_{n+1} \quad (2.238)$$

$$\dot{\mathbf{u}}_{n+1} = \hat{\mathbf{u}}_{n+1} - \frac{\gamma}{\beta\Delta t}\hat{\mathbf{u}}_{n+1} + \frac{\gamma}{\beta\Delta t}\hat{\mathbf{u}}_{n+1} \quad (2.239)$$

Substituting these into [Equation \(2.233\)](#), we have the following recursive equation to solve for displacement \mathbf{u}_{n+1} :

$$\left(\frac{1}{\beta\Delta t^2}\mathbf{M} + \frac{\gamma}{\beta\Delta t}\mathbf{C} + \mathbf{K}\right)\mathbf{u}_{n+1} = \mathbf{p}_{n+1} + \left(\frac{1}{\beta\Delta t^2}\mathbf{M} + \frac{\gamma}{\beta\Delta t}\mathbf{C}\right)\hat{\mathbf{u}}_{n+1} - \mathbf{C}\hat{\mathbf{w}}_{n+1} \quad (2.240)$$

By defining the effective stiffness matrix,

$$\mathbf{K}^* = \frac{1}{\beta\Delta t^2}\mathbf{M} + \frac{\gamma}{\beta\Delta t}\mathbf{C} + \mathbf{K} \quad (2.241)$$

and the effective load vector,

$$\mathbf{p}^* = \mathbf{p}_{n+1} + \left(\frac{1}{\beta\Delta t^2}\mathbf{M} + \frac{\gamma}{\beta\Delta t}\mathbf{C}\right)\hat{\mathbf{u}}_{n+1} - \mathbf{C}\hat{\mathbf{u}}_{n+1} \quad (2.242)$$

then, [Equation \(2.240\)](#) becomes,

$$\mathbf{K}^*\mathbf{u}_{n+1} = \mathbf{p}^* \quad (2.243)$$

which is in the form of the familiar matrix equation for static analysis that can be solved for the displacement vector \mathbf{u}_{n+1} . [Equation \(2.238\)](#) and [Equation \(2.239\)](#) can then be used to solve for acceleration and velocity. Note that the effective stiffness matrix \mathbf{K}^* , which is a combination of global mass, damping and stiffness matrices, is the same for every time step if the time step size does not change. Therefore, the effective stiffness matrix need to be factorized only once if the same time step size is used through out the whole analysis. In NISA, in order to take advantage of this, the time step size within an event is constant.

However, the time step size can be changed from one event to another, but this also means the effective stiffness matrix has to be factorized again.

For the adaptive time stepping scheme (INCREMENTS = ADAPTIVE, n) of linear direct integration, the factorization of the stiffness matrix is necessarily performed at each time step. The scheme is based on minimization of error norm derived from the ordinary differential equation (ODE) of the local error. The local error is computed as the approximate solution of an error ODE, obtained by subtracting a member of a one-parameter family of Newmark ODE from the original ODE [Equation \(2.230\)](#).

One-parameter family of Newmark ODE corresponding to [Equation \(2.230\)](#) over $t \in [t_i, t_{i+1}]$ is,

$$\ddot{\mathbf{u}} = \beta \mathbf{V}_i + (1 - \beta) \mathbf{V}_{i+1} \tag{2.244}$$

$$\tag{2.245}$$

where, $\mathbf{V}_i = \ddot{\mathbf{u}}_i$, and $\mathbf{V}_{i+1} = \ddot{\mathbf{u}}_{i+1}$, the acceleration fields at previous time step t_i and current time step t_{i+1} respectively.

Now the error ODE is obtained as,

$$\ddot{\mathbf{E}}(t) = \mathbf{V}(t) - \{\beta \mathbf{V}_i + (1 - \beta) \mathbf{V}_{i+1}\} \tag{2.246}$$

where $\mathbf{V}(t)$ is the acceleration at any time t . Integrating [Equation \(2.246\)](#) twice over the time interval (t_i, t_{i+1}) , the solution is given by,

$$\dot{E}(t) = E(t_i) + \int_{t_i}^t V(s_1) ds_1 - \{\beta V_i + (1 - \beta)V_{i+1}\}(t - t_i) \quad (2.247)$$

$$E(t) = E(t_i) + \dot{E}(t_i)(t - t_i) + \int_{t_i}^t \int_{t_i}^s V(s_1) ds_1 ds_2 - \{\beta V_i + (1 - \beta)V_{i+1}\} \left(\frac{(t - t_i)^2}{2} \right)$$

Now the local error norm at the end of $t \in [t_i, t_{i+1}]$ is given by $\|E_{i+1}\| + h\|\dot{E}_{i+1}\|$.

Now, with $E(t_i) = \dot{E}(t_i) = 0$ and assuming linear interpolation over $t = [t_i, t_{i+1}]$ for $V(t)$ (Figure 2.4),

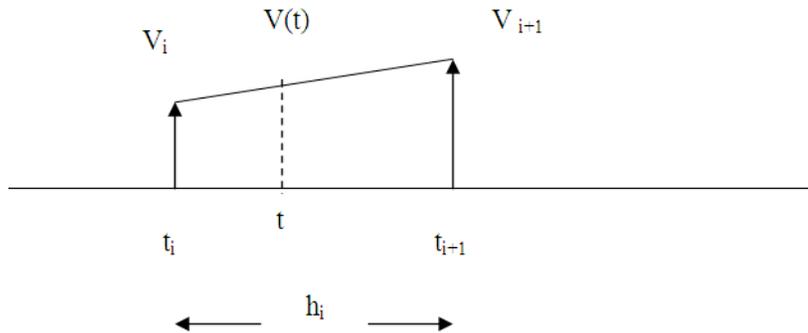


Figure 2.4: Assumed linear variation of acceleration over $t = [t_i, t_{i+1}]$

An expression for the error norm is obtained as,

$$\text{Error norm (ERRTOL)} \leq C(\beta)h_i^2 \left[\|V_i\| + \|V_{i+1}\| \right] \quad (2.248)$$

Assuming at this stage that the two norms $\|V_i\|$ and $\|V_{i+1}\|$ at the ends of the small interval are equal, the bound on the error norm is obtained as,

$$\text{Error norm} \leq 2C(\beta)\|V_i\|h_i^2 \quad (2.249)$$

If ‘ERRTol’ is the tolerance specified by user.

$$h_i \leq \sqrt{\frac{ERRTol}{2C(\beta)\|V_i\|}} \quad (2.250)$$

At the end of each time step, the step size h_i is obtained from [Equation \(2.250\)](#).

2.4.11 Direct Frequency Response Analysis

Direct frequency response analysis is an alternative to modal superposition and is based on transformation of the equations of motion directly into frequency domain and hence no eigenvalue analysis is required. As in the frequency response analysis with modal superposition, in direct frequency response analysis also the aim is to obtain the steady state response of the structure. As shown in [Section 2.4.1](#), the governing differential equations of motion can be written as,

$$M\ddot{\mathbf{u}} + C\dot{\mathbf{u}} + K\mathbf{u} = \mathbf{p}(t) \quad (2.251)$$

where, \mathbf{M} , \mathbf{C} , and \mathbf{K} are respectively the global mass, damping and stiffness matrices; $\mathbf{p}(t)$ is the time-dependent applied force vector and $\ddot{\mathbf{u}}$, $\dot{\mathbf{u}}$ and \mathbf{u} are the nodal acceleration, nodal velocity and nodal displacement vectors, respectively. In frequency domain, [Equation \(2.251\)](#) can be written as,

$$M\ddot{\mathbf{u}}(\Omega) + C\dot{\mathbf{u}}(\Omega) + K\mathbf{u}(\Omega) = \mathbf{p}(\Omega) \quad (2.252)$$

$\mathbf{u}(\Omega)$ and $\mathbf{p}(\Omega)$ are the complex functions of frequency, Ω , representing the response and load vectors respectively. Using the properties of Fourier transformation, [Equation \(2.252\)](#) can be rewritten as,

$$(-\Omega^2 M + i\Omega C + K)u(\Omega) = p(\Omega) \quad (2.253)$$

where, D^* , given by $(-\Omega^2 M + i\Omega C + K)$, the dynamic stiffness matrix, is a complex symmetric matrix. The steady state displacements $u(\Omega)$ can be obtained by solving the simultaneous set of linear [Equation \(2.253\)](#).

In case of support excitation, the equations of motion are shown in equation (2.135) and the partitioned matrices are shown in equation (2.136). With the help of this partitioning, the equations of motion in steady state can be written in the form:

where

$u_g(\Omega)$ is a vector containing the specified support displacements

$u(\Omega)$ is a vector containing the unknown nodal displacements

2.4.12 Ground Motion

The equations of motion for a system subjected to ground excitations in terms of translations and rotations (may not be identical at all support points) have the form K^* and $u(\Omega)$ in [Equation \(2.252\)](#) is partitioned as follows in case of non-uniform /uniform base excitation.

$$\left[\begin{array}{cc} K_{11} & K_{12} \\ K_{21} & K_{22} \end{array} \right] - \omega^2 \left[\begin{array}{cc} M_{11} & M_{12} \\ M_{21} & M_{22} \end{array} \right] + i \left[\begin{array}{cc} C_{11} & C_{12} \\ C_{21} & C_{22} \end{array} \right] \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix} \quad (2.254)$$

where

u_2 is a vector containing the specified displacements

u_1 is a vector containing the nodal displacements

Expanding Equation (2.254)

$$[K_{11} - \omega^2 M_{11} + i\omega C_{11}]\{u_1\} = \{f_1\} - [K_{12} - \omega^2 M_{12} + i\omega C_{12}]\{u_2\} \quad (2.255)$$

Equation (2.255) is solved for u_1

2.5 Buckling Analysis

The objective of the buckling analysis is to estimate the maximum load that a structure can support before it becomes elastically unstable or before it collapses. The analysis here is actually a 'bifurcation buckling'.

This type of bifurcation or eigenvalue buckling is usually limited and nonconservative in its applications. Most practical problems with buckling behavior should be analyzed as large displacement geometrically nonlinear problems. The eigenvalue or bifurcation buckling in NISA involves calculating the points at which the structure's primary load deflection path, which is a straight line for linear static analysis, is bifurcated by a secondary load deflection path. This requires an eigenvalue extraction and the governing equation is given by,

$$(\mathbf{K} + \lambda_i \mathbf{K}_g) \mathbf{u}_i = \mathbf{o} \quad (2.256)$$

where, \mathbf{K} is the linear stiffness matrix, \mathbf{K}_g is the geometric or initial stress stiffness matrix (identified as $\mathbf{K}^{(2)}$ in nonlinear analysis, [Section 2.3](#), [Equation \(2.61\)](#)), λ_i is the i -th eigenvalue or the multiplier to the load from which \mathbf{K}_g is calculated and \mathbf{u}_i is the i -th displacement eigenvector or mode shape.

Buckling analysis in NISA is a two-pass analysis. The first pass is a static analysis which determines the stress for a given reference set of loads. The second pass is an eigenvalue analysis which first computes the geometric stiffness matrix as given by [Section 2.59](#), repeated here for reference,

$$\mathbf{K}_g = \int_V \mathbf{B}_{n/2}^T \hat{\mathbf{S}} \mathbf{B}_{n/2} dV \quad (2.257)$$

where, all the terms are defined in [Section 2.3.3](#). The stresses in Eqn. (2.5-2) will be based on the results of the linear static analysis pass. The second pass results are in terms of load factors (eigenvalues) and mode shapes (eigenvectors).

The eigenvalue extraction methods for buckling analysis are the same as those available for vibration eigenvalue analysis. Refer to [Section 2.4](#) and [Section 3.5.2](#) for a discussion on the choice of method. The inverse iteration method or either of the subspace algorithms can be

employed to extract the load factors. Lanczos method is not recommended for buckling analysis. The load factors obtained from buckling analysis may be negative, indicating that buckling loads are obtained when the reference loads are reversed in direction and scaled up by the load factors. The available eigenvalue extraction methods in NISA can handle negative eigenvalues (load factors). Negative load factors may be obtained in the inverse iteration method by adopting suitable lower and upper cut-off limits. However, for the subspace iteration methods, it is recommended that a negative initial shift smaller than the lowest eigenvalue (largest negative eigenvalue) be specified in cases where negative load factors are suspected to be present. The buckling analysis also has Sturm sequence check capability, in which all the load factors below a user specified upper limit may be calculated or reported in number.

It should be noted that this type of bifurcation buckling or initial elastic stability analysis does not account for initial imperfections, provides no information about post-buckling behavior and is limited to cases where the initial displacement or large deformation stiffness matrix is negligible. Such cases are very limited in practical application. To be able to neglect the effect of large displacements, bending stresses have to be small compared to membrane stresses and the loading should be non-follower, i.e. does not follow the deformed or buckled geometry.

In order to do a proper post-buckling behavior (e.g., snap-through analysis), a full incremental nonlinear static analysis should be performed taking into account both the initial stress and initial displacement stiffness matrices, and applying the load incrementally. If at any load step, the tangential stiffness matrix becomes singular, static instability is predicted.

2.6 Heat Transfer Analysis

2.6.1 Introduction and Basic Equations

Consider a three-dimensional body; [Figure 2.5](#), of volume V and bounded by a surface area S . We assume that the material obeys Fourier's law of heat conduction

$$\mathbf{q} = -\mathbf{K} \cdot \frac{\partial \mathbf{T}}{\partial \mathbf{x}} \quad (2.258)$$

or in component form,

$$\begin{Bmatrix} q_x \\ q_y \\ q_z \end{Bmatrix} = - \begin{bmatrix} k_{xx} & k_{xy} & k_{xz} \\ k_{yx} & k_{yy} & k_{yz} \\ k_{zx} & k_{zy} & k_{zz} \end{bmatrix} \begin{Bmatrix} \partial T / \partial x \\ \partial T / \partial y \\ \partial T / \partial z \end{Bmatrix} \quad (2.259)$$

where, q_i is the rate of heat flow conducted per unit area in the direction i , \mathbf{K} is the thermal conductivity tensor for the material ([Equation \(2.258\)](#) assumes anisotropic material behavior), and $\partial \mathbf{T} / \partial \mathbf{x}$ is the temperature gradient vector in Cartesian coordinates.

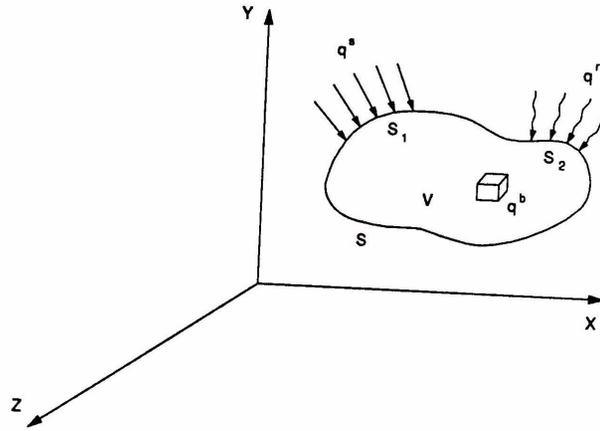


Figure 2.5: Three dimensional conduction in a body

In the material principal directions, Equation (2.259) reduces to the diagonal form

$$\begin{Bmatrix} q_1 \\ q_2 \\ q_3 \end{Bmatrix} = - \begin{bmatrix} k_{11} & 0 & 0 \\ 0 & k_{22} & 0 \\ 0 & 0 & k_{33} \end{bmatrix} \begin{Bmatrix} \partial T / \partial x_1 \\ \partial T / \partial x_2 \\ \partial T / \partial x_3 \end{Bmatrix} \quad (2.260)$$

or simply,

$$q_n = -k_n \frac{\partial T}{\partial n} \quad (2.261)$$

The thermal conductivity matrix in Equation (2.259) may be obtained from that in Equation (2.260) by simple second order tensor transformation as,

$$\mathbf{K}_{x,y,z} = \mathbf{T}^T \mathbf{K}_{1,2,3} \mathbf{T} \quad (2.262)$$

where, \mathbf{T} is the second order transformation tensor between the two sets of axes x, y, z and $1,2,3$. It is noted that the principle of irreversible thermodynamics shows that the tensor \mathbf{K} has to be symmetric. If the material is homogeneous and isotropic, then,

$$\begin{aligned} k_{xy} &= k_{yz} = k_{zx} = 0, \text{ and} \\ k_{xx} &= k_{yy} = k_{zz} = k \end{aligned} \quad (2.263)$$

The heat conduction equation or the law of conservation of energy is obtained by considering the heat flow equilibrium in the interior of the body and it gives,

$$\left[\frac{\partial}{\partial x} \left(k_x \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left(k_y \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left(k_z \frac{\partial T}{\partial z} \right) \right] + q^b = \rho c \frac{\partial T}{\partial t} \quad (2.264)$$

where, k_x, k_y, k_z are the same as k_{xx}, k_{yy}, k_{zz} , respectively, q^b is the rate of heat generated per unit volume, t is the time, ρ is the mass density of the material and c is the specific heat (where in the case of conduction in solids, no distinction is made between the specific heat at constant pressure, c_p and the specific heat at constant volume, c_v).

Equation (2.264) is the general equation for heat conduction in solids and it may take one of the following special cases:

- If the material is isotropic with constant conductivity, k , the equation reduces to,

$$\nabla^2 T + q^b/k = \frac{1}{\alpha} \frac{\partial T}{\partial t} \quad (2.265)$$

where, $\alpha = k/\rho c$ is the thermal diffusivity

- If the material is isotropic with constant conductivity, k , and steady state conditions prevail,

$$\nabla^2 T + q^b/k = 0 \quad (2.266)$$

which is Poisson's equation

- If the material is isotropic with constant conductivity, k , and no heat generation q^b ,

$$\nabla^2 T = \frac{1}{\alpha} \frac{\partial T}{\partial t} \quad (2.267)$$

which is the diffusion equation.

- If the material is isotropic, with constant conductivity, k , no heat generation \dot{q}^b , and steady state conditions exist,

$$\nabla^2 T = 0 \quad (2.268)$$

which is Laplace's equation.

In the above equations,

$$\nabla = \frac{\partial}{\partial x} \mathbf{e}_x + \frac{\partial}{\partial y} \mathbf{e}_y + \frac{\partial}{\partial z} \mathbf{e}_z \quad (2.269)$$

where, \mathbf{e}_x , \mathbf{e}_y and \mathbf{e}_z are the unit vectors in x , y and z directions, respectively, and

$$\nabla^2 = \nabla \cdot \nabla = \left(\frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2} + \frac{\partial^2}{\partial z^2} \right) \quad (2.270)$$

Equation (2.266), Equation (2.267), and Equation (2.268) may be generally subjected to one or more of the following boundary conditions (B. C.);

- (a) Prescribed temperature or Dirichlet B.C.:

$$T(x, t) = T_s(x, t) \quad \text{on } \mathbf{S}_1, \mathbf{S}_1 \subset \mathbf{S} \quad (2.271)$$

- (b) Prescribed heat flow or flux (Cauchy's or Neumann B.C.):

$$-k_n \left. \frac{\partial T}{\partial \mathbf{n}} \right|_{s=s_2} = q^s, \quad \mathbf{S}_2 \subset \mathbf{S} \quad (2.272)$$

- (c) No heat flow or adiabatic B.C.:

$$\left. \frac{\partial T}{\partial \mathbf{n}} \right|_{s=s_3} = 0, \quad \mathbf{S}_3 \subset \mathbf{S} \quad (2.273)$$

which is the natural boundary condition and is a special case of the Neumann or Cauchy B.C.

- (d) Convective heat exchange (Convection B.C.):

When part of the surface of the body, say \mathbf{S}_c where ($\mathbf{S}_c \subset \mathbf{S}$), is in contact with a fluid media with a fluid temperature, T_f , the rate of heat exchange between the body and the fluid is given by Fourier's law in the form,

$$-k \frac{\partial T}{\partial \mathbf{n}} = h(T_s - T_f) \quad (2.274)$$

where, T_s is the body surface temperature, T_f is the fluid temperature, n is the normal to the surface \mathbf{S}_c and h is the convective heat transfer coefficient. The convective heat transfer coefficient, h , may be a function of temperature or time and the fluid temperature, T_f , may be a function of time.

(e) Radiation boundary condition:

Assume a body of surface temperature T_s , is enclosed within a space with wall temperature, T_e (only part of the body surface, say S_r may be considered exposed to radiation from another body or surface with wall temperature T_e). We only consider gray bodies or surfaces for which the emissive power is independent of the radiation wavelength. Under the above assumptions, the radiant energy emitted from the surface of the body per unit time and area is given by q^r ,

$$q^r = \varepsilon \sigma T_s^4 \quad (2.275)$$

where, ε is the emissivity of the surface, σ is the Stefan-Boltzmann constant and T_s is the absolute temperature of the surface. On the other hand, the radiant energy absorbed by the surface due to the wall temperature T_e is given by q^a ,

$$q^a = \alpha \sigma T_e^4 \quad (2.276)$$

where, α is the absorptivity of the surface. Owing to Kirchoff's law, $\alpha = \varepsilon$ and the net heat flux out of the surface is given by q ,

$$q = \varepsilon \sigma [T_s^4 - T_e^4] = -k_n \frac{\partial T}{\partial n} \quad (2.277)$$

Note that ε may be function of temperature, surface finish and view factor between the two bodies.

It is convenient to put [Equation \(2.277\)](#) into the following form which is similar to the convective boundary condition form,

$$q = \kappa (T_s - T_e) \quad (2.278)$$

where,

$$\kappa = \varepsilon \sigma (T_s^2 + T_e^2)(T_s + T_e) \quad (2.279)$$

is an equivalent radiation coefficient which is strongly dependent on temperature.

The functional governing the heat conduction (Equation (2.266), Equation (2.267), and Equation (2.268)) and its corresponding boundary conditions is given by:

$$\begin{aligned} \Pi = \int_V \frac{1}{2} \left\{ k_{xx} \left(\frac{\partial T}{\partial x} \right)^2 + k_{yy} \left(\frac{\partial T}{\partial y} \right)^2 + k_{zz} \left(\frac{\partial T}{\partial z} \right)^2 - 2T q^b \right\} dV \\ - \int_s T^s q^s ds + \int_V T \left(\rho c \frac{\partial T}{\partial t} \right) dV - \sum_i T^i q^i \end{aligned} \quad (2.280)$$

where, q^s is assumed to include all types of surface heat flow, (i.e., prescribed, due to convection and due to radiation), q^i are the concentrated heat flow input at specific points with temperature T^i . Note that the prescribed temperature boundary condition (Dirichlet B.C.) do not alter the form of the functional Π . Using the condition of stationary Π , we obtain:

$$\begin{aligned} \int_V \delta \left(\frac{\partial T}{\partial x} \right)^T K \left(\frac{\partial T}{\partial x} \right) dV + \int_V \delta T \left(\rho c \frac{\partial T}{\partial t} \right) dV \\ = \int_V \delta T q^b dV + \int_s \delta T^s q^s ds + \sum_i \delta T^i q^i \end{aligned} \quad (2.281)$$

where, δ denotes a “variation in” and,

$$\left(\frac{\partial T}{\partial x} \right) = \left[\frac{\partial T}{\partial x} \quad \frac{\partial T}{\partial y} \quad \frac{\partial T}{\partial z} \right]^T \quad (2.282)$$

and K is given by,

$$\mathbf{K} = \begin{bmatrix} k_{xx} & 0 & 0 \\ 0 & k_{yy} & 0 \\ 0 & 0 & k_{zz} \end{bmatrix} \quad (2.283)$$

2.6.2 Finite Element Equations:

Starting from Equation (2.281) and following the usual finite element variational procedure, or starting from Equation (2.264) and the boundary conditions in Equation (2.271) to Equation (2.279) and following the Galerkin finite element procedures, we introduce the following temperature assumption:

$$\begin{aligned} \mathbf{T}(x) &= \sum_{i=1}^n N_i(x) \bar{T}_i, \\ &\equiv \mathbf{N} \bar{\mathbf{T}} \end{aligned} \quad (2.284)$$

$$\begin{aligned} \frac{\partial}{\partial x} \mathbf{T}(x) &= \sum_{i=1}^n \frac{\partial N_i(x)}{\partial x} \bar{T}_i \\ &\equiv \mathbf{B} \bar{\mathbf{T}} \end{aligned} \quad (2.285)$$

where, i ranges from 1 to the number of nodes in the element and $\bar{\mathbf{T}}$ are the element nodal temperatures. The final finite element equation for heat conduction analysis with convection and/or radiation boundary conditions becomes,

$$\mathbf{C} \dot{\mathbf{T}} + (\mathbf{K}_c + \mathbf{K}_h + \mathbf{K}_r) \mathbf{T} = \mathbf{Q}^b + \mathbf{Q}^s + \mathbf{Q}^h + \mathbf{Q}^r \quad (2.286)$$

where, the bar is dropped from the nodal temperatures, and the element matrices and flux vectors are given by,

$$\mathbf{C} = \int_V \rho c \mathbf{N}^T \mathbf{N} dV \quad (2.287)$$

is the thermal capacity matrix,

$$\mathbf{K}_c = \int_V \mathbf{B}^T \mathbf{K} \mathbf{B} dV \quad (2.288)$$

is the thermal conductivity matrix,

$$\mathbf{K}_h = \int_S \mathbf{h} \mathbf{N}^T \mathbf{N} dS \quad (2.289)$$

is an additional thermal conductivity matrix due to convection B.C.,

$$\mathbf{K}_r = \int_S \kappa \mathbf{N}^T \mathbf{N} dS \quad (2.290)$$

is an additional thermal conductivity matrix due to radiation B.C.,

$$\mathbf{Q}^b = \int_V \mathbf{N}^T \mathbf{q}^b dV \quad (2.291)$$

is the heat flux vector due to internal heat generation,

$$\mathbf{Q}^s = \int_S \mathbf{N}^T \mathbf{q}^s dS \quad (2.292)$$

is the heat flux vector due to specified heat flux input,

$$\mathbf{Q}^h = \int_S \mathbf{h} \mathbf{T}_e \mathbf{N}^T dS \quad (2.293)$$

is the heat flux vector due to convection B.C.,

$$\mathbf{Q}^r = \int_S \kappa \mathbf{T}_e \mathbf{N}^T dS \quad (2.294)$$

is the heat flux vector due to radiation boundary condition, \mathbf{T} , $\dot{\mathbf{T}}$ are the vectors of nodal point temperatures and temperature gradients, k is the equivalent radiation coefficient given

by Equation (2.279), and, finally, the shape functions and the B matrix are defined in Equation (2.284) and Equation (2.285).

Equation (2.286), in general, represents a system of nonlinear transient equations. We write the equation at time $(t + \Delta t)$ and iteration (i) and introduce the following linearization assumptions:

The linearized form of Equation (2.273) may, be written as,

$${}^{t+\Delta t} \hat{q}^c(i-1) = \int_V {}^{t+\Delta t} (\rho C)^{(i-1)} N^T N \{ {}^{t+\Delta t} \mathbf{T}^{(i-1)} - {}^t \mathbf{T} \} \left(\frac{1}{\Delta t} \right) dV \quad (2.295)$$

2.6.3 Surface Radiation Heat Transfer

Introduction

Radiation heat transfer differs from conduction in some aspects. Unlike conduction surface radiation does not require any medium for heat transfer. Conduction heat transfer depends solely on the local distribution of temperature, whereas surface radiation exchange depends on the global temperature distribution in the domain. Therefore, inclusion of radiation in a heat transfer problem couples temperature at a 'node' with those at all other nodes in the domain. In addition, surface radiation in a heat transfer between two surfaces is proportional to the difference in fourth powers of their absolute temperatures. This dependency on the fourth power of temperature makes energy equation non-linear.

It is evident from the above discussion that analyzing a problem with surface radiation effects is far more complex than conduction heat transfer alone. However, radiation heat transfer plays an important role in many engineering applications and one cannot afford to neglect its effect on temperature distribution. Some of such applications are found in fire research, crystal growth, glass forming, combustion, solar energy collectors, underhood heat transfer in an automobile and thermal design of planetary robots.

Assumptions and Equations

The analysis of radiation heat transfer is presently based on the following assumptions.

- (i) the surfaces are grey

- (ii) the surfaces considered are diffused
- (iii) the surfaces considered are isothermal

The first two assumptions imply that emissivity and absorptivity of a surface are independent of wavelength and direction of emittance/incidence. In our future versions, some of these restrictions will be relaxed. The third assumption, however, can be controlled by discretization process, at an increased computational cost.

Based on the above assumptions, conservation of radiative energy leads to the following equation [Siegel and Howell]:

$$\sum_{j=1}^N \left(\frac{\delta_{ij}}{\epsilon_j} - \frac{1-\epsilon_j}{\epsilon_j} F_{ij} \right) q_{rj} = \sum_{j=1}^N (\delta_{ij} - F_{ij}) \sigma T_j^4 - \left(1 - \sum_{j=1}^N F_{ij} \right) \sigma T_s^4$$

For convenience the above equation can be represented in matrix form as,

$$[\mathbf{R}]\{\mathbf{q}_r\} = [\mathbf{S}]\{\mathbf{T}\} - \{\mathbf{B}\} \tag{2.296}$$

where,

$$R_{ij} = \frac{\delta_{ij}}{\epsilon_j} - \frac{1-\epsilon_j}{\epsilon_j} F_{ij} \tag{2.297}$$

$$S_{ij} = (\delta_{ij} - F_{ij}) \sigma T_j^3 \tag{2.298}$$

and,

$$B_i = \left[1 - \sum_{j=1}^N F_{ij} \right] \sigma T_s^4 \tag{2.299}$$

where, δ_{ij} is the Kronecker Delta, ϵ_j refers to emissivity of the j -th surface, F_{ij} is a view factor between an i -th and j -th surface, q_{rj} refers to radiative flux leaving (+ve) the j -th

surface, σ is the Stefan-Boltzmann constant, T_j is the absolute temperature of the j -th surface and N is the total number of surfaces with radiative flux, q_r , that couples the radiative heat balance with the overall energy conservation expressed by [Equation \(2.264\)](#).

Note that if $\sum_{j=1}^N F_{ij} = 0$, for the i -th surface, Equation (2.296) reduces to radiation boundary condition,

$$q_{ri} = \varepsilon_i \sigma (T_i^4 - T_s^4) \quad (2.300)$$

as desired.

Computation of View Factors

In order to compute q_r s (in Equation (2.296)), view factors F_{ij} , between all radiating surfaces must be available. In this section, the physical meaning of view factor and its calculation will be discussed.

In the above defined Equation (2.296), F_{ij} , the view factor between the surfaces i and j must be computed. For a better understanding, i, j in the above equation can be replaced with 1 and 2. Thus, view factor, F_{1-2} between two arbitrary surfaces (see Figure 2.6), '1' and '2' is defined as a fraction of diffuse radiant energy leaving surface '1' that arrives at surface '2'. Mathematically,

$$F_{1-2} = \frac{1}{A_1} \int_{A_1} \int_{A_2} \frac{\cos \theta_1 \cos \theta_2}{\pi r_{12}^2} dA_1 dA_2 \quad (2.301)$$

where, A_1 and A_2 are the areas of surfaces 1 and 2, respectively, r_{12} is the distance between the two elemental areas dA_1 and dA_2 , θ_1 is the angle between the position dependent normal vector \vec{n} and the line connecting dA_1 and dA_2 . Angle θ_2 is defined in a similar way. It must be noted that in defining the above expression for the view factor, it is assumed that the two surfaces are gray, diffuse and isothermal. Another important relation connecting the view factors F_{1-2} and F_{2-1} is as follows,

$$A_1 F_{1-2} = A_2 F_{2-1} \quad (2.302)$$

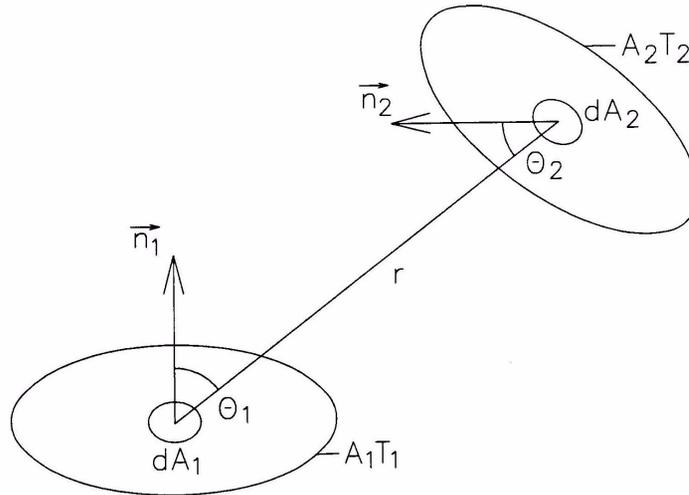


Figure 2.6: Definition of Terms Used in View Factor Calculations

The above relationship, known as reciprocal relationship, can be used to evaluate F_{2-1} if F_{1-2} and the areas A_1 and A_2 are known. It must be noted that $\cos\theta_1$ and $\cos\theta_2$ must be positive in order for the surface dA_1 and dA_2 to ‘see’ each other. If either of the cosines has a negative value, the corresponding view factor, $F_{dA_1-dA_2}$ should be set to zero. Such cases, in which the inactive side of the radiating face acts as an obstructor, will be termed as ‘self obstruction’ cases. Also, view factor F_{1-2} should be set to zero, if a third surface obstructs the view between surfaces 1 and 2.

In order to calculate view factors internally, the user must specify the radiation surfaces in terms of the finite element faces of a discretized domain. These pieces of information can be supplied very easily via *RADSURFACE card in the NISA file of NISAHEAT. Each radiating face is taken as one radiation surface. View factors between the radiating surfaces are automatically generated by NISAHEAT taking into account self obstruction and obstructions due to a third surface. Various parameters controlling the computation of view factors can be input through an Executive command, “SRAD = IOBST, NAXI, NDIV, α ” (See [Section 5.3](#)).

As can be assessed from the preceding discussion, computing view factors can result in usage of excessive computer time. To economize this computation, different techniques are used depending on whether the geometry being analyzed is 2D, 3D or axisymmetric. For example, double area integration method [Siegel and Howell] is employed in comparison with contour integration method [Siegel and Howell] when a 3D geometry, with radiation surfaces, is being analyzed. No special directives are required when computing view factors for axisymmetric geometries. NISA/HEAT internally generates a complete 3D model (with the axis of symmetry as the Y-axis [NISA/HEAT]) to calculate the required view factors. Furthermore, for 2D problems, a completely different approach, called Hottel's crossed-string method [Siegel and Howell], is employed for its computational efficiency and accuracy. Reference [Siegel and Howell] provides more details for evaluating view factors for interested readers.

Solution Procedure

In order to account for surface radiative exchange, discretized Equation (2.296) must be solved for the radiative flux, \mathbf{q}_r . Then its effect must be reflected in the overall energy balance at the affected surfaces. This is achieved by evaluating the first integral on the right hand side of Equation (2.286) as follows,

$$\int_S \mathbf{Q} \mathbf{N}^{ST} dS \quad (2.303)$$

where,

$$\mathbf{Q} = \mathbf{Q}^s + \mathbf{Q}^h + \mathbf{Q}^r$$

In the above equality \mathbf{Q}^s , \mathbf{Q}^h and \mathbf{Q}^r refer to the applied heat flux, effect due to convection boundary condition and effect due to radiation on this surface, S respectively. Radiative effects can be specified via *RADBC card, and/or via *RADSURFACE card. In the latter case, gray-body radiative effects will be considered and \mathbf{q}_r is evaluated by solving Equation (2.296). It must be pointed out here that a temperature field (see the right hand side of Equation (2.296) must be available, if \mathbf{q}_r s, due to gray-body radiation effects are to be included. On the other hand, \mathbf{q}_r s affect the distribution of temperature in the domain. This leads to a coupling between the energy and radiative energy transfer systems. In NISA/

HEAT, a sequential solution procedure is adapted to solve for temperature distribution, T_s and then evaluation of \mathbf{q}_r s using discretized Equation (2.286) and Equation (2.296), respectively. This procedure is undertaken to reduce the wave front size (hence solution time), which would otherwise be large for simultaneous solution of T and \mathbf{q}_r . An iterative loop between the distributions of T_s and \mathbf{q}_r s is established until these are converged below a specified tolerance. Both steady and transient solutions can be attempted using this approach. The transient solution for \mathbf{q}_r is representative of temperature distribution existing at that instant of time on the radiative surfaces and the domain in general.

It has been observed that \mathbf{q}_r (and hence T) solution may not converge or may do so slowly. An under relaxation of \mathbf{q}_r leads to its stabilization. This is achieved as follows,

$$\mathbf{q}_{r^*}^{i+1} = \alpha \mathbf{q}_r^{i+1} + (1 - \alpha) \mathbf{q}_{r^*}^i \quad (2.304)$$

where, the superscript i refers to an iteration number, and α is a relaxation factor $0 < \alpha < 1$ defined by the user via an executive card SRAD = IOBST, NAXI, NDIV, α , and \mathbf{q}_{r^*} refers to a relaxed value of \mathbf{q}_r . Note that the relaxation parameter, α , depends on radiation-conduction parameter, N_{RC} , which is defined as,

$$N_{RC} = \frac{\sigma T_h^4 L}{k(T_h - T_c)} \quad (2.305)$$

where, σ is the Stefan-Boltzman constant, T_h and T_c are upper and lower absolute temperatures, k is the material conductivity and L is characteristic length. If radiation-conduction parameter is much larger than one, low values of relaxation parameter may have to be used in order to get convergence.

Together with checking for convergence of temperature field, convergence is also checked for \mathbf{q}_r .

$$L_2 \text{ Norm} = \frac{\sum_{i=1}^N (\mathbf{q}_{r*}^{i+1} - \mathbf{q}_{r*}^i)^2}{\sum_{i=1}^N (\mathbf{q}_{r*}^{i+1})^2} \quad (2.306)$$

The solution is continued until L_2 norm falls below the user specified tolerance.

2.7 Constitutive Equations

2.7.1 Introduction

Constitutive equations for a particular material are simply the relationship between the applied loads or stresses and the response of the material in terms of displacements and strains. Constitutive relations are mathematical representation of ideal material responses which is designed to approximate the physical behavior of the real material. The constitutive equations, available in NISA, are intended to describe the macroscopic behavior of the material. In the following sections, we outline the stress-strain constitutive relations available in NISA and discuss their assumptions and limitations.

2.7.2 Ideal Elastic Material Model

A material is called ideal elastic material if a body formed of this material recovers its original shape upon removal of the applied loads and if there is a one-to-one relationship between the state of stress and the state of strain. The most common form of the ideal elastic material is the classical elastic constitutive equations which is often called the generalized Hooke's law. These are nine equations expressing the stress components as a linear homogeneous function of the nine strain components, or in a tensorial form,

$$\sigma_{ij} = C_{ijkl} \varepsilon_{kl} \quad (2.307)$$

or $\sigma = C \varepsilon$

where, σ is the Cauchy stress tensor, ε is the small strain tensor, and C is a fourth order constitutive tensor. In the above general form, 81 material constants are required to describe the behavior of the material. Considering symmetry of the stress and strain tensors and assuming the material to be elastically isotropic, Equation (2.307) reduces to [Ref. 2.16],

$$\sigma_{ij} = [\lambda \delta_{ij} \delta_{rs} + \mu (\delta_{ir} \delta_{js} + \delta_{is} \delta_{jr})] \varepsilon_{rs} \quad (2.308)$$

which only needs two constants (Lame's elastic constants λ and μ) to describe the material behavior.

Currently, two linear elastic material models are available in the NISA material library. These are:

- Linear elastic isotropic model
- Linear elastic orthotropic model

In the linear elastic isotropic model, two constants are required to describe the material behavior, the modulus of elasticity or Young's modulus, E , and the Poisson's ratio, ν . Various stress-strain matrices, for this material, for different elements are summarized in [Table 2.1](#). For these matrices, the stress and strain tensors are arranged in a vector form and the constitutive tensor is arranged in a square matrix form.

In the linear elastic orthotropic model, there exists three mutually perpendicular planes of elastic symmetry at each point in the material. A plane of elastic symmetry exists, if the elastic constants are invariant for every pair of coordinate system which are mirror images of each other in this plane. If we identify the principal axes of elastic symmetry by (1, 2, 3), the constitutive relations for orthotropic material in the principal directions may be written as [Ref. 2.17],

$$\begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \tau_{12} \\ \tau_{23} \\ \tau_{31} \end{Bmatrix} = \left(\frac{1}{\alpha}\right) [\mathbf{C}] \begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{12} \\ \gamma_{23} \\ \gamma_{31} \end{Bmatrix} \quad (2.309)$$

where,

$$[C] = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{21} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{31} & C_{32} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{41} & C_{42} & C_{43} & C_{44} & C_{45} & C_{46} \\ C_{51} & C_{52} & C_{53} & C_{54} & C_{55} & C_{56} \\ C_{61} & C_{62} & C_{63} & C_{64} & C_{65} & C_{66} \end{bmatrix}$$

$$C_{11} = E_{11}(1 - \nu_{23}\nu_{32})$$

$$C_{12} = E_{22}(\nu_{12} + \nu_{13}\nu_{32})$$

$$C_{13} = E_{33}(\nu_{13} + \nu_{12}\nu_{23})$$

$$C_{14} = 0$$

$$C_{15} = 0$$

$$C_{16} = 0$$

$$C_{21} = E_{22}(\nu_{12} + \nu_{13}\nu_{32})$$

$$C_{22} = E_{22}(1 - \nu_{13}\nu_{31})$$

$$C_{23} = E_{33}(\nu_{23} + \nu_{21}\nu_{13})$$

$$C_{24} = 0$$

$$C_{25} = 0$$

$$C_{26} = 0$$

$$C_{31} = E_{33}(v_{13} + v_{12}v_{23})$$

$$C_{32} = E_{33}(v_{23} + v_{21}v_{13})$$

$$C_{33} = E_{33}(1 - v_{12}v_{21})$$

$$C_{34} = 0$$

$$C_{35} = 0$$

$$C_{36} = 0$$

$$C_{41} = 0$$

$$C_{42} = 0$$

$$C_{43} = 0$$

$$C_{44} = \alpha G_{12}$$

$$C_{45} = 0$$

$$C_{46} = 0$$

$$C_{51} = 0$$

$$C_{52} = 0$$

$$C_{53} = 0$$

$$C_{54} = 0$$

$$C_{55} = \alpha G_{23}$$

$$C_{56} = 0$$

$$C_{61} = 0$$

$$C_{62} = 0$$

$$C_{63} = 0$$

$$C_{64} = 0$$

$$C_{65} = 0$$

$$C_{66} = \alpha G_{31}$$

$$\alpha = 1 - \nu_{12}\nu_{21} - \nu_{23}\nu_{32} - \nu_{31}\nu_{13} - \nu_{12}\nu_{23}\nu_{31} - \nu_{21}\nu_{13}\nu_{32}$$

and E_{ii} is the elastic modulus in the direction i , ($i = 1,2,3$); ν_{ij} is the Poisson's ratio characterizing the strain in the j -th. direction produced by a stress in the i -th. direction, ($i, j = 1,2,3$), and finally G_{12} , G_{23} , G_{31} are the shear moduli in the (1,2), (2,3), and (3,1) planes, respectively. It should be also noted that symmetry conditions give rise to the following reciprocity relation:

$$\begin{aligned} \nu_{12}E_{22} &= \nu_{21}E_{11} \\ \nu_{13}E_{33} &= \nu_{31}E_{11} \\ \nu_{23}E_{33} &= \nu_{32}E_{22} \end{aligned}$$

$$\text{or } \nu_{ij}E_{jj} = \nu_{ji}E_{ii} \quad (\text{no sum}) \quad (2.310)$$

An extension to the above two linear elastic material models is the case where the constitutive coefficients are temperature dependent. In such case, the constitutive tensor will be constant for a given temperature. In NISA, all the linear elastic constants, for isotropic or orthotropic materials, may be temperature dependent. The user may express a particular property as a polynomial function of temperature with the polynomial order of up to the *fourth* order.

Table 2.1: Stress - Strain matrices for linear isotropic elastic material

PROBLEM	Stain Vector	Stress Vector	Material Matrix D
Bar	e_{xx}	σ_{xx}	E
Beam	K_{xx}	M_{xx}	EI
Plane stress	$\begin{Bmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \gamma_{xy} \end{Bmatrix}$	$\begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \tau_{xy} \end{Bmatrix}$	$\frac{E}{(1-\nu^2)} \begin{bmatrix} 1 & \nu & 0 \\ & 1 & 0 \\ SYM & & \frac{1-\nu}{2} \end{bmatrix}$

PROBLEM	Strain Vector	Stress Vector	Material Matrix D
Plane strain	$\begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{Bmatrix}$	$\begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \tau_{xy} \end{Bmatrix}$	$\frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \frac{\nu}{1-\nu} & 0 \\ & 1 & 0 \\ SYM & & \frac{1-2\nu}{2(1-\nu)} \end{bmatrix}$
Axisymmetric**	$\begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \end{Bmatrix}$	$\begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \tau_{xy} \end{Bmatrix}$	$\frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \frac{\nu}{1-\nu} & \frac{\nu}{1-\nu} & 0 \\ & 1 & \frac{\nu}{1-\nu} & 0 \\ & & 1 & 0 \\ SYM & & & \frac{1-2\nu}{2(1-\nu)} \end{bmatrix}$
3-D	$\begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix}$	$\begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix}$	$\frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \frac{\nu}{1-\nu} & \frac{\nu}{1-\nu} & 0 & 0 & 0 \\ & 1 & \frac{\nu}{1-\nu} & 0 & 0 & 0 \\ & & 1 & 0 & 0 & 0 \\ & & & \frac{1-2\nu}{2(1-\nu)} & 0 & 0 \\ & & & & \frac{1-2\nu}{2(1-\nu)} & 0 \\ SYM & & & & & \frac{1-2\nu}{2(1-\nu)} \end{bmatrix}$

Theoretical Overview
Constitutive Equations

PROBLEM	Stain Vector	Stress Vector	Material Matrix <i>D</i>
Shell*	$\begin{Bmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix}$	$\begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix}$	$\frac{E}{(1-\nu^2)} \begin{bmatrix} 1 & \nu & 0 & 0 & 0 \\ & 1 & 0 & 0 & 0 \\ & & \frac{1-\nu}{2} & 0 & 0 \\ & & & \frac{1-\nu}{2\phi} & 0 \\ SYM & & & & \frac{1-\nu}{2\phi} \end{bmatrix}$

E = Young's modulus

ν = Poisson's ratio

ϕ = shear factor (1.2 is commonly used)

* The stress - strain relationship must be given in shell local coordinate system with z direction through the thickness

** For axisymmetric analysis, x is the radial direction, y is the axial direction and z is the circumferential direction.

2.7.3 Elastoplastic Material Model

Elastoplastic materials are assumed to behave elastically up to a certain stress limit, after which combined elastic and plastic behavior occurs. Plasticity is path-dependent and is characterized by an irreversible permanent straining that occurs in the material once the elastic behavior stress limit is reached.

In order to determine the yield and postyield behavior of the material, any plasticity theory should cover the following fundamental points:

- **Yield criteria:** to define specific stress combinations that will initiate the inelastic response, i.e., defining an initial yield surface.
- **Hardening rule:** to define the evolution of the yield surface with stress, strain, and other parameters.
- **Plastic stress-strain relation or a flow rule:** to relate the plastic strain increment to the current stress level and stress increment.

1. Yield Criteria

A yield criterion is simply a relation defining the limit of elasticity and the onset of plasticity under any possible stress state. If we restrict our attention, for the present, to isotropic materials, the yield criterion may be expressed in terms of stress invariants [2.18],

$$f(I_1, I_2, I_3) = K(k) \tag{2.311}$$

where, I_1, I_2, I_3 are the first, second and third stress invariants, respectively, given by,

$$I_1 = \sigma_{ii} \tag{2.312}$$

$$I_2 = \frac{1}{2}(\sigma_{ij}\sigma_{ij} - \sigma_{ii}\sigma_{jj})$$

$$I_3 = \det\sigma = |\sigma|$$

and K is a constant that may be function of some hardening parameter k .

Based on the above general discussion, the following yield criteria may be assumed:

(a) The von Mises Yield Criterion

The von Mises yield criterion suggests that yielding will occur when the second deviatoric stress invariant, I'_2 , reaches a critical value [2.18],

$$\sqrt{I'_2} = K(k) \quad (2.313)$$

where,

$$\begin{aligned} I'_2 &= \frac{1}{2} \sigma'_{ij} \sigma'_{ij} \\ &= \frac{1}{2} (\sigma'^2_{xx} + \sigma'^2_{yy} + \sigma'^2_{zz}) + \tau^2_{xy} + \tau^2_{yz} + \tau^2_{zx} \end{aligned} \quad (2.314)$$

and

$$\sigma'_{ij} = \sigma_{ij} - \sigma_m, \quad \sigma_m = \frac{1}{3} \sigma_{ii} \quad (2.315)$$

The von Mises yield criterion may also be expressed in the following form,

$$\bar{\sigma} = \sqrt{3}K \quad (2.316)$$

where,

$$\bar{\sigma} = \left[\frac{3}{2} \sigma'_{ij} \sigma'_{ij} \right]^{1/2} \quad (2.317)$$

is the effective, generalized or equivalent stress.

It is noted that the von Mises yield criterion is independent of the hydrostatic pressure (σ_m). Two physical interpretations of the criterion exists. One indicates that yielding occurs when the octahedral shear stress reaches a critical value. The octahedral shear stress

is that acting on the planes of a regular octahedron, the apices of which coincide with principal axes of stress. The second interpretation indicates that yielding will occur when the recoverable or elastic distortion energy reaches a critical value.

(b) The Tresca Yield Criterion

The Tresca yield criterion is a maximum shear stress criterion which is also independent of the hydrostatic stress component (like the von Mises criterion). If the principal stresses are denoted by σ_{11} , σ_{22} and σ_{33} where all $\sigma_{11} \geq \sigma_{22} \geq \sigma_{33}$, then the Tresca criterion indicates that yielding occurs when,

$$\sigma_{11} - \sigma_{33} = Y(k) \quad (2.318)$$

where, Y is a material constant that may be a function of an appropriate hardening parameter k . The constant Y represents the yield strength of the material in pure shear. A symmetric form of the Tresca yield condition may be written as [2.16],

$$[(\sigma_{11} - \sigma_{22})^2 - Y^2][(\sigma_{22} - \sigma_{33}) - Y^2][(\sigma_{33} - \sigma_{11}) - Y^2] = 0 \quad (2.319)$$

which is written in an invariant form as,

$$\frac{27(I'_2)^3}{Y^6} \left[\frac{4}{27} - \frac{(I'_3)^2}{(I'_2)^3} \right] - \left(1 - \frac{3I'_2}{Y^2} \right)^2 = 0 \quad (2.320)$$

where, I'_3 is the third deviatoric stress invariant,

$$\begin{aligned} I'_3 &= \det \sigma' = |\sigma'| & (2.321) \\ &= \frac{1}{3} \sigma'_{ij} \sigma'_{jk} \sigma'_{ki} \end{aligned}$$

For design purposes, if no plastic yielding is to be permitted, the Tresca criterion is more conservative than the von Mises criterion.

The maximum difference between the two criteria is 15% and occurs for pure shear condition. For most metals, the von Mises criteria fits the experimental data more closely than the Tresca criterion. A graphical two-dimensional and II-plane presentation for the two criteria is shown in [Figure 2.7](#).

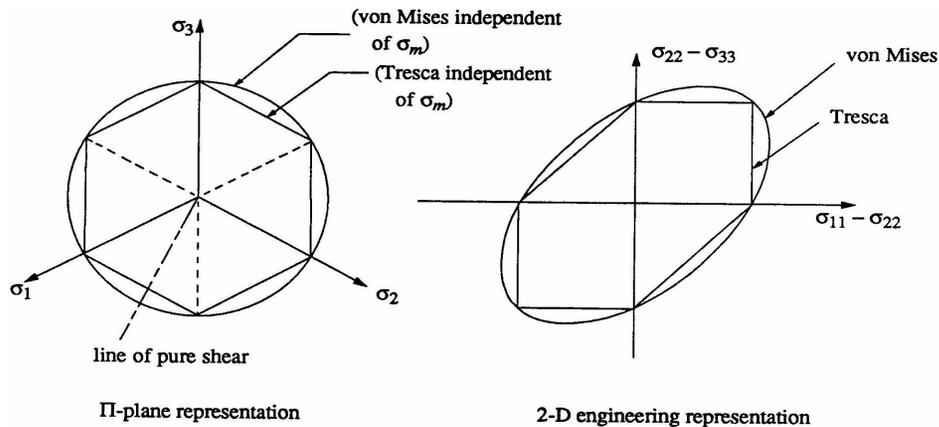


Figure 2.7: II-plane and two dimensional presentation of von Mises and Tresca yield criteria

(c) The Mohr - Coulomb Yield Criterion

This is a generalization of the Coulomb friction law for failure and may be written in the form [2.19],

$$(\sigma_{11} - \sigma_{33}) = 2C \cos \phi - (\sigma_{11} + \sigma_{33}) \sin \phi \quad (2.322)$$

where, $\sigma_{11} \geq \sigma_{22} \geq \sigma_{33}$ are the principal stresses, C is the cohesion of the material and ϕ is the angle of internal friction. A Mohr circle representation of the above criterion is given in [Figure 2.8](#).

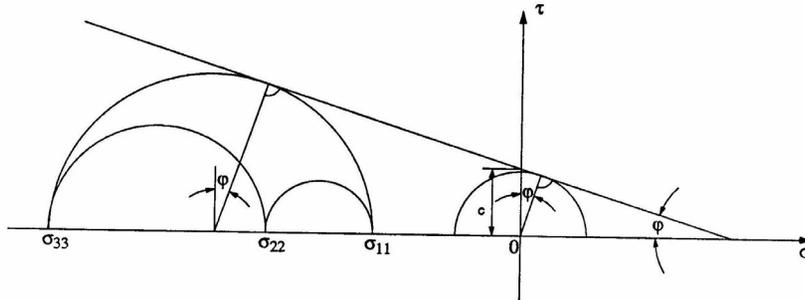


Figure 2.8: Mohr circle representation for Mohr - Coulomb criterion

In principal stress space, the Mohr-Coulomb yield criterion gives a conical surface whose cross-section at any point along the axis is an irregular hexagon, see [Figure 2.9](#). The conical surface implies that the yield criterion is a function of the hydrostatic pressure. In other words, the size of the cross-section increases with the increase of hydrostatic pressure, σ_m .

(d) The Drucker-Prager Yield Criterion

The Drucker-Prager criterion is an extension to the von Mises criterion, which takes account of the effect of hydrostatic pressure on the yield surface. The equation representing Drucker - Prager condition is [2.20]

$$\alpha I_1 + (I'_2)^{1/2} = K' \quad (2.323)$$

where, α , K' are material parameter, and I_1 , is the first stress invariant. In the principal stress space, [Equation \(2.323\)](#) represents a circular cone. In order to make the Drucker - Prager circle coincide with the outer/inner apices of the Mohr-Coulomb hexagon, we should have,

$$\alpha = \left(\frac{1}{\beta}\right) 2 \sin \phi, K' = \left(\frac{1}{\beta}\right) 6 C \cos \phi \quad (2.324)$$

where,

$$\beta = \sqrt{3}(3 \mp \sin \phi)$$

C is the cohesion of the material and ϕ is the angle of internal friction, and where the ‘-’ and ‘+’ signs, in Equation (2.325), make the Drucker-Prager circle coincide with the outer and inner apices of the Mohr-Coulomb hexagon, respectively. In NISA, the outer Drucker-Prager circle is used, Figure 2.9 shows a graphical representation of the Mohr-Coulomb and the Drucker-Prager Criteria in the II - plane.

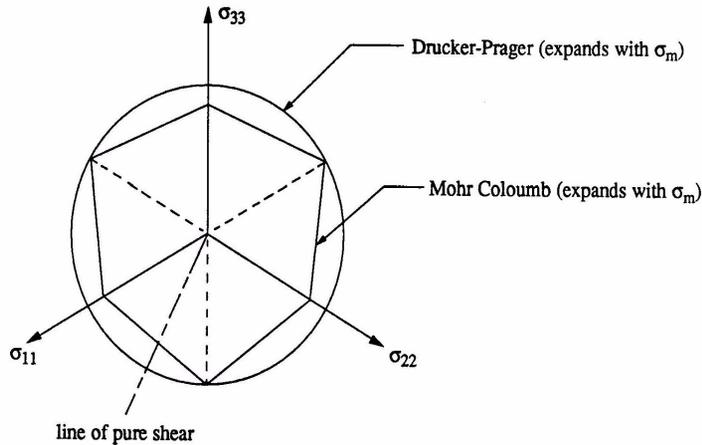


Figure 2.9: II-plane representation for Mohr - Coulomb and Drucker-Prager yield criteria

(e) The Ilyushin Yield Criterion

The Ilyushin criterion is formulated in terms of the stress-resultants, the membrane forces and bending moments. The equation representing Ilyushin yield condition is [2.53, 2.54],

$$f = \bar{Q}_M + \bar{Q}_B + |\bar{Q}_{MB}| \gamma - (\sigma_y)^2 = 0 \quad (2.325)$$

where, \bar{Q}_M , \bar{Q}_B , \bar{Q}_{MB} function of the section membrane forces and bending moments, σ_y , is the current yield stress obtained from a simple tension test, and γ is a constant. Figure 2.10 depicts this yield function.

2. Hardening Rule

The hardening rule determines how the yield function changes during plastic deformation after the initial onset of yielding. Few postulates are available in the literature and the following ones are implemented in the NISA program.

(a) Perfect Plasticity

This is the simplest possible postulate and it assumes that the yield surface is constant or does not change with plastic deformation. This leads to the classical mathematical theory of perfect plasticity. A typical behavior of perfectly plastic solids is that the plastic strain increments are not completely determined by the stress state. This is due to the fact that the flow rule (see [Equation \(2.335\)](#)) will continue to be satisfied if all the plastic strain components are multiplied by an arbitrary factor. Perfect plasticity postulate may cause convergence problems. Such convergence problems may be reduced if the problem is switched to displacement controlled situation or if a small hardening modulus is introduced. [Figure 2.11\(a\)](#) shows graphical representation of the perfect plasticity postulate.

(b) Isotropic Hardening

In this postulate, the yield surface is assumed not to change in shape (in the stress space) while its size increases as a function of some hardening parameter. Two different schemes are being used to define the hardening parameter, k [2.16]. One assumes that k is function of the plastic work done, such that,

$$k = F(W^P) = F\left(\int \sigma_{ij} d\bar{\varepsilon}_{ij}^P\right) \quad (2.326)$$

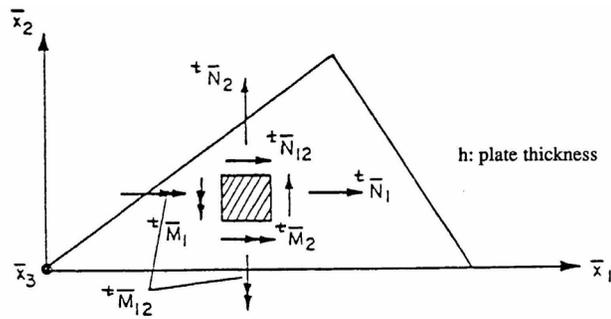
where, $d\varepsilon_{ij}^P$ are the incremental plastic strains. The other approach assumes that the hardening parameter k is a function of an accumulative plastic strain measure, namely the effective, generalized or equivalent plastic strain, such that,

$$k = H\left(\int d\bar{\varepsilon}^P\right) \quad (2.327)$$

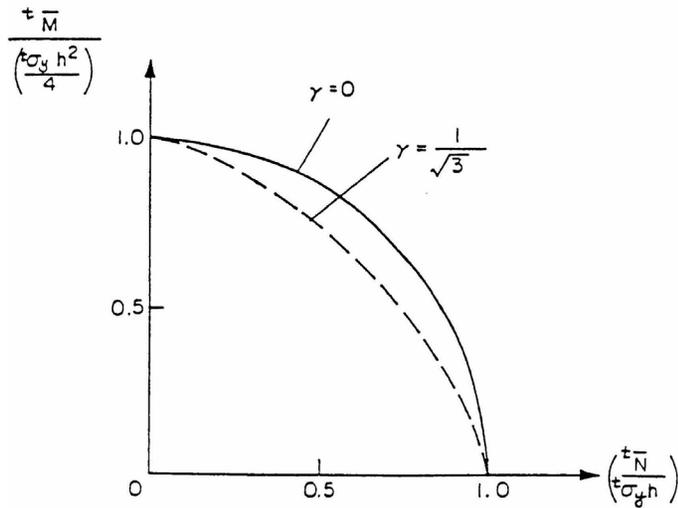
where, the incremental effective, generalized or equivalent plastic strain increment is given by,

$$d\bar{\varepsilon}^P = \left[\frac{2}{3} d\varepsilon_{ij}^P d\varepsilon_{ij}^P \right]^{1/2} \quad (2.328)$$

Both schemes give the same results in the case of von Mises yield criterion. The work hardening hypothesis is, however, more general from a thermodynamics point of view than the strain hardening hypothesis and is the one used in NISA. Figure 2.11(b) shows a graphical representation of the isotropic hardening postulate.



(a) Element section forces and moments



(b) Uniaxial yield condition

Figure 2.10: Ilyushin yield criterion

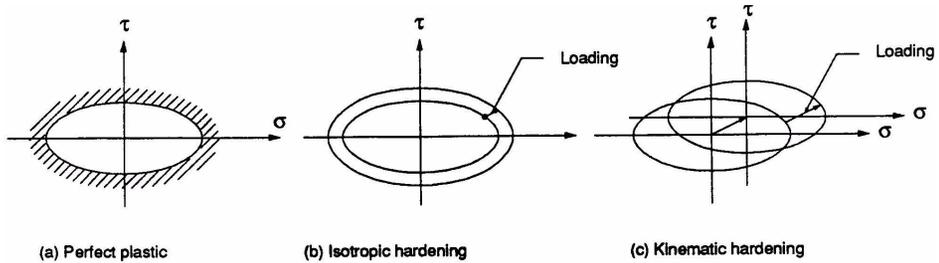


Figure 2.11: Presentation of various hardening assumptions

(c) Kinematic Hardening

Prager [2.21] proposed a postulate in which the yield surface only translates in the stress space and does not change in size. Therefore, the subsequent yield surfaces, after the initial one, are controlled by the equation,

$$f(\sigma_{ij} - \alpha_{ij}) = 0 \tag{2.329}$$

where, α_{ij} are the nine coordinates of the new center of the yield surface in the stress space, and where f has exactly the same dependence on $(\sigma_{ij} - \alpha_{ij})$ as the initial yield condition had on σ_{ij} .

Two expressions are available for the definition of the shift of the yield surface center, α_{ij} .

First is a linear hardening assumption, due to Prager, which assumes,

$$d\alpha_{ij} = C d\varepsilon_{ij}^p \tag{2.330}$$

where, C is a constant characterizing the material and may be a function of stress, stress history and plastic work. The second is a modification to Equation (2.330), due to Ziegler [2.22], which assumes,

$$d\alpha_{ij} = (\sigma_{ij} - \alpha_{ij})d\mu, \quad d\mu > 0 \tag{2.331}$$

Prager's assumption implies that the yield surface moves in the direction of the outward normal to the yield surface at the stress point, which led to some complications in applications. Ziegler's modification assumes that the yield surface still does not change in size and only translates but in the direction of the vector connecting the center of the yield surface with the stress point. The constant $d\mu$ is determined by the condition of yielding and is given by,

$$d\mu = \frac{\frac{\partial f}{\partial \sigma_{ij}} d\sigma_{ij}}{(\sigma_{kl} - \alpha_{kl}) \frac{\partial f}{\partial \sigma_{kl}}} \quad (2.332)$$

Ziegler's assumption is the one adopted in NISA. Generally, kinematic hardening may give better representation for yield surface changes if load reversals and Bauschinger effects are to be accounted for. There are some experimental evidence that this is the case for moderate plastic deformations. However, the actual yield surface changes need a model which changes in shape and translates as a function of the deformation history. Such model may require microscopic rather than macroscopic considerations as well as residual stress considerations, and is not generally established in the literature as of this date. [Figure 2.11\(c\)](#) shows a graphical representation of the kinematic hardening postulate.

(d) Mixed Hardening

Mixed hardening is simply a combination of the expansion and the translation of the yield surface. In NISA, a linear combination of the two models is available through specifying a single parameter β , where $0.0 \leq \beta \leq 1.0$. If $\beta = 0.0$, the model will be purely kinematic hardening, whereas if $\beta = 1.0$, the model will be purely isotropic hardening. Combination of the two models is achieved with $0.0 < \beta < 1.0$.

3. Elastoplastic Stress-Strain Relations

After the initial yielding of the material, the behavior will be partly elastic and partly plastic. During a stress increment, additive decomposition of the strain components is assumed such that,

$$d\varepsilon_{ij}^{tot} = d\varepsilon_{ij}^e + d\varepsilon_{ij}^p \quad (2.333)$$

where, $d\varepsilon_{ij}^{tot}$, $d\varepsilon_{ij}^e$ and $d\varepsilon_{ij}^p$ are the total, elastic and plastic incremental strain components, respectively.

The elastic constitutive equations establish the elastic strains as,

$$d\varepsilon_{ij}^e = C_{ijkl}^{-1} d\sigma_{kl} \quad (2.334)$$

where, C_{ijkl}^{-1} is the inverse of the elastic constitutive tensor. The plastic strain components are assumed to be proportional to the gradient of a function of the stress components, Q , called the '*plastic potential function*', such that,

$$d\varepsilon_{ij}^p = d\lambda \frac{\partial Q}{\partial \sigma_{ij}} \quad (2.335)$$

where, $d\lambda$ is a 'proportionality constant' or a 'plastic multiplier'. Equation (2.335) is termed the '*flow rule*' or the '*normality criterion*' since it govern the plastic flow after yielding and it assumes it to be in a direction normal to the plastic potential function. Two postulates are given for the definition of the plastic potential function, Q . These are,

□ *Associated Plasticity*

In which it is assumed that the plastic potential function is the same as the yield function, i.e.,

$$Q = f \quad (2.336)$$

Certain variational principals and uniqueness theorems may be formulated under the above assumption.

□ *Non Associated Plasticity*

In which it is assumed that the plastic potential function and the yield function are not the same, i.e.,

$$Q \neq f \quad (2.337)$$

In NISA, only associated plasticity is available.

To calculate the proportionality factor $d\lambda$ in Equation (2.335), we start by the following general form of the yield function. To calculate the proportionality factor $d\lambda$ in Equation (2.335), we start by the following general form of the yield function,

$$f = f(\sigma_{ij}, k, T, \dot{\varepsilon}) \quad (2.338)$$

where, σ_{ij} are the stress components, k is a hardening parameter, T is the temperature and $\dot{\varepsilon}$ is the strain rate.

During small incremental plastic deformation, the equilibrium condition requires that the plastic energy variation be stationary, which implies that $df = 0$,

$$df = 0 = \frac{\partial f}{\partial \sigma_{ij}} d\sigma_{ij} + \frac{\partial f}{\partial k} dk + \frac{\partial f}{\partial T} dT + \frac{\partial f}{\partial \dot{\varepsilon}} d\dot{\varepsilon} \quad (2.339)$$

but,

$$\frac{\partial f}{\partial k} dk = \frac{\partial f}{\partial k} \frac{\partial k}{\partial \varepsilon_{ij}^p} d\varepsilon_{ij}^p = \frac{\partial f}{\partial k} \frac{\partial k}{\partial \varepsilon_{ij}^p} d\lambda \frac{\partial f}{\partial \sigma_{ij}} \quad (2.340)$$

and,

$$d\sigma_{ij} = C_{ijkl}^e d\varepsilon_{kl}^e = C_{ijkl}^e \left[d\varepsilon_{kl}^{tot} - \left(d\varepsilon_{kl}^{th} + d\varepsilon_{kl}^T + d\varepsilon_{kl}^{\dot{\varepsilon}} + d\varepsilon_{kl}^p \right) \right] \quad (2.341)$$

where,

$d\varepsilon_{kl}^{tot}$: incremental total strain components

$d\varepsilon_{kl}^{th}$: incremental thermal strain components

$d\varepsilon_{kl}^T$: incremental strain components due to temperature dependent material properties

$d\varepsilon_{kl}^{\dot{\varepsilon}}$: incremental strain components due to strain rate dependent material properties

$d\varepsilon_{kl}^p$: incremental plastic strain components

and we have,

$$d\varepsilon_{kl}^{th} = \alpha_{kl} dT \Rightarrow \begin{cases} \alpha_{kk} \neq 0 \text{ (no sum)} \\ \alpha_{kl} = 0 \text{ for } k \neq l \end{cases} \quad (2.342)$$

where, α_{kl} : the instantaneous thermal expansion coefficients, and,

$$d\varepsilon_{kl}^T = \frac{\partial [C_{klmn}^e]^{-1}}{\partial T} \sigma_{mn} (dT) \quad (2.343)$$

$$d\varepsilon_{kl}^{\dot{\varepsilon}} = \frac{\partial [C_{klmn}^e]^{-1}}{\partial \dot{\varepsilon}} \sigma_{mn} (d\dot{\varepsilon}) \quad (2.344)$$

$$d\varepsilon_{kl}^p = d\lambda \frac{\partial f}{\partial \sigma_{kl}} \quad (2.345)$$

Substituting Equation (2.340) to Equation (2.345) into Equation (2.339) we get,

$$d\lambda \left\{ \frac{\partial f}{\partial \sigma_{ij}} C_{ijkl}^e \frac{\partial f}{\partial \sigma_{kl}} - \frac{\partial f}{\partial k} \frac{\partial k}{\partial \varepsilon_{ij}^p} \frac{\partial f}{\partial \sigma_{ij}} \right\} = \frac{\partial f}{\partial \sigma_{ij}} C_{ijkl}^e (d\varepsilon_{kl}^e)$$

$$\left(\frac{\partial f}{\partial T} dT + \frac{\partial f}{\partial \dot{\varepsilon}} d\dot{\varepsilon} \right)$$

where,

$$(d\varepsilon_{kl}^e) = d\varepsilon_{kl}^{tot} - \alpha_{kl}(dT) - \frac{\partial [C_{klmn}^e]^{-1}}{\partial T} \sigma_{mn}(dT) - \frac{\partial [C_{klmn}^e]^{-1}}{\partial \dot{\varepsilon}} \sigma_{mn}(d\dot{\varepsilon})$$

which gives $d\lambda$ as,

$$d\lambda = \frac{1}{A} \left[\frac{\partial f}{\partial \sigma_{ij}} C_{ijkl}^e (d\varepsilon_{kl}^e) + \left(\frac{\partial f}{\partial T} dT + \frac{\partial f}{\partial \dot{\varepsilon}} d\dot{\varepsilon} \right) \right] \quad (2.346)$$

where,

$$A = \left[\frac{\partial f}{\partial \sigma_{ij}} C_{ijkl}^e \frac{\partial f}{\partial \sigma_{kl}} - \frac{\partial f}{\partial k} \frac{\partial k}{\partial \varepsilon_{ij}^p} \frac{\partial f}{\partial \sigma_{ij}} \right] \quad (2.347)$$

or in matrix form,

$$d\lambda = \frac{1}{A} \left[\mathbf{a}^T \mathbf{D}^e \left\{ d\varepsilon^{tot} - \alpha dT - \frac{\partial (D^e)^{-1}}{\partial T} \sigma dT - \frac{\partial (D^e)^{-1}}{\partial \dot{\varepsilon}} \sigma d\dot{\varepsilon} \right\} + \left(\frac{\partial f}{\partial T} dT + \frac{\partial f}{\partial \dot{\varepsilon}} d\dot{\varepsilon} \right) \right] \quad (2.348)$$

$$\mathbf{A} = \left[\mathbf{a}^T \mathbf{D}^e \mathbf{a} - \frac{\partial f}{\partial k} \left(\frac{\partial k}{\partial \varepsilon^p} \right)^T \mathbf{a} \right], \quad (2.349)$$

where,

$$\mathbf{a} \equiv \frac{\partial f}{\partial \sigma_{ij}} \text{ arranged in a vector form} \quad (2.350)$$

$$\text{and } \mathbf{D}^e \equiv C_{ijkl}^e \text{ arranged in a matrix form} \quad (2.351)$$

Currently in NISA, in the application of Equation (2.346), the terms $(\partial/\partial \mathbf{T})$ and $(\partial/\partial \dot{\varepsilon})$ are not accounted for.

The second term in of Equation (2.347) is the hardening parameter and may be given by,

$$\begin{aligned} \left(\frac{\partial f}{\partial k} \right) \left(\frac{\partial k}{\partial \varepsilon_{ij}^p} \right) \left(\frac{\partial f}{\partial \sigma_{ij}} \right) &= \left(\frac{\partial f}{\partial \bar{\sigma}} \frac{\partial \bar{\sigma}}{\partial k} \right) \left(\frac{\partial k}{\partial \varepsilon_{ij}^p} \right) \left(\frac{\partial f}{\partial \sigma_{ij}} \right) \\ &= \left(\frac{\partial f}{\partial \bar{\sigma}} \frac{\partial \bar{\sigma}}{\partial \varepsilon^p} \frac{\partial \varepsilon^p}{\partial k} \right) \left(\frac{\partial k}{\partial \varepsilon_{ij}^p} \right) \left(\frac{\partial f}{\partial \sigma_{ij}} \right) \end{aligned} \quad (2.352)$$

Now, the work hardening parameter k will be represented by the amount of plastic work. This work hardening hypothesis is more general from a thermodynamics point of view than the strain hardening hypothesis.

Therefore, we may express:

$$dk = \sigma_{ij} d\varepsilon_{ij}^p \equiv \bar{\sigma} d\varepsilon^p \text{ hence} \quad (2.353)$$

$$\frac{d\varepsilon^p}{dk} = \frac{1}{\bar{\sigma}}, \frac{dk}{d\varepsilon_{ij}^p} = \sigma_{ij} \text{ and we also have} \quad (2.354)$$

$$\frac{\partial \bar{\sigma}}{\partial \bar{\varepsilon}^p} = \bar{H}' \quad (2.355)$$

where, \bar{H}' is the slope of the plastic part of the effective stress - effective strain curve, or the material plastic modulus.

Substitute (Equation (2.353) to Equation (2.355)) into (Equation (2.352)) we get:

$$\begin{aligned} \left(\frac{\partial f}{\partial k}\right) \left(\frac{\partial k}{\partial \varepsilon_{ij}^p}\right) \left(\frac{\partial f}{\partial \sigma_{ij}}\right) &= \left(\frac{\partial f}{\partial \bar{\sigma}} \frac{\partial \bar{H}'}{\partial \bar{\sigma}}\right) (\sigma_{ij}) \left(\frac{\partial f}{\partial \sigma_{ij}}\right) \\ &\equiv \left(\frac{\partial f}{\partial \bar{\sigma}} \frac{\partial \bar{H}'}{\partial \bar{\sigma}}\right) (\sigma^T)(a) \end{aligned} \quad (2.356)$$

4. Integration of the Plastic Constitutive Relation:

Numerical integration of the elastoplastic constitutive relations is an essential part in the analysis of elastoplastic problems. In principle, large number of integration procedures may be employed. Practically, efficiency, accuracy, and stability considerations govern the choice of a particular integrating scheme [2.23, 2.24].

One of the basic integration schemes is to use an explicit method like the Euler forward technique. In such a case, the solution is simply obtained by forward integration of the constitutive relations over a sufficient number of sub-increments. On the other hand if an implicit integration scheme is to be employed, iterations are necessary to predict the stress level at the end of each sub-increment. Two families of such integration schemes are generally adopted; the generalized midpoint scheme and the generalized trapezoidal scheme [2.23]. Other schemes such as the radial return, the mean normal and the closest point procedures are special cases of these two families of algorithms.

The generalized midpoint and the generalized trapezoidal schemes are basically an elastoplastic operator 'split' methods which allow the splitting of elastoplastic constitutive relations into two additive parts; elastic and plastic. The algorithm is based on a two step approach. First, an elastic predictor produces a trial 'elastic' stress state based on integrating

the elastic constitutive equation with the assumption of no plasticity. Second, a plastic corrector projects the stress increments backward (return mapping) in order to restore consistency of plastic conditions.

In the NISA program, two different schemes are being used based on the initial status of the Gauss point. These are the generalized trapezoidal scheme and a modified explicit/implicit scheme that utilizes the generalized trapezoidal and the Euler schemes [2.24]. The modified scheme in NISA, generally improves the convergence and reduces the number of iterations with various degrees depending on the spread of the plastic zone and the type of loading.

(a) Generalized Trapezoidal Scheme

In this scheme [2.23] the stress is updated in two steps. The first one is simply an elastic predictor which is obtained by integrating the elastic constitutive equations at the current status, e.g. if temperature dependent, to move the stress state from ${}^t\sigma$ to ${}^{t+\Delta t}\sigma^*$ (The star is an indication that the stress state is an intermediate or temporary one). The elastic predictor ${}^{t+\Delta t}\sigma^*$ is then subsequently mapped onto a suitable yield surface. First the stresses ${}^{t+\Delta t}\sigma^*$ are projected along the initial plastic flow direction ${}^t\mathbf{d}$ to obtain ${}^{t+\Delta t}\sigma^{(i-1/2)}$ and then mapped onto the updated flow direction ${}^{t+\Delta t}\mathbf{d}$ to obtain ${}^{t+\Delta t}\sigma^{(i)}$. The scheme is implicit, and iterations are required till the assumed final flow direction matches, within the required accuracy, with the final flow direction calculated from the final projected stress state.

Figure 2.12 shows a graphical presentation of the scheme and the following equations formulate the procedure:

$${}^{t+\Delta t}\sigma^* = {}^t\sigma + {}^t\mathbf{C}^e : ({}^{t+\Delta t}\Delta\varepsilon - {}^{t+\Delta t}\Delta\varepsilon^{th}) \quad (2.357)$$

$${}^{t+\Delta t}\sigma = {}^{t+\Delta t}\sigma^* - (d\lambda) {}^t\mathbf{C}^e : [(1-\alpha) {}^t\mathbf{d} + \alpha {}^{t+\Delta t}\mathbf{d}] \quad (2.358)$$

$${}^{t+\Delta t}\varepsilon^p = {}^t\varepsilon^p + (d\lambda)[(1-\alpha)\mathbf{d}_n + \alpha\mathbf{d}_{n+1}] \quad (2.359)$$

where σ is the Cauchy stress tensor, C^e is the elastic constitutive tensor, $\Delta \varepsilon$ is the incremental total strain tensor, $\Delta \varepsilon^{th}$ is the incremental thermal strain tensor, $d\lambda$ is the incremental plastic parameter or proportionality constant, \mathbf{d} is the plastic flow direction, α is a factor ranging from 0.0 to 1.0, and ':' indicates double product of the two tensors, i.e. $C:\varepsilon \equiv C_{ijkl}\varepsilon_{kl}$.

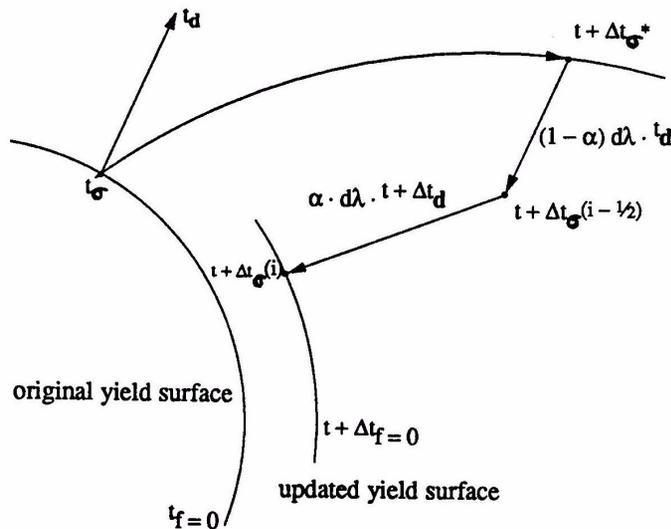


Figure 2.12: Geometric representation of the generalized trapezoidal rule

(b) Mixed Implicit/Explicit Scheme

As indicated above the generalized trapezoidal scheme is based on an elastoplastic operator split method which starts with an elastic predictor to obtain the trial stress state ${}^{t+\Delta t} \sigma^*$. If the Gauss point is already plastic and it is loading, i.e., $\Delta f > 0$, then the elastic predictor will over estimate the stress status. In general situations where the plastic zone is not spread very much and the loading is general, the return mapping procedure is shown to converge in reasonable number of iterations. If the plastic zone is reasonably spread in the structure, it is noted that the return mapping procedure, though converges, takes much more, and sometimes unreasonable, number of iterations. This is thought to be due to the difficulty in

readjusting or redistributing the over estimated stress increments when many neighboring Gauss points have the same condition. Hence, it is thought that if we better approximate the trial stress increment, the elastoplastic constitutive matrix may be available at the time of stress calculations and, therefore, a very good estimate for the stress increment may be obtained. This, however, eliminates the possibility of using the above generalized schemes, since they are based on elastic predictors, and a simple explicit e.g., Euler forward scheme, may be used. The governing equations may be written as,

$${}^t\sigma^{(i)} = \sigma + \int_{t_\varepsilon}^{t + \Delta t_\varepsilon^{(i)}} \mathbf{C}^{ep} : \quad (2.360)$$

where, \mathbf{C}^{ep} is the elastoplastic constitutive tensor at the start of the iteration. The solution is thus obtained by simply integrating over a sufficient number of subincrements.

2.7.4 Hyperelastic and Rubber-Like Material Model

1. Introduction

The name 'hyperelasticity' is given to elastic materials for which there exists an 'elastic potential function' or a 'strain energy density function', W , a scalar function of the strain tensor or the deformation tensor, whose derivatives with respect to the strain components determine the stress components [2.25]. Hyperelastic constitutive relations are usually suitable for describing rubber-like material behavior, and they possess the elastic characteristics of complete reversibility of the deformation as well as the characteristic that the response is independent of the deformation history. Experience suggests that rubber like solids are isotropic relative to the undistorted state. This assumption of isotropy has played an important role in simplifying the associated mathematical formulation. In order to construct a strain energy function consistent with the observed behavior of rubber like solids when deformed isothermally, it is usually thought expedient to use certain strain invariants. The most common invariants used are the invariants of the Green - Lagrange strain tensor (\mathbf{E}) or, more conveniently, the invariants of the Cauchy - Green deformation tensor (\mathbf{C}). The principal invariant of the deformation tensor and their derivatives with respect to the deformation components may be expressed in the following forms:

$$I_1 = tr(\mathbf{C}) = C_{ii} \quad (2.361)$$

$$I_2 = | \mathbf{C} | \text{tr}(\mathbf{C})^{-1} \equiv \frac{1}{2}(I_1^2 - C_{ij}C_{ij}) \quad (2.362)$$

$$I_3 = | \mathbf{C} | \quad (2.363)$$

and,

$$\frac{\partial I_1}{\partial \mathbf{C}} = \mathbf{I} \quad (2.364)$$

$$\frac{\partial I_2}{\partial \mathbf{C}} = I_1 \mathbf{I} - \mathbf{C} \quad (2.365)$$

$$\frac{\partial I_3}{\partial \mathbf{C}} = |\mathbf{C}| \mathbf{C}^{-1} \quad (2.366)$$

where, \mathbf{C} is the Cauchy - Green deformation tensor, given by,

$$\mathbf{C} = \mathbf{F}^T \mathbf{F} \quad (2.367)$$

and \mathbf{F} is the deformation gradient tensor, and \mathbf{I} is the identity tensor or matrix. See [Section 2.3](#) for the definition of \mathbf{F} and \mathbf{E} .

The strain energy function is generally function of I_1 , I_2 and I_3 . For an incompressible material, the third strain invariant is, however, unity and the strain energy density may be considered as function of I_1 and I_2 .

For the problem statement, we assume a body with an initial volume V , fixed along part of its boundary, S_u and has a prescribed surface traction on a part of its boundary, S_σ . The total potential of the body may be expressed as:

$$W = \int_V W(I_1, I_2) d'V - \int_V \mathbf{b} \cdot \mathbf{u} d^{\circ}V - \int_{S_\sigma} \mathbf{t} \cdot \mathbf{u} d^{\circ}S \quad (2.368)$$

where, \mathbf{b} is the prescribed body forces per unit undeformed volume, \mathbf{t} is the prescribed surface traction per unit deformed surface area. The resulting deformation field \mathbf{u} is a stationary point of the functional W , satisfying the incompressibility constrain,

$$|\mathbf{F}| - 1 = 0 \quad (2.369)$$

where, $|\mathbf{F}|$ is the determinant of the deformation gradient tensor \mathbf{F} .

Initiated by the original work of Mooney [2.26] and Rivlin [2.27], many authors have been concerned with the form of the strain energy function that accounts for different material types of incompressible behavior of various rubber materials, see for example references [2.31-2.37].

The strain energy density function given by Equation (2.368) implies that the addition of an arbitrary hydrostatic pressure will affect the stress but it will not affect the strains. For finite compressibility condition, the strain energy may be regarded as a function of the third invariant, i.e.,

$$W(I_1, I_2, I_3) = \hat{W}(I_1, I_2) + \chi G(I_3) \quad (2.370)$$

where, \hat{W} is given by Equation (2.368) and χ is a constant. If G is taken to be (Equation (2.369)) the formulation reduces to a penalty approach.

Once the strain energy function is established, the stress and the constitutive tensors are easily defined by,

$$\mathbf{S}_{ij} = \frac{\partial W}{\partial \mathbf{E}_{ij}} \text{ and} \quad (2.371)$$

$$\mathbf{D}_{ijkl} = \frac{\partial \mathbf{S}_{ij}}{\partial \mathbf{E}_{kl}} = \frac{\partial^2 W}{\partial \mathbf{E}_{ij} \partial \mathbf{E}_{kl}} \quad (2.372)$$

The main concern in the formulation of hyperelastic materials is the method by which the nonlinear constraint (Equation (2.369)) is applied. In NISA, there are two approaches; the first is an average constraint approach, and the second utilizes the bubble-function [see Ref. 2.38 for details]. In what follows, we briefly describe the above two approaches and present the wide range of strain energy functions available in NISA.

2. Average Constraint Approach

In this approach, the constraint function is *weakened* on the element level. This is done by choosing a constraint space with less degrees of freedom than the actual problem, and hence the term 'average or approximate constraint' arises for this procedure. Then the penalty function is introduced on the average constraint on the element level. This amounts to finding a stationary point for the functional,

$$W' = W + \frac{1}{2\varepsilon} \int_V \left[\frac{1}{V} \int_V (I_3 - 1) dV \right]^2 dV \quad (2.373)$$

where the constraint has been first *averaged* on the element level through the first volume integral. The averaging of the constraint equation may be dealt by projecting the constraint function over the base of the constraint space. The basis functions are defined on the reference element level and the constrained space has less degrees of freedom than those for the space of the actual problem.

The average constraint method is directly applicable to low - as well as high - order elements and seems to be independent of element distortions or the amount of strain the element is undergoing.

3. Bubble Function Approach

In a multi-field approach, e.g. independent variables are the displacements u and the pressures p , let m stand for degrees of freedom of the original variable, i.e., u and n stands for the degrees of freedom for the auxiliary variable, i.e. p . Zienkiewicz et al. [2.39] define a 'freedom number' β as,

$$\beta = \frac{\Delta m}{\Delta n} \quad (2.374)$$

by which the element efficiency and performance may be assessed. For example, for optimum element performance in two-dimensional problems, the freedom number β is required to be greater than 2.

The idea of using bubble functions to handle the incompressibility behavior seems to be logical extension of the above discussion. The bubble function will have the effect of increasing the degrees of freedom for the original variable u over those for the auxiliary variable p and hence increasing the freedom number β . This extension is simple and straight forward in linear analysis. In nonlinear analysis, however, the bubble function or nodeless degrees of freedom have to be condensed and recovered in each iteration. Moreover, these nodeless degrees of freedom should not be associated with any external or residual forces. This is a key point for the convergence of this approach. To achieve this, a brief description is given below, [See reference [2.38] for details].

Starting with the final equilibrium equations in the partitioned form,

$$\begin{bmatrix} \mathbf{K}_{uu} & \mathbf{K}_{ua} \\ \mathbf{K}_{au} & \mathbf{K}_{aa} \end{bmatrix} \begin{Bmatrix} \Delta \mathbf{u} \\ \Delta \mathbf{a} \end{Bmatrix} = \begin{Bmatrix} \Delta \mathbf{F}_u \\ \Delta \mathbf{F}_a \end{Bmatrix} \quad (2.375)$$

where, $\Delta \mathbf{u}$ is the incremental displacement vector, $\Delta \mathbf{a}$ is the incremental displacement vector corresponding to the bubble function degrees of freedom. D.O.F., $\Delta \mathbf{F}_u$, and $\Delta \mathbf{F}_a$, are the corresponding incremental force vectors. A special loop has to be performed in the stress calculations routine in which the recovered condensed D.O.F. $\Delta \mathbf{a}$,

$$\Delta \mathbf{a}^{(i)} = -(\mathbf{K}_{aa}^{-1} \mathbf{K}_{au})^{(i)} \Delta \mathbf{u}^{(i)} \quad (2.376)$$

have to be iterated upon till the equivalent nodal forces $\Delta \mathbf{F}_a$, become zero, where,

$$\begin{Bmatrix} \Delta \mathbf{F}_u \\ \Delta \mathbf{F}_a \end{Bmatrix}^{(i+1)} = \begin{bmatrix} \mathbf{K}_{uu} & \mathbf{K}_{ua} \\ \mathbf{K}_{au} & \mathbf{K}_{aa} \end{bmatrix}^{(i+1)} \begin{Bmatrix} \Delta \mathbf{u} \\ \Delta \mathbf{a} \end{Bmatrix} = \int_V \mathbf{B}^{(i+1)T} \mathbf{S}^{(i+1)} dV \quad (2.377)$$

where, \mathbf{S} is the element stress vector.

The corrective displacements $\Delta \mathbf{a}^{(i+1)}$ are calculated from,

$$\Delta \mathbf{a}^{(i+1)} = (\mathbf{K}_{aa}^{-1})^{(i+1)} (\Delta \mathbf{F}_a^{(i+1)} - \mathbf{K}_{au}^{(i+1)} \Delta \mathbf{u}^{(i)}) \quad (2.378)$$

4. Various Strain Energy Forms

(a) *Neo-Hookean model* [2.28]

Based on the Gaussian statistics to a simple model of a network of long chain molecules, W may be expressed as,

$$W = C_{11}(I_1 - 3) \tag{2.379}$$

This form is a reasonable first order approximation, which by a proper choice of the constant, gives a reasonably good correlation between experiments and theory in a small range of deformations for uniaxial and equibiaxial tension of a sheet. However, the theory does not yield good correlation with experiments at the large strain condition.

(b) *Mooney - Rivlin model* [2.29]

Based on more extensive experiments with rubbers, Rivlin suggests a more general form of strain energy density function,

$$W = C_{11}(I_1 - 3) + C_{21}(I_2 - 3) \tag{2.380}$$

This form has been widely used in problems with large deformation since it is a higher order theory that still maintains a certain amount of mathematical simplicity. Although for uniaxial extension and simple shear, the Mooney assumption is not a bad approximation, it gives poor correlation with biaxial and pure shear experiments.

(c) *Generalized Mooney - Rivlin model* [2.30]

In this assumption, the strain-energy-density function can be expanded as an infinite series in terms of I_1 and I_2 . Thus,

$$W = \sum_{i=1}^{N_1} C_{1i}(I_1 - 3)^i + \sum_{j=1}^{N_2} C_{2j}(I_2 - 3)^j \tag{2.381}$$

Although this is a more general and higher order approximation, correlation between [Equation \(2.381\)](#) and experimental data is not significantly better than the Mooney form, and analysis of this form as well as getting the constants experimentally become much more difficult as the order of approximation increases.

(d) *Swanson model* [2.31]

In order to achieve good material representation with a minimum number of terms, Swanson suggested the use of non-integer powers as opposed to integer ones in the above [Equation \(2.379\)](#) to [Equation \(2.381\)](#),

$$\begin{aligned}
 W = & \sum_{i=1}^{N_1} \frac{3}{2} \frac{C_{1i}}{(1 + \alpha_i)} \left(\frac{I_i}{3}\right)^{1 + \alpha_i} \\
 & + \sum_{j=1}^{N_2} \frac{3}{2} \frac{C_{2j}}{(1 + \beta_j)} \left(\frac{I_2}{3}\right)^{1 + \beta_j}
 \end{aligned}
 \tag{2.382}$$

where, α_i and β_j are constants that are not necessarily integers.

Accuracy of the material representation can be enhanced by simply including more terms in the series.

(e) *Blatz - Ko model* [2.32]

The nearly incompressible material idealization suggested by Blatz and Ko is,

$$W = \frac{1}{2} \mu \left[(I_1 - 3) + \frac{2}{\beta} \left(I_3^{\frac{\beta}{2}} - 1 \right) \right]
 \tag{2.383}$$

where, μ and β are material constants.

Because of its simplicity and its ability to represent the salient characteristics of rubber behavior based on easily obtained test data, Blatz - Ko model may be well suited to practical computation. However, it shares the common limitation of the two terms Mooney - Rivlin idealization, i.e., real material behavior can be represented over a wide range of deformation only in a qualitative fashion.

(f) *IHT model* [2.33]

Using network theory with a non-Gaussian statistics, Isihara, Hashitsume and Tatibane suggested the form,

$$W = C_{11}(I_1 - 3) + C_{12}(I_1 - 3)^2 + C_{21}(I_2 - 3) \quad (2.384)$$

The above equation gives a fair correlation with the equibiaxial experiments. However, the uniaxial response obtained using the same constants is much poorer representation than the neo - Hookean or Mooney - Rivlin model.

(g) *Biderman model* [2.34]

Biderman suggested the following form,

$$W = C_{11}(I_1 - 3) + C_{12}(I_1 - 3)^2 + C_{13}(I_1 - 3)^3 + C_{21}(I_2 - 3) \quad (2.385)$$

The model shows good correlation between experiment and theory for uniaxial tension, uniaxial compression and pure shear condition, but gives poor correlation under equibiaxial condition.

(h) *Klosner - Segal model* [2.35]

In this model, the strain energy function takes the form,

$$W = C_{11}(I_1 - 3) + C_{21}(I_2 - 3) + C_{22}(I_2 - 3)^2 + C_{23}(I_2 - 3)^3 \quad (2.386)$$

(i) *Hart - Smith model* [2.36]

Recognizing the shortcoming of the polynomial form presented above, Hart and Smith suggested an exponential form as,

$$W = G_1 \int e^{C_{12}(I_1-3)^2} dI_1 + C_{21} \ln \frac{I_2}{3} \quad (2.387)$$

where, G_1, C_{12}, C_{21} are material constants.

This model gives good correlation with the data for small and moderate strains, and the resulting stress-strain curve has the correct trend at high strains. However, it is not suitable to represent the behavior of synthetic rubber at moderate strain.

(j) *Alexander model* [2.37]

On the basis of his experiments on neoprene, Alexander constructed the following form,

$$W = G_1 \int e^{C_{12}(I_1-3)^2} dI_1 + C_{21}(I_2-3) + C_{22} \ln \frac{(I_2-3) + G_2}{G_2} \quad (2.388)$$

where, G_1, C_{12}, C_{22} and G_2 are constants.

2.7.5 Constitutive Relations for Thermo-Elastic-Plastic Creep

The plasticity matrix \mathbf{D}_p , is defined as,

$$\mathbf{D}_p = \frac{1}{A} \mathbf{D}^e \mathbf{a}^T \mathbf{a} \mathbf{D}^e \quad (2.389)$$

The scalar A , the vector \mathbf{a} and the matrix \mathbf{D}^e are defined in [Equation \(2.349\)](#), [Equation \(2.350\)](#) and [Equation \(2.351\)](#). The elasto-plasticity matrix \mathbf{D}_{ep} , is given as,

$$\mathbf{D}_{ep} = \mathbf{D}^e - \mathbf{D}_p \quad (2.390)$$

The thermo-elastic-plastic constitutive equation is written as,

$$\begin{aligned} d\sigma = \mathbf{D}_{ep} d\varepsilon - \mathbf{D}_{ep} \left(\alpha dT + \frac{\partial(\mathbf{D}^e)^{-1}}{\partial T} \sigma dT + \frac{\partial(\mathbf{D}^e)^{-1}}{\partial \dot{\varepsilon}} \sigma d\dot{\varepsilon} \right) \\ - \mathbf{D}^e \frac{a}{A} \left(\frac{\partial f}{\partial T} dT \right) \left(\frac{\partial f}{\partial \dot{\varepsilon}} d\dot{\varepsilon} \right) \end{aligned} \quad (2.391)$$

where, the total incremental strain is given by,

$$d\varepsilon = d\varepsilon_{ep} + d\varepsilon_c \quad (2.392)$$

and $d\varepsilon_{ep}$ is the incremental thermo-elastic-plastic strain vector and $d\varepsilon_c$ is, the incremental creep strain vector.

The incremental creep strain can be expressed in terms of a creep potential function [2.55] as,

$$\dot{\varepsilon}_c = \dot{\beta} \frac{\partial \psi(\sigma)}{\partial \sigma} \quad (2.393)$$

where, β is a positive parameter depending on the loading history and $\psi(\sigma)$ the creep potential function similar to plastic potential function f . For a material obeying von Mises criterion, initially isotropic and homogeneous, the incremental creep strain vector is expressed as,

$$d\varepsilon_c = \frac{3}{2} \frac{d\bar{\varepsilon}_c}{\bar{\sigma}} \sigma' \quad (2.394)$$

where, $d\bar{\varepsilon}_c$ is the incremental equivalent effective creep strain, $\bar{\sigma}$ is the effective stress and σ' is the deviatoric stress components.

The constitutive equation for thermo elastic-plastic creep behavior is written as,

$$d\sigma = \mathbf{D}_{ep} d\varepsilon - \mathbf{D}_{ep} \left(\alpha d\mathbf{T} + \frac{\partial(\mathbf{D}^e)^{-1}}{\partial\mathbf{T}} \sigma d\mathbf{T} + \frac{\partial(\mathbf{D}^e)^{-1}}{\partial\dot{\varepsilon}} \sigma d\dot{\varepsilon} + d\varepsilon_c \right) \quad (2.395)$$

$$- \mathbf{D}^e \frac{a}{A} \left(\frac{\partial f}{\partial\mathbf{T}} d\mathbf{T} + \frac{\partial f}{\partial\dot{\varepsilon}} d\dot{\varepsilon} \right)$$

Creep behavior is regarded as a time-dependent plasticity and is a transient problem. A proper time integration scheme is essential to achieve both convergence and stability of the solution. An explicit/implicit time integration scheme is implemented for integrating the transient creep behavior.

2.7.6 Constitutive Relations for Elasto-Plastic-Cracking concrete material model

Description of constitutive material model: The constitutive model is as developed by Channakeshava and Sundara Raja Iyengar [Ref. 2.64]. In this model a simple method of including multiple cracking at the same point is incorporated and nonlinearities due to concrete cracking in tension, strain softening in tension, aggregate interlock, multiple cracking and plasticity at crack tip, plasticity of reinforcement steel and Tension stiffening are considered. The validity of representing the cracks using the smeared cracking concept in finite element model is assumed.

It is also assumed that the total strains in concrete can be resolved into an elastic component, a cracking component and a plastic component. Hence, the following expression,

$$\{\Delta\varepsilon\} = \{\Delta\varepsilon^e\} + \{\Delta\varepsilon^{cr}\} + \{\Delta\varepsilon^p\} \quad (2.396)$$

where, $\{\Delta\varepsilon\}$ is the total strain incremental vector

$\{\Delta\varepsilon^e\}$ is the elastic component

$\{\Delta\varepsilon^{cr}\}$ is the cracking component

$\{\Delta\varepsilon^p\}$ is the plastic component.

The elastic component of the strain increment is given by,

$$\{\Delta\varepsilon^e\} = [D^e]^{-1} \{\Delta\sigma\} \quad (2.397)$$

$[D^e]$ is the elastic constitutive matrix (refer [Table 2.1](#))

$\{\Delta\sigma\}$ is the total stress increment vector.

The crack strain increments in global directions $\{\Delta\boldsymbol{\varepsilon}^{cr}\}$ are obtained by assembling the transformed components of the local crack strain increments $\{\boldsymbol{e}_i^{cr}\}$ of each crack.

$$\{\Delta\boldsymbol{\varepsilon}^{cr}\} = \sum_{i=1}^n [\boldsymbol{N}_i][\Delta\boldsymbol{e}_i^{cr}] \quad (2.398)$$

n is the number of cracks

$[\boldsymbol{N}_i]$ is the transformation matrix,

$$[\boldsymbol{N}_i] = \begin{bmatrix} \cos^2\theta_i & -\sin\theta_i & \cos\theta_i \\ \sin^2\theta_i & \sin\theta_i & \cos\theta_i \\ \sin 2\theta_i & \cos^2\theta_i & \sin^2\theta_i \end{bmatrix} \quad (2.399)$$

where, θ_i is the crack angle of the i th crack (see [Figure 2.13](#)). The local crack strain increment vector is given by,

$$\left\{ \Delta\boldsymbol{e}_i^{cr} \right\} = \left\{ \Delta\boldsymbol{e}_{nn_i}^{cr}, \Delta\boldsymbol{e}_{nt_i}^{cr} \right\}^T$$

where, $\left\{ \Delta\boldsymbol{e}_{nn_i}^{cr} \right\}$ is the normal crack strain increment and $\left\{ \Delta\boldsymbol{e}_{nt_i}^{cr} \right\}$ is the shear crack strain increment.

The crack interface stress increments $\left\{ \Delta \mathbf{s}_i^{cr} \right\}$ are in equilibrium with the total stress increments and given as,

$$\left\{ \Delta \mathbf{s}_i^{cr} \right\} = [\mathbf{N}_i]^T \{ \Delta \boldsymbol{\sigma} \} \quad (2.400)$$

where,

$$\left\{ \Delta \mathbf{s}_i^{cr} \right\} = \left\{ \Delta \sigma_{nn_i}^{cr}, \Delta \sigma_{nt_i}^{cr} \right\}^T \quad (2.401)$$

It is assumed that crack stress increment and crack strain increments are related through the crack matrix $[\mathbf{D}_i^{cr}]$.

$$\left\{ \Delta \mathbf{s}_i^{cr} \right\} = [\mathbf{D}_i^{cr}] \left\{ \Delta \mathbf{e}_i^{cr} \right\} \quad (2.402)$$

The crack stiffness matrix is given by,

$$[\mathbf{D}_i^{cr}] = \begin{bmatrix} \mathbf{D}_c & 0 \\ 0 & \mathbf{G}_c \end{bmatrix}$$

where,

\mathbf{D}_c is calculated by one of the three methods below:

1. $D_c = D_c^o$ is the slope of the crack normal stress-normal strain plot for opening cracks
2. The slope D_c^o is related to the slope (E_T) of the strain softening curve and is given by the expression,

$$D_c^o = -\frac{EE_T}{E - E_T}$$

where, E is the young's modulus of concrete.

3. $D_c^o = \frac{f_{ct}^2 H}{2G_f}$

where, f_{ct} is the tensile strength of concrete.

G_f is the fracture energy.

H is the width of the element/Gauss point over which the crack is assumed to be fractured.

$D_c = D_c^s$ is the secant stiffness for closing and closed cracks

G_c is the crack interface shear stiffness and is equal to a fraction of the uncracked shear modulus. It represents the shear transfer capability due to aggregate interlock,

$$G_c = \frac{\beta G}{1 - \beta}$$

where,

G is the uncracked (original) shear modulus

β is the shear retention factor

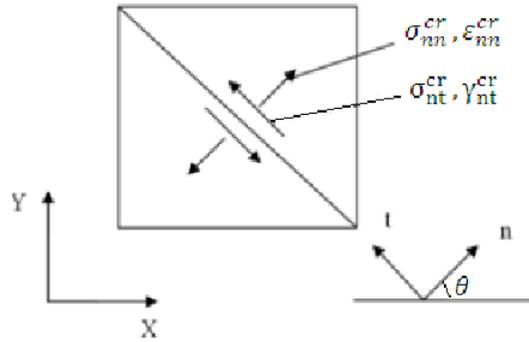


Figure 2.13: Crack Configuration

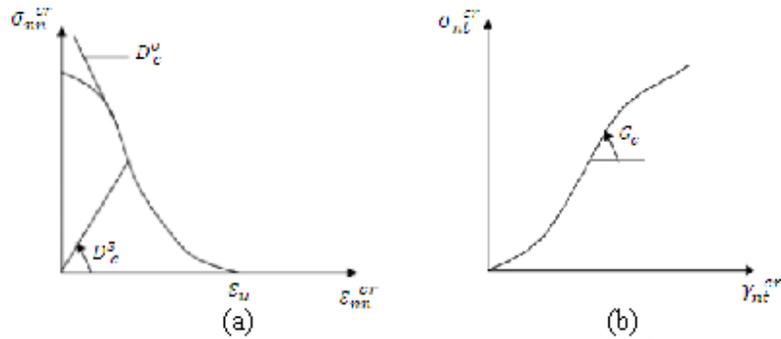


Figure 2.14: Crack modulus. (a) Normal stress-normal strain crack modulus and (b) Shear stress-shear strain crack modulus

By inverting the eqn. (2.399) and substituting from eqn. (2.397),

$$\left\{ \Delta \mathbf{e}_i^{cr} \right\} = [\mathbf{D}_i^{cr}]^{-1} [\mathbf{N}_i]^T \{ \Delta \boldsymbol{\sigma} \}$$

Multiplying by $[\mathbf{N}_i]$, summing up over the number of cracks and substituting from eqn. (2.395), the crack strain components in the global directions are given by,

$$\{\Delta \varepsilon^{cr}\} = \left[\sum_{i=1} [D_i^{cr}]^{-1} [N_i]^T [N_i]^T \right] \{\Delta \sigma\} \quad (2.403)$$

The plastic component of the strain increment is given by,

$$\{\Delta \varepsilon^p\} = d\lambda \left\{ \frac{\partial \mathbf{g}}{\partial \sigma} \right\} \quad (2.404)$$

where, $d\lambda$ is a scalar proportionality factor

$\mathbf{g} = \mathbf{g}(\sigma)$ is the plastic flow potential

If a flow rule based on a function $f(\sigma, k)$ is assumed, we have, $f(\sigma, k) = 0$,

where, k is a hardening parameter.

Continuity condition implies that $df = 0$.

$$\left\{ \frac{\partial f}{\partial \sigma} \right\}^T \{\Delta \sigma\} + \left\{ \frac{\partial f}{\partial k} \right\} d\kappa = 0$$

$$\left\{ \frac{\partial f}{\partial \sigma} \right\}^T \{\Delta \sigma\} = \left(-\frac{1}{d\lambda} \frac{\partial f}{\partial k} d\kappa \right) d\lambda = A d\lambda$$

By substituting in the eqn.(2.393), and rearranging the terms, we get $d\lambda$ as,

$$d\lambda = \frac{\left\{ \frac{\partial f}{\partial \sigma} \right\}^T [\mathbf{D}^{ecr}] \{ \Delta \varepsilon \}}{A + \left\{ \frac{\partial f}{\partial \sigma} \right\}^T [\mathbf{D}^{ecr}] \left\{ \frac{\partial \mathbf{g}}{\partial \sigma} \right\}}$$

where,

$$[\mathbf{D}^{ecr}] = \left[[\mathbf{D}^e]^{-1} + \sum_{i=1}^n [N_i] [\mathbf{D}_i^{cr}]^{-1} [N_i]^T \right]^{-1}$$

The total stress increment vector can be expressed as,

$$\{ \Delta \sigma \} = [\mathbf{D}^{ecr}] \left[[I] - \frac{\left\{ \frac{\partial f}{\partial \sigma} \right\} \left\{ \frac{\partial f}{\partial \sigma} \right\}^T [\mathbf{D}^{ecr}]}{A + \left\{ \frac{\partial f}{\partial \sigma} \right\}^T [\mathbf{D}^{ecr}] \left\{ \frac{\partial \mathbf{g}}{\partial \sigma} \right\}} \right] \{ \Delta \varepsilon \} \quad (2.405)$$

where,

$[I]$ is a 3×3 unit matrix.

Failure Criteria

Cracking failure

In tension, concrete is assumed to crack along the direction normal to the principal stress direction, when the tensile stress at that point exceeds the tensile strength. Subsequently, this direction is assumed fixed. Another crack is allowed to occur at an angle to the first crack. Opening and closing of cracks are considered. A crack is assumed to be closed when the normal crack strain component reduces to zero value. However, unlike many other investigations [Ref. 2.63], the material is not assumed to heal completely and any tensile stress greater than the previous maximum value before the closing of cracks, is not permitted.

2.8 NISA Elements

2.8.1 Concentrated Mass Elements (NKTP = 25-30)

The concentrated mass elements in NISA include 2-D mass (NKTP = 25), 3-D mass (NKTP = 26), 2-D general mass with different masses in X- and Y- directions (NKTP = 27), 3-D general mass with different masses in X-, Y- and Z- directions (NKTP = 28), 2-D general mass with rotary inertia (NKTP = 29), and 3- D general mass with rotary inertia (NKTP = 30).

Concentrated mass elements may be used to represent inertia properties and to apply loads due to body forces. These elements possess no structural stiffness. The mass matrix for the 3-D general mass element with rotary inertia effects (NKTP = 30), associated with 6 DOFs (3 translations and 3 rotations) is given by,

$$\mathbf{M} = \text{diag}(M_x, M_y, M_z, I_{xx}, I_{yy}, I_{zz}) \quad (2.406)$$

where,

M_x, M_y, M_z : masses in the X, Y, and Z directions respectively. Normally M_y and M_z , are equal to M_x , unless it is desired that the resistance to acceleration be orientation dependent.

I_{xx}, I_{yy}, I_{zz} : mass moments of inertia about the element X, Y and Z axes, respectively. The element coordinate axes are located at the mass center of gravity and are parallel to the global Cartesian coordinate axes. The moments of inertia are defined as,

$$I_{xx} = \int (y^2 + z^2) dm; \quad I_{yy} = \int (x^2 + z^2) dm; \quad I_{zz} = \int (x^2 + y^2) dm, \quad (2.407)$$

where, x, y and z are measured from the mass center of gravity.

The mass matrix given in Equation (2.406) is specialized for the NKTP = 25-29 elements according to the DOF for each element.

Body force vectors for the above elements are available for gravity loading or linear acceleration, centrifugal loading due to angular velocity and tangential loads due to angular acceleration. See Section 2.2 for discussion of the equations of body force calculations.

2.8.2 Spring, Spar and Cable Elements (NKTP = 14-22, 38 and 45)

(i) Shape Functions

This is a group of line elements that may be used for 1-D, 2-D and 3-D analyses. Each element is a straight one with two nodes (NORDR = 1). The cable element (NKTP = 45) is a tension only element. For nonlinear analysis, both cable and spar elements have initial tension capabilities. The spring elements may be used as translational and/or rotational springs.

The axial displacement in these elements is governed by the shape function equation,

$$\mathbf{u} = \sum_{k=1}^2 N_k \bar{\mathbf{u}}^k \quad (2.408)$$

where,

$$\begin{aligned} N_k &= \left(1 - \frac{x}{L}\right) \text{ for } k = 1 \\ &= \frac{x}{L} \quad \text{for } k = 2 \end{aligned} \quad (2.409)$$

and L is the element length.

In case of the spar (NKTP = 14, 15) and cable (NKTP = 45) elements, the element cross-section area varies linearly along the element axis according to the equation,

$$A = \left(1 - \frac{x}{L}\right)A_1 + \frac{x}{L}A_2 \quad 0 \leq x \leq L \quad (2.410)$$

where, A_1 and A_2 are the cross-sectional areas at the two end nodes 1 and 2, respectively.

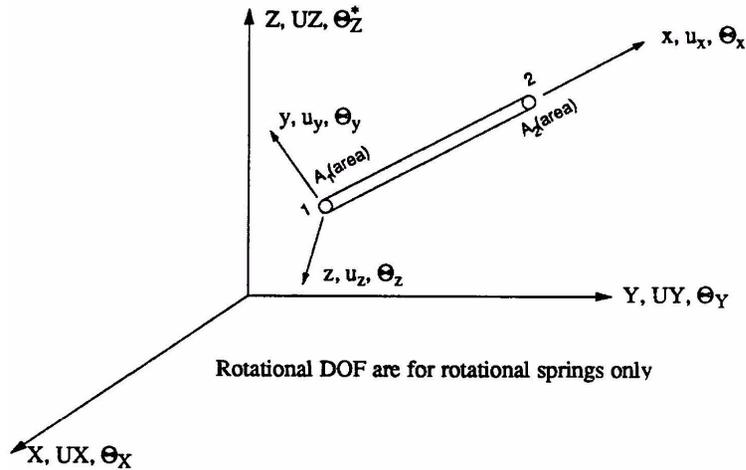


Figure 2.15: General element configuration (NKTP = 14-22, 38 and 45)

(ii) Element Characteristic Matrices

Element matrices are generated explicitly in the element local axes and transformed to global coordinates before assembly. Denoting the element nodal displacement vector by,

$$\mathbf{u}^e = [u_1 \ v_1 \ w_1 \ u_2 \ v_2 \ w_2]^T \quad (2.411)$$

then, the element stiffness matrix in element local axes for the 3-D spar and cable elements is given by,

$$\mathbf{K}' = \frac{E(A_1 + A_2)}{2L} \begin{bmatrix} 1 & 0 & 0 & -1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -1 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \quad (2.412)$$

where, E is the modulus of elasticity of the element material.

The element stiffness in global coordinates may be obtained by the transformation matrix \mathbf{T} . For 3-D elements, the matrix \mathbf{T} is a 3×3 matrix whose columns represent the cosine directions of the local axes with respect to the global axes. The global element stiffness matrix, \mathbf{K} , is then obtained by the transformation,

$$\mathbf{K} = \mathbf{R} \mathbf{K}' \mathbf{R}^T \quad (2.413)$$

where,

$$\mathbf{R} = \begin{bmatrix} \mathbf{T} & \mathbf{O} \\ \mathbf{O} & \mathbf{T} \end{bmatrix} \quad (2.414)$$

and \mathbf{O} is a 3×3 or 2×2 null matrix for 3-D and 2-D elements, respectively.

The element consistent mass matrix is obtained in global coordinates by assuming a linear displacement field (Equation (2.409)). For 3-D spar elements, this is given by,

$$\mathbf{M} = \frac{\rho L}{12} \begin{bmatrix} \alpha_1 & 0 & 0 & \beta & 0 & 0 \\ 0 & \alpha_1 & 0 & 0 & \beta & 0 \\ 0 & 0 & \alpha_1 & 0 & 0 & \beta \\ \beta & 0 & 0 & \alpha_2 & 0 & 0 \\ 0 & \beta & 0 & 0 & \alpha_2 & 0 \\ 0 & 0 & \beta & 0 & 0 & \alpha_2 \end{bmatrix} \quad (2.415)$$

where,

$$\alpha_1 = 3A_1 + A_2 \quad (2.416)$$

$$\alpha_2 = A_1 + 3A_2 \quad \text{and} \quad (2.417)$$

$$\beta = (A_1 + A_2) \quad (2.418)$$

where, ρ is the material density. For spring elements (NKTP = 17, 18, 21, 22 and 38), the spring constants replace the coefficient multiplier in Equation (2.412) in each specific direction and a null mass matrix, Equation (2.415) is assumed. Such elements may be used in dynamic analysis provided that it is connected to other elements with non-zero masses.

As an example to the stiffness matrix for spring elements, we consider the 3-D general spring for which the stiffness matrix may be expressed as,

$$\mathbf{K}' = \begin{bmatrix} \mathbf{K}'_s & -\mathbf{K}'_s \\ -\mathbf{K}'_s & \mathbf{K}'_s \end{bmatrix} \quad \text{and} \quad (2.419)$$

$$\mathbf{K}'_s = \text{diag}(k_x, k_y, k_z, k_{rx}, k_{ry}, k_{rz}) \quad (2.420)$$

where, k_x , k_y and k_z , are translational spring constants in the local x , y and z axes, respectively, and k_{rx} , k_{ry} and k_{rz} are the torsional spring constants in local x , y and z axes, respectively.

In buckling and nonlinear analysis, the initial stress or geometric stiffness matrix for the 3-D spar and cable elements in the element local coordinates is given by,

$$\mathbf{K}'_G = \frac{P}{L} \begin{bmatrix} 1 & 0 & 0 & -1 & 0 & 0 \\ 0 & 1 & 0 & 0 & -1 & 0 \\ 0 & 0 & 1 & 0 & 0 & -1 \\ -1 & 0 & 0 & 1 & 0 & 0 \\ 0 & -1 & 0 & 0 & 1 & 0 \\ 0 & 0 & -1 & 0 & 0 & 1 \end{bmatrix} \quad (2.421)$$

where, P is the axial force in the element. For spring elements, (NKTP = 17-22 and 38) the initial stress stiffness matrix is assumed to be *null*. The elements may be used, however, in buckling analysis provided that it is connected to other elements with non-zero geometric stiffness matrices.

(iii) Force-Deflection Curve (for NKTP-17 and NKTP-21).

For nonlinear analysis using the spring element (NKTP = 17 and NKTP-21), the user inputs the spring characteristics in terms of a force-deflection curve. Various types of data fitting are available in NISA. These are the piecewise linear, [Figure 2.16\(a\)](#), the cubic spline, and the arbitrary polynomial fit.

Two types of unloading are available for nonlinear spring; conservative and non-conservative unloading. If unloading is conservative, [Figure 2.16\(b\)](#), there will be no energy losses and the loading/unloading cycles will always stay on the originally defined force deflection curve. For non-conservative unloading, the element will experience energy losses and the cycles for loading/unloading will not stay on the originally defined force-deflection curve. Two case of non-conservative unloading are described:

- The unloading path does not cross the zero force line, [Figure 2.16](#); in this case the unloading path follows the slope of the original loading portion. If reloading occurs, it will follow the same path of unloading in an opposite direction i.e, from point 2 to 3 on the curve of [Figure 2.16](#), and then continues the original loading path, point 2 to 4 on the curve.
- The unloading path crosses the zero force line, [Figure 2.16](#); in this case, similar to the one above, the unloading path follows the slope of the original loading portion, points 2 to 3 on the curve, [Figure 2.16](#). After crossing the zero force line, the unloading path follows the negative or compressive loading portion of the curve, points 3 to 4 to 5, i.e., an origin shift will be applied to the negative portion of the curve. If re-loading occurs from point 5, it follows similar behavior of the original unloading and it goes through points 5-6-7 and 8.

The stiffness of the nonlinear spring element is history dependent and the instantaneous stiffness obtained from the force-deflection curve during loading/unloading is used to calculate the pertinent stiffness matrix.

(iv) Element Load Vectors

For spar and cable elements, load vector due to body forces is available for linear or gravity acceleration, centrifugal loading due to angular velocity and tangential loads due to angular acceleration. See [Section 2.2.1](#) for discussion of the equations for body force calculations. Load vector due to thermal effects is also available for spar and cable elements. Temperature variation along the element axial direction is assumed to be linear,

$$T(x) = \left(1 - \frac{x}{L}\right)T_1 + \frac{x}{L}T_2 \quad (2.422)$$

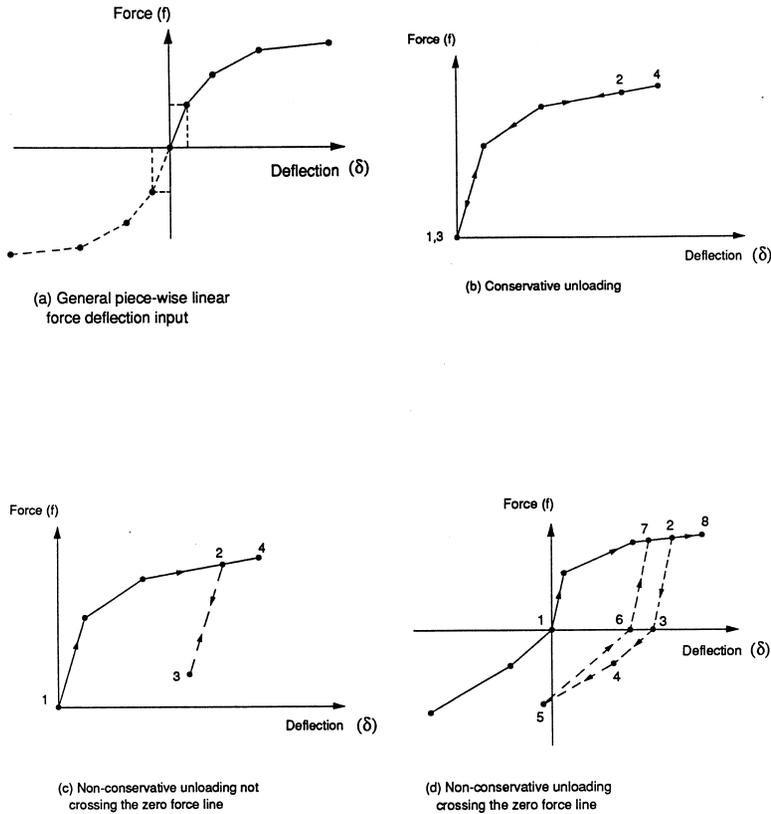


Figure 2.16: Force-deflection curve and types of loading for spring elements, NKTP = 17 and NKTP = 21

where, T_1 and T_2 are the nodal temperatures at the two end points. Therefore, the element thermal load vector, $\{f\}^{th}$ in the element local coordinates is given by,

$$\{f\}^{th} = \left(\frac{E\alpha}{6}\right) [-a \ 0 \ 0 \ a \ 0 \ 0]^T \quad (2.423)$$

where,

$$a = T_1(2A_1 + A_2) + T_2(A_1 + A_2) - 3T_{sfree}(A_1 + A_2) \quad (2.424)$$

a is the coefficient of thermal expansion, E is Young's modulus and T_{sfree} is the stress-free temperature.

No pressure load vector is available for spar and cable elements.

Once the element load vector in element local coordinates, f' , is established, a simple vector transformation gives the load vector in global coordinates as,

$$f = Rf' \quad (2.425)$$

where, the transformation matrix R is defined in [Equation \(2.414\)](#).

2.8.3 2-D Solid Elements (NKTP = 1, 2, and 3)

(i) Introduction

The 2-D solid elements in NISA include plane stress elements (NKTP = 1), plane strain elements (NKTP = 2) and axisymmetric ring elements (NKTP = 3). The element may be shaped as a 4 to 12 node quadrilateral, or as a 3 or 6 node triangle depending on the selected NORDR value. Each node has two degrees of freedom u_x and u_y . The state of stress is characterized by three components: σ_{xx} , σ_{yy} , and τ_{xy} , for the plane stress, NKTP = 1, elements (with $\sigma_{zz} = 0$) and by four components: σ_{xx} , σ_{yy} , σ_{zz} , and τ_{xy} for the plane strain, ($\varepsilon_{zz} = 0$), NKTP = 2, and the axisymmetric, NKTP = 3, elements. In axisymmetric elements, the axis of rotation is considered to be the y-axis whereas the circumferential direction is considered to be the z-axis.

Dynamic capabilities are available for the elements including, eigenvalue, transient dynamic, frequency response, random vibration and shock spectrum analyses. Consistent or

lumped mass formulation are available. Direct transient dynamic analysis is also available for the elements.

In nonlinear analysis, geometric nonlinearity analysis with large displacements, rotations and strains is available. The choice of total or updated Lagrangian formulation with deformation dependent loading is available. In material nonlinearity, elastoplastic, hyperelastic and creep models are available.

A bubble function for the lower order 4 node elements, NORDR = 12, is used to enhance the element performance in bending problems.

(ii) Shape Functions

The elements are isoparametric with the following shape functions for both displacements and coordinates:

$$\begin{aligned}
 u &= \sum_{i=1}^n N_i(\xi, \eta) \bar{u}_i & , v &= \sum_{i=1}^n N_i(\xi, \eta) \bar{v}_i \\
 x &= \sum_{i=1}^n N_i(\xi, \eta) \bar{x}_i & , y &= \sum_{i=1}^n N_i(\xi, \eta) \bar{y}_i
 \end{aligned}
 \tag{2.426}$$

where, \bar{u}_i, \bar{v}_i are the nodal point displacements in x and y directions, respectively, \bar{x}_i, \bar{y}_i are the nodal coordinates, and $N_i(\xi, \eta)$ are the shape functions given in [Table 2.2](#) and [Table 2.3](#) [2.40] and n is the number of nodes. The element geometry and coordinate system are shown in [Figure 2.17](#), for both the quadrilateral and the triangular elements

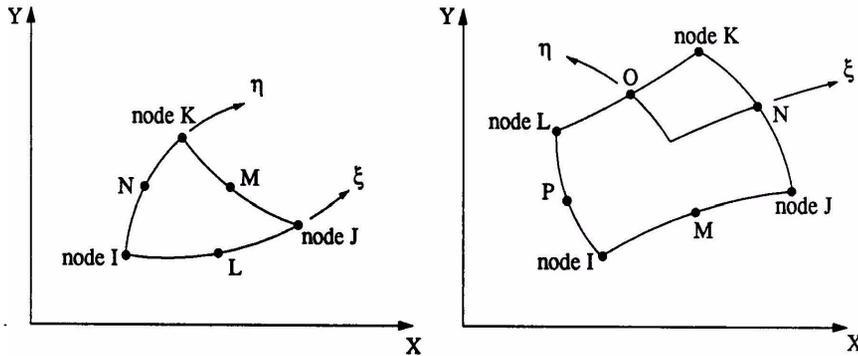


Figure 2.17: Geometry and coordinate system for 2-D elements (Note: substitute the numbers 1,2,... for I, J,... depending on the NORDR values)

It should be noted that for the triangular elements, the area or natural coordinates are used. These are defined by the following relations for a typical point *P* (see [Figure 2.18](#)).

Table 2.2: Interpolation functions of 4- and 8-node quadrilateral 2-D elements

Shape Function*		include only if the mid-side node exists			
		node <i>M</i>	node <i>N</i>	node <i>O</i>	node <i>P</i>
$N_I(\xi, \eta) =$	$\frac{1}{4}(1 + \xi)(1 + \eta)$	$-\frac{1}{2}N_M$	0	0	$-\frac{1}{2}N_P$
$N_J(\xi, \eta) =$	$\frac{1}{4}(1 - \xi)(1 + \eta)$	$-\frac{1}{2}N_M$	$-\frac{1}{2}N_N$	0	0
$N_K(\xi, \eta) =$	$\frac{1}{4}(1 - \xi)(1 - \eta)$	0	$-\frac{1}{2}N_N$	$-\frac{1}{2}N_O$	0
$N_L(\xi, \eta) =$	$\frac{1}{4}(1 + \xi)(1 - \eta)$	0	0	$-\frac{1}{2}N_O$	$-\frac{1}{2}N_P$
$N_M(\xi, \eta) =$	$\frac{1}{2}(1 - \xi^2)(1 + \eta)$	0	0	0	0
$N_N(\xi, \eta) =$	$\frac{1}{2}(1 - \xi)(1 - \eta^2)$	0	0	0	0

Shape Function*		include only if the mid-side node exists			
		node <i>M</i>	node <i>N</i>	node <i>O</i>	node <i>P</i>
$N_O(\xi, \eta) =$	$\frac{1}{2}(1 - \xi^2)(1 - \eta)$	0	0	0	0
$N_P(\xi, \eta) =$	$\frac{1}{2}(1 + \xi)(1 - \eta^2)$	0	0	0	0

* Note: The subscripts *I, J, K,...* should be substituted with the numbers 1, 2, 3,... depending on the node locations. For example, for NORDR = 1, *I, J, K, L* = 1, 2, 3, 4 and *M, N, O, P* do not exist.

Table 2.3: Interpolation functions of 3- and 6-node triangular 2-D elements

Shape Function*		include only if the mid-side node exists		
		node <i>L</i>	node <i>M</i>	node <i>N</i>
$N_I(\xi, \eta) =$	$1 - \xi - \eta$	$\frac{1}{2}N_L$	0	$\frac{1}{2}N_N$
$N_J(\xi, \eta) =$	ξ	$-\frac{1}{2}N_L$	$-\frac{1}{2}N_M$	0
$N_K(\xi, \eta) =$	η	0	$-\frac{1}{2}N_M$	$-\frac{1}{2}N_N$
$N_L(\xi, \eta) =$	$4\xi(1 - \xi - \eta)$	0	0	0
$N_M(\xi, \eta) =$	$4\xi\eta$	0	0	0
$N_N(\xi, \eta) =$	$4\eta(1 - \xi - \eta)$	0	0	0

* Note: For the 3-node triangular element, *I, J, K* = 1, 2, 3 and nodes *L, M, N* do not exist. For the triangular element *I, J, K* = 1, 3, 5 and *L, M, N* = 2, 4, 6

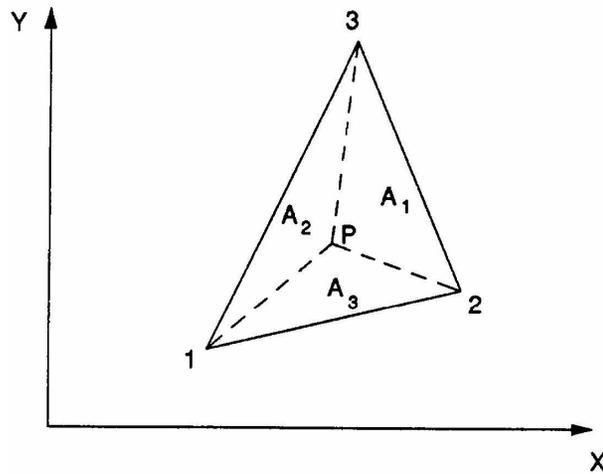


Figure 2.18: Area or natural coordinates for the triangular 2-D elements

$$L_1 = \frac{A_1}{A}, \quad L_2 = \frac{A_2}{A}, \quad L_3 = \frac{A_3}{A} \quad (2.427)$$

where, A is the total area of the triangle. Hence,

$$L_1 + L_2 + L_3 = 1 \quad (2.428)$$

and the relation between the area coordinates and the isoparametric coordinates is given by,

$$L_1 = 1 - \xi - \eta \quad L_2 = \xi \quad L_3 = \eta \quad (2.429)$$

from which [Table 2.3](#) may be expressed in terms of L_1 , L_2 and L_3 .

In the case of cubical shape functions, for elements with 4 nodes on any side, the following forms are used (refer to [Figure 2.19](#)):

$$N_i(\xi, \eta) = \frac{1}{32}(1 + \xi_o)(1 + \eta_o)(9(\xi^2 + \eta^2) - 10) \quad (2.430)$$

for $i = 1, 4, 7$ and 10

$$N_i(\xi, \eta) = \frac{9}{32}(1 + \xi_o)(1 + \eta^2)(1 + 9\eta_o) \quad (2.431)$$

for $i = 5, 6, 11$ and 12

$$N_i(\xi, \eta) = \frac{9}{32}(1 + \eta_o)(1 - \xi^2)(1 + 9\xi_o) \quad (2.432)$$

for $i = 2, 3, 8$ and 9

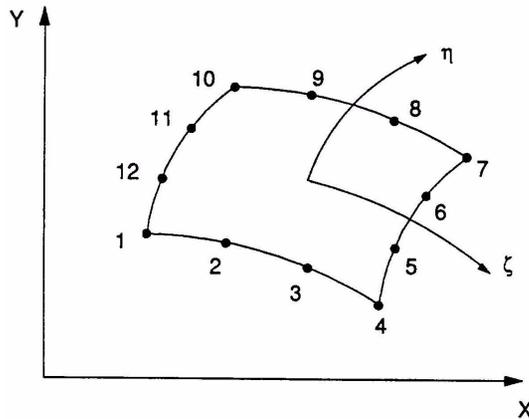


Figure 2.19: Geometry and coordinate system for elements with cubic shape functions

where,

$$\xi_o = \xi \xi_i \text{ and } \eta_o = \eta \eta_i \text{ for node } i \quad (2.433)$$

The eight node element is easily adaptable to crack-tip analysis by simply shifting the mid-side node to a quarter point location. Doing so, a singularity of the type $1/\sqrt{r}$, where, r is the distance measured from the corner node, is produced. This matches exactly the singular stress field around the crack-tip. The elements may be also used in transition regions of mesh grading. These transition elements produce better accuracy, however, when the element shape is nearly rectangular and when the non-corner nodes are located in their natural position, e.g., mid-side nodes at the center of the edge. Therefore, the user should use such transition elements with caution.

In the case of the linear 4-node element with NORDR = 12, two extra shape functions are added in both u and v directions to improve the element behavior in bending, and we have,

$$u = \sum_{i=1}^n N_i(\xi, \eta) \bar{u}_i + (1-\xi^2)a_1 + (1-\eta^2)a_3 \quad (2.434)$$

$$v = \sum_{i=1}^n N_i(\xi, \eta) \bar{v}_i + (1-\xi^2)a_2 + (1-\eta^2)a_4$$

where, a_1, a_2, a_3 and a_4 are internal or nodeless degrees of freedom that are eliminated on the element level. The element, as such, is incompatible because the nodeless degrees of freedom may be activated in one element but not in its neighboring elements.

The extra degrees of freedom are condensed out before the stiffness assembly and recovered at the stress calculation level. However, an extra iterative loop is needed, in nonlinear analysis, in order to ensure no load application at the nodeless degrees of freedom for the subsequent iterations.

(iii) Element Characteristics:

Element stiffness and Mass Matrices:

Starting from the element shape functions in the form,

$$x = \sum_{i=1}^n N_i \bar{x}_i, \quad y = \sum_{i=1}^n N_i \bar{y}_i$$

and

$$u = \sum_{i=1}^n N_i \bar{u}_i, \quad v = \sum_{i=1}^n N_i \bar{v}_i \quad (2.435)$$

The strains for the 2-D elements are given by,

For plane stress / plane strain case

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{xy} \end{Bmatrix} = \begin{Bmatrix} \partial u / \partial x \\ \partial v / \partial y \\ \partial u / \partial y + \partial v / \partial x \end{Bmatrix} \quad (2.436)$$

$$= \begin{bmatrix} \left[\begin{array}{c|c} \frac{\partial N_i}{\partial x} & \mathbf{0} \\ \hline \mathbf{0} & \frac{\partial N_i}{\partial y} \end{array} \right] & \dots \\ \dots & \left[\begin{array}{c|c} \frac{\partial N_i}{\partial y} & \frac{\partial N_i}{\partial x} \\ \hline \dots & \dots \end{array} \right] \end{bmatrix} \begin{Bmatrix} \vdots \\ \bar{u}_i \\ \vdots \\ \bar{v}_i \\ \vdots \end{Bmatrix} \quad (2.437)$$

$$= \mathbf{B} \bar{\mathbf{u}} \quad (2.438)$$

where, \mathbf{B} is the strain-displacement matrix. The derivatives in the \mathbf{B} matrix are evaluated using the Jacobian transformation as follow,

$$\frac{\partial}{\partial \xi} = \frac{\partial}{\partial x} \frac{\partial x}{\partial \xi} + \frac{\partial}{\partial y} \frac{\partial y}{\partial \xi}$$

$$\frac{\partial}{\partial \eta} = \frac{\partial}{\partial x} \frac{\partial x}{\partial \eta} + \frac{\partial}{\partial y} \frac{\partial y}{\partial \eta}$$

or,

$$\begin{Bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial x \partial y}{\partial \xi \partial \xi} \\ \frac{\partial x \partial y}{\partial \eta \partial \eta} \end{Bmatrix} \begin{Bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{Bmatrix}$$

$$\frac{\partial}{\partial \xi} = \mathbf{J} \frac{\partial}{\partial x} \quad (2.439)$$

from which,

$$\begin{Bmatrix} \partial/\partial x \\ \partial/\partial y \end{Bmatrix} = [J]^{-1} \begin{Bmatrix} \partial/\partial \xi \\ \partial/\partial \eta \end{Bmatrix} \quad (2.440)$$

where,

$$[J]^{-1} = \frac{1}{|J|} \begin{bmatrix} \frac{\partial y}{\partial \eta} & -\frac{\partial y}{\partial \xi} \\ -\frac{\partial x}{\partial \eta} & \frac{\partial x}{\partial \xi} \end{bmatrix} \quad (2.441)$$

Therefore,

$$\begin{Bmatrix} \partial N_i/\partial x \\ \partial N_i/\partial y \end{Bmatrix} = [J]^{-1} \begin{Bmatrix} \partial N_i/\partial \xi \\ \partial N_i/\partial \eta \end{Bmatrix} \quad (2.442)$$

It should be noted that in the case of plane stress, the strain in the third direction is calculated from,

$$\varepsilon_{zz} = \frac{-\nu}{E}(\sigma_{xx} + \sigma_{yy}) \quad (2.443)$$

For axial symmetry case (wherein x , y , and z represent radial, axial and circumferential directions, respectively),

$$\varepsilon = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{u}{x} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \end{Bmatrix}$$

$$= \begin{bmatrix} \left| \begin{array}{cc} \frac{\partial N_i}{\partial x} & 0 \\ 0 & \frac{\partial N_i}{\partial y} \end{array} \right| \dots \left\{ \begin{array}{c} \cdot \\ \cdot \\ \bar{u}_i \\ \cdot \\ \bar{v}_i \\ \cdot \end{array} \right\} \\ \dots \left| \begin{array}{cc} (N_i/x) & 0 \\ \frac{\partial N_i}{\partial y} & \frac{\partial N_i}{\partial x} \end{array} \right| \dots \left\{ \begin{array}{c} \cdot \\ \cdot \\ \bar{u}_i \\ \cdot \\ \bar{v}_i \\ \cdot \end{array} \right\} \end{bmatrix} \quad (2.444)$$

$$= \mathbf{B} \bar{\mathbf{u}}$$

where, the derivatives in the \mathbf{B} matrix are evaluated using the Jacobian transformation as discussed above.

The stiffness and the consistent mass matrices are then calculated from the following equations,

$$\mathbf{K} = \int_V \mathbf{B}^T \mathbf{C} \mathbf{B} dV \quad (2.445)$$

$$\mathbf{M} = \int_V \rho \mathbf{N}^T \mathbf{N} dV \quad (2.446)$$

where,

$$dV = t |J| d\xi d\eta \text{ for plane stress/strain cases,}$$

$$= 2\pi x |J| d\xi d\eta \text{ for axisymmetric case,}$$

$$t = \text{element thickness (} \equiv 1.0 \text{ for plane strain)}$$

$$x = \sum_{i=1}^n N_i \bar{x}_i$$

ρ : the material mass density

\mathbf{C} : the stress-strain constitutive matrix given in [Section 2.7.1](#)

In the case of elements with extra shape functions (bubble functions), NORDR = 12, the above equations take the following forms,

$$\boldsymbol{\varepsilon} = [\mathbf{B}, \mathbf{B}'] \begin{Bmatrix} \bar{\mathbf{u}} \\ \bar{\mathbf{a}} \end{Bmatrix} \quad (2.447)$$

where, \mathbf{B}' is the partition of the strain displacement matrix due to the extra shape functions and $\bar{\mathbf{a}}$ is the vector of the nodeless degrees of freedom.

The stiffness matrix of the element will thus take the form of,

$$[\mathbf{K}] = \begin{bmatrix} \mathbf{K}_{uu} & \mathbf{K}_{ua} \\ \mathbf{K}_{au} & \mathbf{K}_{aa} \end{bmatrix} \quad (2.448)$$

where,

\mathbf{K}_{uu} : is the regular stiffness matrix due to nodal D.O.F. (displacement),

$$\mathbf{K}_{uu} = \int_V \mathbf{B}^T \mathbf{C} \mathbf{B} dV \quad (2.449)$$

\mathbf{K}_{aa} : is an extra stiffness contribution due to nodeless D.O.F.

$$\mathbf{K}_{aa} = \int_V \mathbf{B}'^T \mathbf{C} \mathbf{B}' dV \quad (2.450)$$

\mathbf{K}_{ua} : is the coupling stiffness matrix between displacement and nodeless D.O.F.,

$$\mathbf{K}_{ua} = \int_V \mathbf{B}^T \mathbf{C} \mathbf{B}' dV \quad (2.451)$$

and

$$\mathbf{K}_{ua} = \mathbf{K}_{au}^T \quad (2.452)$$

It should be noted that the extra shape or 'bubble' functions allow the element to move more flexibly. This added flexibility leads to better simulation of bending behavior. The bubble functions, on the other hand, may cause the element to be 'incompatible' and it may fail the patch test (unless elements are shaped as parallelogram in plane stress/strain case, see reference [2.41]).

To avoid the above problem, the following numerical scheme [2.41] is utilized in NISA which leads to automatic satisfaction of the patch test for plane stress and plane strain cases. Somewhat similar treatment is provided for the axisymmetric case.

For this element to pass the patch test, we assume an imposed displacement field, $\bar{\mathbf{u}}_c$ which corresponds constant strain/stress state. The element equation becomes,

$$\begin{bmatrix} \mathbf{K}_{uu} & \mathbf{K}_{ua} \\ \mathbf{K}_{au} & \mathbf{K}_{aa} \end{bmatrix} \begin{Bmatrix} \bar{\mathbf{u}}_c \\ \mathbf{O} \end{Bmatrix} = \begin{Bmatrix} \mathbf{P} \\ \mathbf{O} \end{Bmatrix} \quad (2.453)$$

where, we assumed no loading for incompatible modes, and we require that the nodeless or incompatible D.O.F. to be zero, if the patch test is to be passed. The second of [Equation \(2.453\)](#) gives,

$$\mathbf{K}_{au} \bar{\mathbf{u}}_c = \mathbf{O} \quad (2.454)$$

which upon substitution of [Equation \(2.451\)](#) gives,

$$\left(\int_V \mathbf{B}'^T \mathbf{C} \mathbf{B} dV \right) \bar{\mathbf{u}}_c = \left(\int_V \mathbf{B}'^T dV \right) \sigma_c = 0 \quad (2.455)$$

where, σ_c , is the arbitrary constant stress state. Therefore, for plane elements we require the following condition for the automatic satisfaction of the patch test.

$$\int_{-1}^{+1} \int_{-1}^{+1} \mathbf{B}'^T \mathbf{t} |J| d\xi d\eta = 0 \quad (2.456)$$

Noting that \mathbf{B}' contains first powers of ξ and η , the satisfaction of Equation (2.456) is guaranteed if \mathbf{t} and $|\mathbf{J}|$ are constants. This is attained by calculating $|\mathbf{J}|$ at the centroid of the element, i.e., at $\xi = \eta = 0$, regardless of the actual coordinates of the Gauss integration point.

The nodeless D.O.F. are condensed on the element level according to the following static condensation scheme:

$$\begin{bmatrix} \mathbf{K}_{uu} & \mathbf{K}_{ua} \\ \mathbf{K}_{au} & \mathbf{K}_{aa} \end{bmatrix} \begin{Bmatrix} \bar{\mathbf{u}}_c \\ \bar{\mathbf{a}}_c \end{Bmatrix} = \begin{Bmatrix} \mathbf{p}_u \\ \mathbf{p}_a \end{Bmatrix} \quad (2.457)$$

The second of the above equations give,

$$\bar{\mathbf{a}} = \mathbf{K}_{aa}^{-1} (\mathbf{p}_a - \mathbf{K}_{au} \bar{\mathbf{u}}) \quad (2.458)$$

which upon substitution in the first equation gives,

$$\mathbf{K} \bar{\mathbf{u}} = \mathbf{p} \quad (2.459)$$

where,

$$\mathbf{K} = \mathbf{K}_{uu} - \mathbf{K}_{ua} \mathbf{K}_{aa}^{-1} \mathbf{K}_{au} \quad (2.460)$$

$$\mathbf{p} = \mathbf{p}_u - \mathbf{K}_{ua} \mathbf{K}_{aa}^{-1} \mathbf{p}_a \quad (2.461)$$

after which the element is treated as a regular element with only nodal D.O.F. In the stress calculations, the nodeless D.O.F. are recovered from Equation (2.458) and the strains are calculated, based on the recovered values, using Equation (2.447).

Note that [Equation \(2.457\)](#) to [Equation \(2.461\)](#) represent standard static condensation algorithm. For linear static analysis, no loads are applied to the nodeless degrees of freedom, i.e., $\mathbf{p}_a = \mathbf{0}$. However, in nonlinear static analysis, an iterative loop is performed to eliminate the residual load vector corresponding to the nodeless degrees of freedom.

Element Load Vectors

Following the development in [Section 2.2](#), the equivalent load vector of the element due to pressure on one or more of its sides is given by,

$$\mathbf{p}_s = \int_S \mathbf{N}^{sT} \mathbf{t}^s dS \quad (2.462)$$

where, \mathbf{N}^s is the element shape function calculated at the surface where pressure is acting and \mathbf{t}^s is the surface traction vector (due to pressure) on the pertinent surface. The surface traction vector may represent constant pressure on the element face or variable pressure specified by different pressure values at the nodes of the particular face of the element.

The equivalent load vector of the element due to body forces is given by,

$$\mathbf{p}_b = \int_V \mathbf{N}^T \mathbf{b} dV \quad (2.463)$$

where the integration is taken over the element volume and \mathbf{b} is the body force vector. For a detailed discussion of the body force vector \mathbf{b} , see [Section 2.2.1](#).

The equivalent element load vector due to thermal loading is calculated from the thermal strain vector,

$$\boldsymbol{\varepsilon}^{th} = [\alpha \Delta \mathbf{T}, \alpha \Delta \mathbf{T}, 0]^T \quad (2.464)$$

for isotropic plane stress case,

$$\boldsymbol{\varepsilon}^{th} = [\alpha (1 + \nu) \Delta \mathbf{T}, \alpha (1 + \nu) \Delta \mathbf{T}, 0]^T \quad (2.465)$$

for isotropic plane strain case, and

$$\varepsilon^{th} = [\alpha\Delta T, \alpha\Delta T, \alpha\Delta T, 0]^T \quad (2.466)$$

for isotropic axisymmetric case

where, α is the coefficient of thermal expansion, and ΔT is the difference between the actual Gauss point temperature and the stress-free reference temperature. For orthotropic materials. Equation (2.464) to Equation (2.466) take the form,

$$\varepsilon^{th} = [\alpha_x\Delta T, \alpha_y\Delta T, 0]^T \quad (2.467)$$

for orthotropic plane stress case,

$$\varepsilon^{th} = [(v_{zx}\alpha_z + \alpha_x)\Delta T, (v_{zx}\alpha_z + \alpha_y)\Delta T, 0]^T \quad (2.468)$$

for orthotropic plane strain case,

$$\varepsilon^{th} = [\alpha_x\Delta T, \alpha_y\Delta T, \alpha_z\Delta T, 0]^T \quad (2.469)$$

for orthotropic axisymmetric case,

In the plane strain case, the total ε_{zz} is ensured to be zero, from which the stress in the third direction σ_{zz} , is given by,

$$\sigma_{zz} = \nu(\sigma_{xx} + \sigma_{yy}) - E\alpha\Delta T \quad (2.470)$$

for isotropic case,

$$\sigma_{zz} = (\nu_{zx}\sigma_{xx} + \nu_{zy}\sigma_{yy}) - E_{zz}\alpha_z\Delta T \quad (2.471)$$

for orthotropic case,

On the other hand, for plane stress case, the stress in the third direction σ_{zz} is ensured to be zero, from which the strain is given by,

$$\varepsilon_{zz} = \frac{-\nu}{E}(\sigma_{xx} + \sigma_{yy}) + \alpha\Delta T \quad (2.472)$$

for isotropic case,

$$\varepsilon_{zz} = \frac{-1}{E_{zz}}(\nu_{zx}\sigma_{xx} + \nu_{zy}\sigma_{yy}) + \alpha_z\Delta T \quad (2.473)$$

for orthotropic case,

(iv) Nonlinear Analysis

Shape Functions

For nonlinear analysis, we assume both the displacement and the displacement increment to have the same shape function approximation, i.e.,

$${}^t u_i(\tau x) = \sum_{i=1}^n N_k(\tau x) {}^t \bar{u}_i^k, \quad (2.474)$$

$$\Delta {}^t u_i(\tau x) = \sum_{i=1}^n N_k(\tau x) \Delta {}^t \bar{u}_i^k \quad (2.475)$$

where, $\tau x = {}^o x$ for total Lagrangian formulation and $\tau x = {}^t x$ for updated Lagrangian formulation, and τ indicates the last updated configuration. Same shape functions given in [Table 2.2](#) and [Table 2.3](#) are used here, and also those for bubble function treatment.

Element Characteristics

Following the development given in [Section 2.3](#), the following characteristic matrices for the element are obtained:

The strain-displacement relationship is given by,

$$\Delta_o^t \mathbf{E} = ({}_o^t \mathbf{B}_{lin} + {}_o^t \mathbf{B}_{nl1}) \Delta_o^t \bar{\mathbf{u}} \quad (2.476)$$

where,

$${}_o^t \mathbf{B}_{lin} = \left[\begin{array}{c|cc|c} & {}_o^t N_{k,1} & 0 & \\ \cdots & 0 & {}_o^t N_{k,2} & \cdots \\ & \frac{{}_o^t N_k}{{}_o \bar{x}_1} & 0 & \\ & {}_o^t N_{k,2} & {}_o^t N_{k,1} & \end{array} \right] \quad (2.477)$$

where,

$${}_o^t N_{k,i} = \frac{\partial {}_o^t N_k}{\partial {}_o x_i} \quad (2.478)$$

$${}_o \bar{x}_i = \sum_{k=1}^n {}_o^t N_k {}_o \bar{x}^k \quad (2.479)$$

and the third row corresponds to the hoop strain ($\mathbf{E}_{\theta\theta}$, where, θ is the circumferential direction) and is only added for axisymmetric problems,

$${}^t\mathbf{B}_{nl1} = \left[\begin{array}{c|cc|c} \dots & l_{11} \frac{{}^tN_{k,1}}{o} & l_{21} \frac{{}^tN_{k,1}}{o} & \dots \\ & l_{12} \frac{{}^tN_{k,2}}{o} & l_{22} \frac{{}^tN_{k,2}}{o} & \\ & l_{33} \frac{{}^tN_k}{o_{\bar{x}_1}} & 0 & \\ \dots & (l_{11} \frac{{}^tN_{k,2}}{o} + l_{12} \frac{{}^tN_{k,1}}{o}) & (l_{21} \frac{{}^tN_{k,2}}{o} + l_{22} \frac{{}^tN_{k,1}}{o}) & \dots \end{array} \right] \quad (2.480)$$

where,

$$l_{ij} = \sum_{k=1}^n \frac{{}^tN_{k,j}}{o} \frac{{}^t\bar{u}_i^k}{o} \quad (2.481)$$

and the third row corresponds to $E_{\theta\theta}$ and is only added for axisymmetric problems.

In the case of updated Lagrangian formulation, the matrix ${}^t\mathbf{B}_{nl1}$ vanishes, and all other quantities are with respect to the most updated reference configuration, i.e., ${}^t\mathbf{N}$, ${}^t\mathbf{x}$, etc.

The element stiffness matrix is given by,

$${}^t\mathbf{K} = {}^t\mathbf{K}^{(1)} + {}^t\mathbf{K}^{(2)} + {}^t\mathbf{K}^{(3)} - {}^t\mathbf{K}^{(4)} \quad (2.482)$$

where the component stiffness matrices and, $\mathbf{K}^{(1)}$, $\mathbf{K}^{(2)}$, $\mathbf{K}^{(3)}$, and $\mathbf{K}^{(4)}$ defined in [Section 2.3](#). In these nonlinear stiffness contributions, the nonlinear strain-displacement transformation matrix ${}^t\mathbf{B}_{nl2}$ is given by,

The initial load or load correction matrix of [Equation \(2.484\)](#) is computed from the following matrices,

$${}^o\hat{\bar{\mathbf{S}}} = \begin{bmatrix} {}^o\bar{\mathbf{S}}_{(1)} & {}^o\bar{\mathbf{S}}_{(2)} & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & {}^o\bar{\mathbf{S}}_{(1)} & {}^o\bar{\mathbf{S}}_{(2)} \end{bmatrix} \quad (2.485)$$

and

$${}^t\hat{\bar{\mathbf{N}}} = \left[\begin{array}{c|cc|c} & {}^t\bar{\mathbf{N}}_k & 0 & \\ \dots & {}^t\bar{\mathbf{N}}_k & 0 & \dots \\ & 0 & {}^t\bar{\mathbf{N}}_k & \\ & 0 & {}^t\bar{\mathbf{N}}_k & \end{array} \right] \quad (2.486)$$

where the bar indicates that the quantity is calculated at the surface where the traction is applied, and $({}^o\bar{\mathbf{S}}_{(1)}, {}^o\bar{\mathbf{S}}_{(2)})$ are the components of the surface traction vector.

(v) Special Features

Bubble Functions

The nodeless or bubble function degrees of freedom, [Equation \(2.434\)](#), are extended to both geometric and material nonlinear analysis. A special loop has to be performed in the stress calculation routine in order to ensure that the equivalent forces on the bubble function DOF are zero. [Section 2.7.4](#) gives the outlines of this computation.

It is shown that the use of bubble functions in the linear elements (NORDR = 12) leads to almost identical results with the quadratic elements (NORDR = 2), even for considerable degree of nonlinearity, see for example problem 3.21 in the NISA Verification Problem Manual.

Stress Measures

The program provides the choice of printing the results in terms of the Cauchy stress measure (true stress) or the second Piola-Kirchhoff stress measure. The Cauchy stress tensor relates the force in the current configuration to the deformed area in the same configuration. In the second Piola-Kirchhoff stress measure, the force in the current configuration is first transformed in a manner similar to the transformation of a material vector, i.e.,

$$d\mathbf{p}' = {}^t\mathbf{F}^{-1} d\mathbf{p} \quad (2.487)$$

similar to,

$$d\mathbf{X} = {}^t\mathbf{F}^{-1} d{}^t\mathbf{x} \quad (2.488)$$

where, $d\mathbf{p}$ is the force in the current configuration, and \mathbf{F}^{-1} is the inverse of the deformation gradient tensor. The second Piola-Kirchhoff stress tensor, then, relates $d\mathbf{p}'$ to the undeformed original area. The relation between the two tensors is given by [Equation \(2.47\)](#)

2.8.4 3-D Solid Element (NKTP = 4)

(i) Introduction

The 3-D elements in NISA are based on a general state of stress characterized by six components, $(\sigma_{xx}, \sigma_{yy}, \sigma_{zz}, \tau_{xy}, \tau_{yz}, \tau_{zx})$. The elements have three degrees of freedom per node, (u_x, u_y, u_z) and they are suitable for modeling 3-D solid structures with general loadings. The elements can be shaped as an 8 or 20 node hexahedron (brick) element, a 6 or 15 node wedge element, or a 4 or 10 node tetrahedron element depending on the selected NORDR value.

Dynamic capabilities are available for the elements including, eigenvalue, transient dynamic, frequency response, random vibration and shock spectrum analyses. Consistent or lumped mass formulations are available and direct transient dynamic analysis is also available for these elements.

In nonlinear analysis, geometric nonlinearity analysis with large displacements, rotations and finite strains is available. The user may choose the type of formulation as total or updated Lagrangian formulation and the loading may be deformation dependent. Various material nonlinearity models are available including elastoplasticity, hyperelasticity and creep models.

A bubble function for the lower order 8 node element, NORDR = 1, is available for both linear and nonlinear analysis.

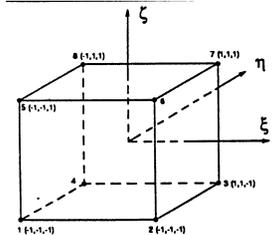
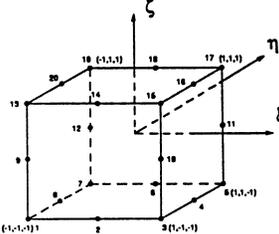
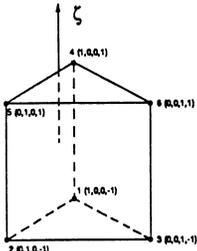
(ii) Shape Functions

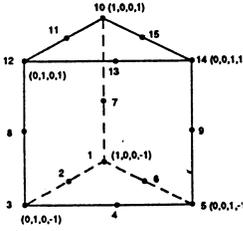
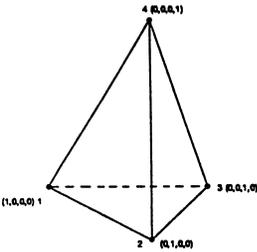
The elements are isoparametric with the following shape function relations for both displacements and coordinates,

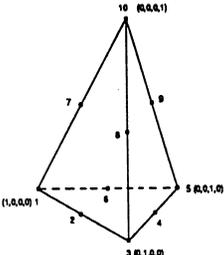
$$\begin{aligned}
 u &= \sum_{i=1}^n N_i(\xi, \eta, \zeta) \bar{u}_i \quad , \quad x = \sum_{i=1}^n N_i(\xi, \eta, \zeta) \bar{x}_i \\
 v &= \sum_{i=1}^n N_i(\xi, \eta, \zeta) \bar{v}_i \quad , \quad y = \sum_{i=1}^n N_i(\xi, \eta, \zeta) \bar{y}_i \\
 w &= \sum_{i=1}^n N_i(\xi, \eta, \zeta) \bar{w}_i \quad , \quad z = \sum_{i=1}^n N_i(\xi, \eta, \zeta) \bar{z}_i
 \end{aligned}
 \tag{2.489}$$

where, $\bar{u}_i, \bar{v}_i, \bar{w}_i$ are the nodal point displacements in $x, y,$ and z directions, respectively, $\bar{x}_i, \bar{y}_i, \bar{z}_i$ are the nodal point coordinates, and $N_i(\xi, \eta, \zeta)$ are the element shape functions. [Table 2.4](#) lists all shape functions for various 3-D element orders, [2.43]

Table 2.4: Interpolation functions for 3-D solid elements

Element Shape/NORDR	Shape Functions ⁺
 <p style="text-align: center;">8 node brick NORDR = 1, 12</p>	$N_i = \frac{1}{8}(1 + \xi_0)(1 + \eta_0)(1 + \zeta_0); i = 1, 2, 3, \dots, 8$
 <p style="text-align: center;">20 node brick NORDR = 2</p>	$N_i = \frac{1}{8}(1 + \xi_0)(1 + \eta_0)(1 + \zeta_0)(\xi_0 + \eta_0 + \zeta_0 - 2);$ $i = 1, 3, 5, 7, 13, 15, 17, 19$ $N_i = \frac{1}{4}(1 - \xi^2)(1 + \eta_0)(1 + \zeta_0) ; i = 2, 6, 14, 18$ $N_i = \frac{1}{4}(1 - \eta^2)(1 + \xi_0)(1 + \zeta_0) ; i = 4, 8, 16, 20$ $N_i = \frac{1}{4}(1 - \zeta^2)(1 + \xi_0)(1 + \eta_0) ; i = 9, 10, 11, 12$
 <p style="text-align: center;">6 node wedge NORDR = 10</p>	$N_i = \frac{1}{2}L_1(1 + \zeta_0) ; i = 1, 4$ $N_i = \frac{1}{2}L_2(1 + \zeta_0) ; i = 2, 5$ $N_i = \frac{1}{2}L_3(1 + \zeta_0) ; i = 3, 6$

Element Shape/NORDR	Shape Functions ⁺
$+\xi_0 = \xi\xi_i, \eta_0 = \eta\eta_i, \zeta_0 = \zeta\zeta_i$	
 <p data-bbox="257 664 396 716">15 node wedge NORDR = 11</p>	$N_i = \frac{1}{2}L_1(2L_1 - 1)(1 + \zeta_0) - \frac{1}{2}L_1(1 - \zeta^2); \quad i = 1, 10$ $N_i = \frac{1}{2}L_2(2L_2 - 1)(1 + \zeta_0) - \frac{1}{2}L_2(1 - \zeta^2); \quad i = 3, 12$ $N_i = \frac{1}{2}L_3(2L_3 - 1)(1 + \zeta_0) - \frac{1}{2}L_3(1 - \zeta^2); \quad i = 5, 14$ $N_i = 2L_1L_2(1 + \zeta_0) \quad ; \quad i = 2, 11$ $N_i = 2L_2L_3(1 + \zeta_0) \quad ; \quad i = 4, 13$ $N_i = 2L_3L_1(1 + \zeta_0) \quad ; \quad i = 6, 15$ $N_7 = L_1(1 - \zeta^2)$ $N_8 = L_2(1 - \zeta^2)$ $N_9 = L_3(1 - \zeta^2)$
 <p data-bbox="225 1407 410 1459">4 node tetrahedron NORDR = 20</p>	$N_1 = L_1$ $N_2 = L_2$ $N_3 = L_3$ $N_4 = L_4$

Element Shape/NORDR	Shape Functions ⁺
 <p data-bbox="270 595 448 651">10 node tetrahedron NORDR = 21</p>	$N_1 = L_1(2L_1 - 1) \quad ; \quad N_2 = 4L_1L_2$ $N_3 = L_2(2L_2 - 1) \quad ; \quad N_4 = 4L_2L_3$ $N_5 = L_3(2L_3 - 1) \quad ; \quad N_6 = 4L_3L_1$ $N_7 = 4L_1L_4 \quad ; \quad N_8 = 4L_2L_4$ $N_9 = 4L_3L_4 \quad ; \quad N_{10} = L_4(2L_4 - 1)$
$+ \xi_0 = \xi \xi_i, \quad \eta_0 = \eta \eta_i, \quad \zeta_0 = \zeta \zeta_i$	

It should be noted that for the wedge and tetrahedron elements, volume coordinates are used. These are defined in terms of the isoparametric coordinates as:

$$L_1 = 1 - \xi - \eta - \zeta$$

$$L_2 = \xi$$

$$L_3 = \eta$$

$$L_4 = \zeta$$

(2.490)

and

$$L_1 + L_2 + L_3 + L_4 = 1$$

(2.491)

In the case of the linear 8 node element with NORDR = 1, three extra shape functions are added in the u , v , and w directions to improve the element behavior in bending, and we have,

$$\begin{aligned}
 u &= \sum_{i=1}^n N_i(\xi, \eta, \zeta) \bar{u}_i + (1-\xi^2)a_1 + (1-\eta^2)a_4 + (1-\zeta^2)a_7 \\
 v &= \sum_{i=1}^n N_i(\xi, \eta, \zeta) \bar{v}_i + (1-\xi^2)a_2 + (1-\eta^2)a_5 + (1-\zeta^2)a_8 \\
 w &= \sum_{i=1}^n N_i(\xi, \eta, \zeta) \bar{w}_i + (1-\xi^2)a_3 + (1-\eta^2)a_6 + (1-\zeta^2)a_9
 \end{aligned} \tag{2.492}$$

where, a_1, a_2, \dots, a_9 , are internal or nodeless degrees of freedom that are eliminated on the element level. The element as such is incompatible because the nodeless degrees of freedom may be activated in one element but not in its neighboring elements.

The extra degrees of freedom are condensed out before the stiffness assembly and recovered at the stress calculation level. However, an extra iterative loop is required. In nonlinear analysis, in order to ensure no load application at the nodeless degrees of freedom for the subsequent iterations.

2.8.5 3-D General Shell Element (NKTP = 20)

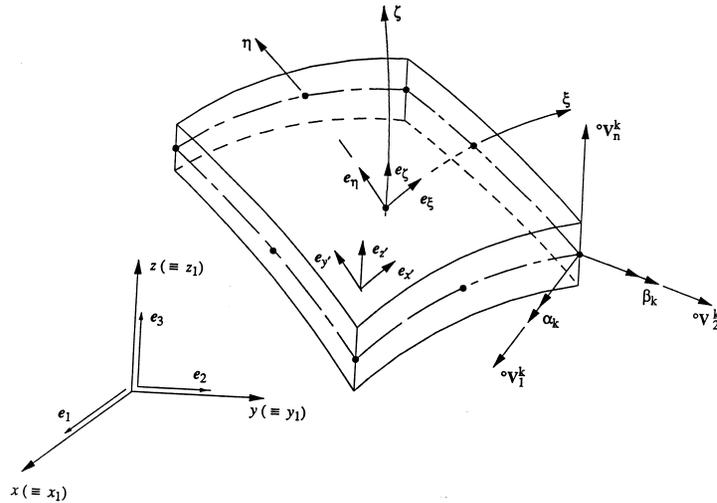


Figure 2.20: Coordinate systems for shell elements

(i) Introduction and Coordinate Systems

The following coordinate systems [Figure 2.20](#) are defined for later reference in linear and/or nonlinear analysis.

- Global Cartesian coordinate system, $(X_1, X_2, X_3) \equiv (X, Y, Z)$ with the basis $(\mathbf{e}_1, \mathbf{e}_2, \mathbf{e}_3) \equiv (\mathbf{e}_x, \mathbf{e}_y, \mathbf{e}_z)$. This global system is used to define nodal coordinates and displacements.

The Local Cartesian coordinate system with basis, $(\mathbf{e}_{x'}, \mathbf{e}_{y'}, \mathbf{e}_{z'})$ is used to define local stresses and strains at any point within the element and is defined as follows,

$$\mathbf{e}_{x'} = \frac{\partial \mathbf{x}}{\partial \xi} / \left\| \frac{\partial \mathbf{x}}{\partial \xi} \right\| \quad (2.493)$$

$$\frac{\partial \mathbf{x}}{\partial \xi} = \left[\frac{\partial x}{\partial \xi} \quad \frac{\partial y}{\partial \xi} \quad \frac{\partial z}{\partial \xi} \right]^T \quad (2.494)$$

$$\mathbf{e}_{z'} = \mathbf{e}_{x'} \times \mathbf{e}_{y'} \quad (2.495)$$

where,

$$\mathbf{e}_{\eta} = \left[\frac{\partial x}{\partial \eta} \quad \frac{\partial y}{\partial \eta} \quad \frac{\partial z}{\partial \eta} \right]^T \quad (2.496)$$

and finally,

$$\mathbf{e}_{y'} = \mathbf{e}_{z'} \times \mathbf{e}_{x'} \quad (2.497)$$

The local coordinate system varies throughout the element and the stress and strain tensors calculated in this system have to be transformed to the global Cartesian coordinate system through the following direction cosines tensor,

$$[T] = [\mathbf{e}_{x'} \quad \mathbf{e}_{y'} \quad \mathbf{e}_{z'}] = \begin{bmatrix} l_1 & l_2 & l_3 \\ m_1 & m_2 & m_3 \\ n_1 & n_2 & n_3 \end{bmatrix} \quad (2.498)$$

If the stress and the strain tensors are arranged in a vector form, the corresponding (6x6) constitutive matrix may be transformed from the local $(\mathbf{x}', \mathbf{y}', \mathbf{z}')$ coordinates to the global (x, y, z) coordinates through the following transformation matrix,

$$[Q] = \begin{bmatrix} Q_{11} & Q_{12} & Q_{13} & Q_{14} & Q_{15} & Q_{16} \\ Q_{21} & Q_{22} & Q_{23} & Q_{24} & Q_{25} & Q_{26} \\ Q_{31} & Q_{32} & Q_{33} & Q_{34} & Q_{35} & Q_{36} \\ Q_{41} & Q_{42} & Q_{43} & Q_{44} & Q_{45} & Q_{46} \\ Q_{51} & Q_{52} & Q_{53} & Q_{54} & Q_{55} & Q_{56} \\ Q_{61} & Q_{62} & Q_{63} & Q_{64} & Q_{65} & Q_{66} \end{bmatrix} \quad (2.499)$$

$$Q_{11} = l_1^2$$

$$Q_{12} = m_1^2$$

$$Q_{13} = n_1^2$$

$$Q_{14} = l_1 m_1$$

$$Q_{15} = m_1 n_1$$

$$Q_{16} = n_1 l_1$$

$$Q_{21} = l_2^2$$

$$Q_{22} = m_2^2$$

$$Q_{23} = n_2^2$$

$$Q_{24} = l_2 m_2$$

$$Q_{25} = m_2 n_2$$

$$Q_{26} = n_2 l_2$$

$$Q_{31} = l_3^2$$

$$Q_{32} = m_3^2$$

$$Q_{33} = n_3^2$$

$$Q_{34} = l_3 m_3$$

$$Q_{35} = m_3 n_3$$

$$Q_{36} = n_3 l_3$$

$$Q_{41} = 2l_1l_2$$

$$Q_{42} = 2m_1m_2$$

$$Q_{43} = 2n_1n_2$$

$$Q_{44} = l_1m_2 + l_2m_1$$

$$Q_{45} = m_1n_2 + m_2n_1$$

$$Q_{46} = n_1l_2 + n_2l_1$$

$$Q_{51} = 2l_2l_3$$

$$Q_{52} = 2m_2m_3$$

$$Q_{53} = 2n_2n_3$$

$$Q_{54} = l_2m_3 + l_3m_2$$

$$Q_{55} = m_2n_3 + m_3n_2$$

$$Q_{56} = n_2l_3 + n_3l_2$$

$$Q_{61} = 2l_3l_1$$

$$Q_{62} = 2m_3m_1$$

$$Q_{63} = 2n_3n_1$$

$$Q_{64} = l_3m_1 + l_1m_3$$

$$Q_{65} = m_3n_1 + m_1n_3$$

$$Q_{66} = n_3l_1 + n_1l_3$$

- Nodal Cartesian coordinate system with basis (V_1^k, V_2^k, V_n^k) at a nodal point k is attached to the shell middle surface. This system is used to express nodal rotations and is defined as follows,

$$V_n = \left(\frac{\partial \mathbf{x}}{\partial \xi} \right)^k \times \left(\frac{\partial \mathbf{x}}{\partial \eta} \right)^k / a \quad (2.500)$$

where,

$$a = \left\| \left(\frac{\partial \mathbf{x}}{\partial \xi} \right)^k \times \left(\frac{\partial \mathbf{x}}{\partial \eta} \right)^k \right\| \quad (2.501)$$

and

$$V_1 = \frac{\mathbf{e}_y \times V_n}{\|\mathbf{e}_y \times V_n\|} \quad (2.502)$$

where, \mathbf{e}_y is the unit vector in the global Y -direction. If V_n is parallel to \mathbf{e}_y , we define $V_1 = \mathbf{e}_z$. Finally,

$$V_2 = V_n \times V_1 \quad (2.503)$$

- Curvilinear coordinate system with basis $(\mathbf{e}_\xi, \mathbf{e}_\eta, \mathbf{e}_\zeta)$ at any point in the element. This system is used for local interpolation of strain components (shear and membrane strains) and is defined by,

$$(\mathbf{e}_\xi, \mathbf{e}_\eta, \mathbf{e}_\zeta) = \left(\frac{\partial \mathbf{x}}{\partial \xi}, \frac{\partial \mathbf{x}}{\partial \eta}, \frac{\partial \mathbf{x}}{\partial \zeta} \right) \quad (2.504)$$

(ii) Kinematics and Shape Functions

The coordinates of a generic point in the shell element, with n nodal points are given by,

$${}^t x_i = \sum_{k=1}^n N_k {}^t x_i^k + \frac{\zeta}{2} \sum_{k=1}^n t_k N_k {}^t V_{ni}^k \quad (2.505)$$

where,

${}^t x_i$: Cartesian coordinates (${}^t x_1, {}^t x_2, {}^t x_3$) of a generic point on the shell element at time t

${}^t x_i^k$: Cartesian coordinates of a nodal point k on the shell middle surface at time t

${}^t V_{ni}^k$: component of the unit vector V_n , in direction i and at nodal point k .

N_k : shape function at nodal point k , $N_k = N_k(\xi, \eta)$

t_k : thickness of the shell in the ζ direction and at nodal point k .

The displacement of a generic point at time t is given by,

$$\begin{aligned} {}^t u_i &= {}^t x_i - {}^o x_i \\ &= \sum_{k=1}^n N_k {}^t u_i^k + \frac{\zeta}{2} \sum_{k=1}^n t_k N_k ({}^t V_{ni}^k - {}^o V_{ni}^k) \end{aligned} \quad (2.506)$$

Assuming small displacements (linear analysis) and defining α_k, β_k to be the rotations of the normal vector ${}^o V_n^k$ about ${}^o V_1^k, {}^o V_2^k$, respectively, then (for small angles α_k, β_k),

$${}^t V_n^k - {}^o V_n^k = {}^o V_1^k \beta_k - {}^o V_2^k \alpha_k \quad (2.507)$$

and, therefore, the displacements will be given by,

$${}^t u_i = \sum_{k=1}^n N_k {}^t u_i^k + \frac{\zeta}{2} \sum_{k=1}^n t_k N_k (-{}^o V_{2i}^k \alpha_k + {}^o V_{1i}^k \beta_k) \quad (2.508)$$

The displacement derivatives with respect to isoparametric coordinates will be given by,

$$\begin{Bmatrix} \partial^t u_i / \partial \xi \\ \partial^t u_i / \partial \eta \\ \partial^t u_i / \partial \zeta \end{Bmatrix} = \sum_{k=1}^n \begin{bmatrix} N_{k,\xi} & N_{k,\xi} g_{1i}^k \zeta & N_{k,\xi} g_{2i}^k \zeta \\ N_{k,\eta} & N_{k,\eta} g_{1i}^k \zeta & N_{k,\eta} g_{2i}^k \zeta \\ 0 & N_k g_{1i}^k & N_k g_{2i}^k \end{bmatrix} \begin{Bmatrix} u_i^k \\ \alpha_k \\ \beta_k \end{Bmatrix} \quad (2.509)$$

where,

$$\mathbf{g}_1^k = -\frac{1}{2} t_k {}^o V_2^k \text{ and } \mathbf{g}_2^k = \frac{1}{2} t_k {}^o V_1^k \quad (2.510)$$

The global displacement derivatives are then obtained by the Jacobian transformation as,

$$\begin{Bmatrix} u_{i,x} \\ u_{i,y} \\ u_{i,z} \end{Bmatrix} = [\mathbf{J}]^{-1} \begin{Bmatrix} u_{i,\xi} \\ u_{i,\eta} \\ u_{i,\zeta} \end{Bmatrix} = \begin{bmatrix} x_{,\xi} & y_{,\xi} & z_{,\xi} \\ x_{,\eta} & y_{,\eta} & z_{,\eta} \\ x_{,\zeta} & y_{,\zeta} & z_{,\zeta} \end{bmatrix} \begin{Bmatrix} u_{i,\xi} \\ u_{i,\eta} \\ u_{i,\zeta} \end{Bmatrix} \quad (2.511)$$

where, the ‘,’ indicates derivatives, i.e., $u_{i,x} = \partial u_i / \partial x$, and the left superscript t is dropped for convenience.

The strains may be now calculated in either Cartesian systems, i.e. the global Cartesian coordinates (x, y, z) or the local Cartesian coordinates (x', y', z') .

In the global Cartesian system (x, y, z) , the strains are obtained directly from the global displacement derivatives, and the strain tensor, arranged in a vector form, is given by,

$$\{\varepsilon\} = \left[\varepsilon_{xx} \ \varepsilon_{yy} \ \varepsilon_{zz} \ \gamma_{xy} \ \gamma_{yz} \ \gamma_{zx} \right]^T \quad (2.512)$$

It should be noted that in this case, ε_{xx} , can not be ignored and to obtain the stresses, the shell element **D** matrix has to be transformed from the local coordinates (x', y', z') into the global ones (x, y, z) . This may be convenient in linear analysis when flat shells or plates are considered, in which case the transformation need only be evaluated once or at certain points.

In the local Cartesian system (x', y', z') , the global displacement derivatives are first transformed into local Cartesian axes as,

$$[\mathbf{u}',_{x'y'z'}] = \mathbf{T}^T [\mathbf{u},_{xyz}] [\mathbf{T}] \quad (2.513)$$

where, $[\mathbf{T}]$ is the transformation matrix given in [Equation \(2.498\)](#) and,

$$[\mathbf{u}',_{x'y'z'}] = \begin{bmatrix} u'_{,x'} & v'_{,x'} & w'_{,x'} \\ u'_{,y'} & v'_{,y'} & w'_{,y'} \\ u'_{,z'} & v'_{,z'} & w'_{,z'} \end{bmatrix} \quad (2.514)$$

where, u', v' and w' are the displacement components in the x', y', z' system. A similar expression for $[\mathbf{u}',_{xyz}]$, can be written. The strain vector in this coordinate system may be expressed as,

$$\{\varepsilon'\} = [\varepsilon_{x'x'} \ \varepsilon_{y'y'} \ \varepsilon_{z'z'} \ \gamma_{x'y'} \ \gamma_{y'z'} \ \gamma_{z'x'}]^T \quad (2.515)$$

where it is noted that $\varepsilon_{z'z'}$ is ignored. The constitutive matrix is then simply a 5×5 matrix in the local Cartesian system (x', y', z') in which $\sigma_{z'z'} = 0$.

This approach is essential if local interpolation of certain strain components is required, which is the case in nonlinear analysis.

(iii) Element Characteristics

Utilizing Equation (2.509) to Equation (2.511), we may write the global displacement derivatives in the following form:

$$\begin{Bmatrix} u_{i,x} \\ u_{i,y} \\ u_{i,z} \end{Bmatrix} = \sum_{k=1}^n \begin{bmatrix} N_{k,x} & \mathbf{g}_{1i}^k \mathbf{G}_1^k & \mathbf{g}_{2i}^k \mathbf{G}_1^k \\ N_{k,y} & \mathbf{g}_{1i}^k \mathbf{G}_2^k & \mathbf{g}_{2i}^k \mathbf{G}_2^k \\ N_{k,z} & \mathbf{g}_{1i}^k \mathbf{G}_3^k & \mathbf{g}_{2i}^k \mathbf{G}_3^k \end{bmatrix} \begin{Bmatrix} u_i^k \\ \alpha_k \\ \beta_k \end{Bmatrix} \quad (2.516)$$

where,

$$N_{k,i} = \mathbf{J}_{i1}^{-1} N_{k,\xi} + \mathbf{J}_{i2}^{-1} N_{k,\eta} \quad (2.517)$$

$$\mathbf{G}_i^k = \zeta(\mathbf{J}_{i1}^{-1} N_{k,\xi} + \mathbf{J}_{i2}^{-1} N_{k,\eta}) + \mathbf{J}_{i3}^{-1} N_k \quad (2.518)$$

The element B-matrix may be then constructed as,

$$\{\boldsymbol{\varepsilon}\} = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial w}{\partial z} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\ \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \\ \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \end{Bmatrix} = \sum_{k=1}^n [\mathbf{B}]^k \{\mathbf{u}\}^k \quad (2.519)$$

where,

$$\{\mathbf{u}\}^k = [u^k \ v^k \ w^k \ \alpha^k \ \beta^k]^T \quad (2.520)$$

and

$$[\mathbf{B}]^k = \begin{bmatrix} N_{k,1} & 0 & 0 & \mathbf{g}_{11}^k \mathbf{G}_1^k & \mathbf{g}_{21}^k \mathbf{G}_1^k \\ 0 & N_{k,2} & 0 & \mathbf{g}_{12}^k \mathbf{G}_2^k & \mathbf{g}_{22}^k \mathbf{G}_2^k \\ 0 & 0 & N_{k,3} & \mathbf{g}_{13}^k \mathbf{G}_3^k & \mathbf{g}_{23}^k \mathbf{G}_3^k \\ N_{k,2} & N_{k,1} & 0 & (\mathbf{g}_{12}^k \mathbf{G}_2^k + \mathbf{g}_{12}^k \mathbf{G}_1^k) & (\mathbf{g}_{21}^k \mathbf{G}_2^k + \mathbf{g}_{22}^k \mathbf{G}_1^k) \\ 0 & N_{k,3} & N_{k,2} & (\mathbf{g}_{12}^k \mathbf{G}_3^k + \mathbf{g}_{13}^k \mathbf{G}_2^k) & (\mathbf{g}_{22}^k \mathbf{G}_3^k + \mathbf{g}_{23}^k \mathbf{G}_2^k) \\ N_{k,3} & 0 & N_{k,1} & (\mathbf{g}_{11}^k \mathbf{G}_3^k + \mathbf{g}_{13}^k \mathbf{G}_1^k) & (\mathbf{g}_{21}^k \mathbf{G}_3^k + \mathbf{g}_{23}^k \mathbf{G}_1^k) \end{bmatrix} \quad (2.521)$$

where, u^k , v^k , and w^k are the global components of displacement at node k . It should be noted that the nodal variables α^k, β^k are then transformed to global rotations through the relation,

$$\begin{Bmatrix} \alpha \\ \beta \end{Bmatrix}^k = \begin{bmatrix} l_1 & m_1 & n_1 \\ l_2 & m_2 & n_2 \end{bmatrix} \begin{Bmatrix} \theta_x \\ \theta_y \\ \theta_z \end{Bmatrix}^k \quad (2.522)$$

where, (l_1, m_1, n_1) and (l_2, m_2, n_2) are the direction cosines of the vectors V_1 and V_2 , respectively, and $\theta_x, \theta_y, \theta_z$ are the rotations about the global axes.

With the above \mathbf{B} matrix, the constitutive matrix for the shell element should be also referred to the global Cartesian system. This is obtained by utilizing the transformation matrix in Equation (2.499) and,

$$[\mathbf{D}_{xyz}] = [\mathbf{Q}]^T [\mathbf{D}_{x'y'z'}] [\mathbf{Q}] \quad (2.523)$$

where the constitutive matrix in the local Cartesian system is given by,

$$[\mathbf{D}_{x'y'z'}] = \frac{E}{(1-\nu^2)} \begin{bmatrix} 1 & \nu & 0 & 0 & 0 & 0 \\ & 1 & 0 & 0 & 0 & 0 \\ & & 0 & 0 & 0 & 0 \\ & & & (\frac{1-\nu}{2}) & 0 & 0 \\ & & & & k(\frac{1-\nu}{2}) & 0 \\ \text{Sym.} & & & & & k(\frac{1-\nu}{2}) \end{bmatrix} \quad (2.524)$$

where the normal stress in the z' -direction is assumed to be zero, and k is the shear factor.

The stiffness and the mass matrices of the element are then obtained through the regular finite element equations,

$$\mathbf{K} = \int_V \mathbf{B}^T \mathbf{D} \mathbf{B} |J| d\xi d\eta d\zeta \quad (2.525)$$

$$\mathbf{M} = \int_V \mathbf{N}^T \rho \mathbf{N} |J| d\xi d\eta d\zeta \quad (2.526)$$

2.8.6 2-D Gap and Friction Element (NKTP = 49)

This is a 2-node nonlinear element which may be used to model node-to-node contact between two bodies with or without friction. [Section 4.37](#) gives input and output description for this element. The element configuration is shown in [Figure 2.21](#).

The force-deflection relationships in the normal and tangential directions n and t , respectively, are shown in [Figure 2.22](#). The gap has an initial width of g_0 (referred to in the element real constant table as GW). A negative value indicates an initial interference (or overlap). Coulomb law is used for friction. Frictionless contact may be modeled by specifying a zero coefficient of friction. It should be noted that no distinction is made in the element formulation between the static and kinetic coefficients of friction.

As shown in [Figure 2.22](#) the element consists of a pair of coupled nonlinear orthogonal springs, which are assumed to be very stiff relative to the bodies they are attached to [Ref. 2.55, 2.56]. Small springs, much softer than the stiff springs ($\mathbf{S}_n \ll \mathbf{K}_n, \mathbf{S}_t \ll \mathbf{K}_t$), may optionally be included to prevent rigid body motion initially, upon break in contact, or during relative sliding motion.

The element may assume an open status (no contact) or a closed status depending on the relative displacement in the normal direction. The closed status may be sticking or sliding depending on whether the friction limit $\mu |f_n|$ is reached, where μ is the coefficient of friction and f_n is the normal compressive force in the gap.

The following gap function determines whether the status is open or closed,

$$\mathbf{g} = \mathbf{u}_2 - \mathbf{u}_1 + \mathbf{g}_0 \quad (2.527)$$

where, $(\mathbf{u}_2 - \mathbf{u}_1) = (\mathbf{u}_2 - \mathbf{u}_1) \cdot \mathbf{n}$ is the relative displacement in the gap normal direction (i.e., normal to the interface). The gap function g is referred to as the current gap width in the element output. The gap condition is represented by the following inequality constraint equation $\mathbf{g} \geq 0$

The normal stiffness (K_n) represents a penalty number and the constraint is satisfied approximately, such that

$\mathbf{g} > 0$ represents an open status
 $\mathbf{g} \leq 0$ represents a closed status

In other words, penetration is present in the element formulation, and the normal force in the gap is proportional to the amount of penetration, that is,

$$\mathbf{g} = \frac{f_n}{K_n} \text{ (for a closed gap)} \quad (2.528)$$

The above equation, (as depicted in Figure 2.22(a)) represents weak form of the contact condition. However, if $K_n \rightarrow \infty$, the exact contact condition ($\mathbf{g} = 0$) is recovered.

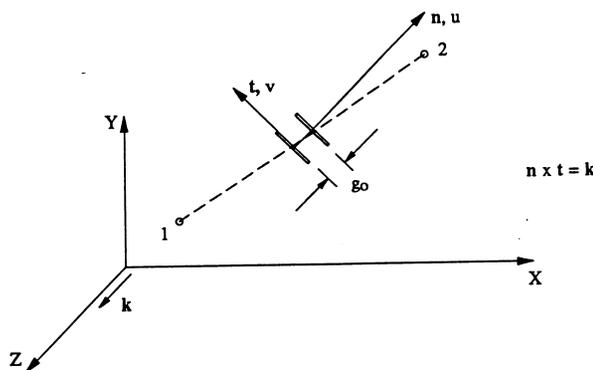


Figure 2.21: 2-D Gap/Friction element

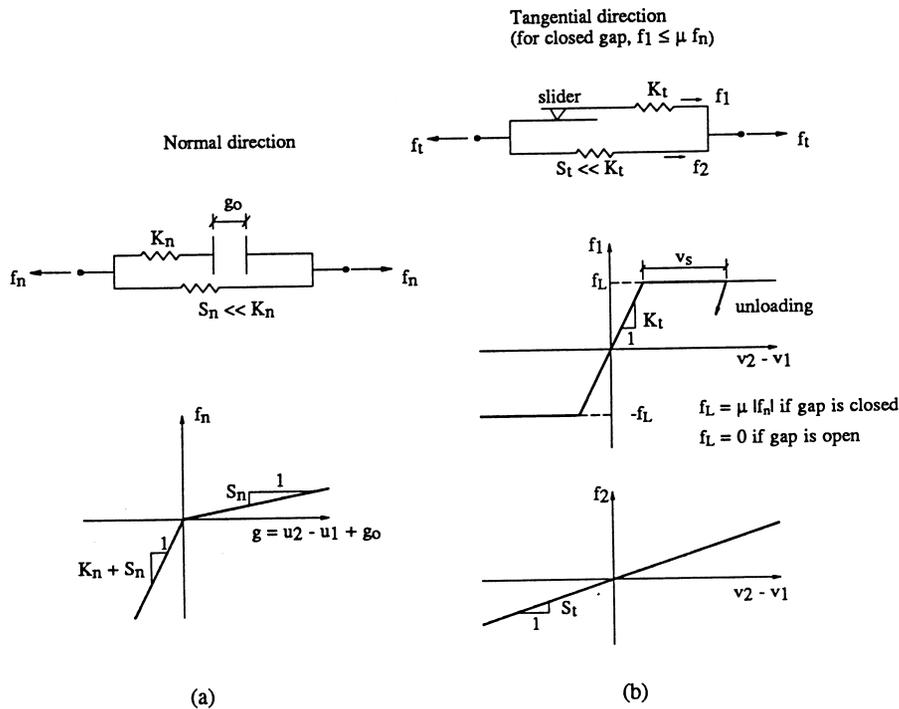


Figure 2.22: Force-deflection relationships: (a) Normal direction; (b) Tangential direction

When the element is in contact and the shear force is below the friction limit ($= \mu |f_n|$), the element is said to be in a sticking mode. In this case, the increment in the relative tangential displacement $\Delta(v_2 - v_1)$ represents recoverable (elastic) deformation. If the predicted elastic shear force exceeds the friction limit, the shear force is set to the frictional limit and the element is said to be in a sliding mode. In this case, the increment $\Delta(v_2 - v_1)$ includes irrecoverable part (sliding displacement, similar to plastic deformation as shown by v_s in

Figure 2.22(b)).

The element stiffness and internal force vector are given below in the element coordinate system (n, t) . Standard coordinate transformation are used to obtain the element matrices in

global coordinates. The element displacement vector in the element local coordinate system is given by:

$$\mathbf{u}^T = [\mathbf{u}_1^T, \mathbf{u}_2^T] = [u_1, v_1, u_2, v_2] \quad (2.529)$$

The stiffness matrix is given by

$$\mathbf{K} = \begin{bmatrix} A_n & 0 & -A_n & 0 \\ 0 & A_t & 0 & -A_t \\ -A_n & 0 & A_n & 0 \\ 0 & -A_t & 0 & A_t \end{bmatrix} \quad (2.530)$$

The element internal force vector is given by

$$\mathbf{f}^T = [-f_n \ -f_t \ f_n \ f_t] \quad (2.531)$$

where the coefficients A_n , A_t are given in [Table 2.5](#) for each gap status type. The forces f_n and f_t are calculated using the rheological models and corresponding load-deflection relationships shown in [Figure 2.22](#).

Table 2.5: Stiffness coefficients for different Gap Status

Coefficient Label	Element Status		
	Open (code = 0)	Closed	
		Sticking (code = 1)	Sliding (code = ± 2)
A_n	S_n	$K_n + S_n$	$K_n + S_n$
A_t	S_t	$K_t + S_t$	S_t

NISA Capabilities

3.1 Introduction

This chapter describes the overall NISA capabilities and provides general background information common to all analysis types. To facilitate the preparation of the input data and the interpretation of the results, cross-reference to the pertinent data group identification names or to the pertinent section numbers are given whenever applicable.

The major capabilities in NISA include linear and nonlinear static, dynamic, buckling and heat transfer analyses. A comprehensive library of elements is available. This includes 2-D and 3-D point masses, springs, spars and beams, 2-D plane and axisymmetric solid elements, 3-D solid elements, axisymmetric and 3-D shell elements, 3-D layered composite solid and sandwich shell elements, gap elements and axisymmetric shell and solid elements with non-axisymmetric loading. Complete description of the element library is given in [Chapter 4](#). The available material models in NISA include linear isotropic and orthotropic elastic materials, elastoplastic material with various yield and flow rule criteria, and hyperelastic (or rubber-like) material model with various forms of constitutive relationships. Other capabilities include extensive data checking with explicit diagnostic messages, efficient wavefront solution technique and element resequencing algorithm, multiple load cases with different boundary conditions, and numerous output options.

NISA is completely interfaced with the DISPLAY program for model and input data generation and postprocessing of analysis results. Refer to the DISPLAY II user's manual for details.

The NISA input data deck consists of three data blocks. These are:

1. Executive command data block:

This consists of alphanumeric commands specifying general control parameters for the analysis, e.g., specifying the type of analysis to be performed.

2. Model data block:

This block generally represents the bulk of the input data. It describes the model characteristics in terms of nodes, elements, material and geometric properties, etc.

3. Analysis data block:

This block describes data pertinent to various analysis types, e.g., loading, boundary conditions, output control, etc.

The model and analysis data blocks typically consists of distinct data groups, each with a unique alphanumeric group identification card indicating the function of the data group. For example, the *CFORCE data group specifies all concentrated forces and moments.

Two formats types, Regular and Large, can be selected to define the element and node IDs. In the regular format the IDs can have a maximum of 6 digits, where as for the large format the maximum is 8 digits. As default NISA reads in the regular format. Large format can be selected by specifying the executive card.

3.2 Coordinate Systems, Degrees of Freedom and Sign Convention

3.2.1 Types of Coordinate Systems

There are several kinds of coordinate systems used in NISA:

1. *Global Coordinate System*

The global coordinate system is an essential part of the finite element model. It is used to define the model geometry by specifying the locations of node points in space. The location of the origin of this global system is arbitrary. Three types of global coordinate system are defined in NISA as shown in [Figure 3.1](#).

- (a) Cartesian system (X, Y, Z)
- (b) Cylindrical system (R, θ , Z)
- (c) Spherical system (R, θ , ϕ)

Any of these systems may be used to define the coordinates of a typical node, and they are referenced by the geometry coordinate system ID of 0, 1, 2, respectively (see *NODES data group). Ultimately, nodal coordinates are internally expressed in the global Cartesian coordinate system. Also, the element contributions to the overall system equations are assembled in a common coordinate system; usually the global Cartesian system. The global Cartesian system is the default displacement coordinate system used to express the components of motion and forces. As shown in [Figure 3.1\(d\)](#) positive translations (UX, UY, UZ) and forces (FX, FY, FZ) are along the +X, +Y and +Z directions, respectively. Positive (right-hand-rule) rotations (ROTX, ROTY, ROTZ) and moments (MX, MY, MZ) are about the +X, +Y and +Z directions, respectively. A *local* displacement coordinate system may be defined at any node directly, or it may be *internally* generated for nodes defined in the global cylindrical or spherical systems, see item 3 below.

2. *Specified Local Coordinate System*

In addition to the three basic coordinate systems defined above, a local coordinate system (X_s , Y_s , Z_s) may be used to express the *geometry* (nodal coordinates) as shown in

Figure 3.1(d). Several specified local coordinate systems may be used. The specified local system may be oriented anywhere in space, and is defined with respect to the global Cartesian coordinate system by specifying the position vector of the origin $0'$ of the system, the direction of the $+X_s$ axis and a point (or an orientation vector) in the X_s, Y_s plane. Alternatively, the local system (X_s, Y_s, Z_s) may be defined using three successive rotation angles, see *LCSYSTEM data group for details. The specified local coordinate system *must* have a coordinate system ID number ≥ 3 , and it may be Cartesian, cylindrical, or spherical. Once a local system is defined in the *LCSYSTEM, it may be used as the geometry coordinate system to define the coordinates of any node, see *NODES for details. Note that the geometry coordinate system is *independent* from the displacement coordinate system. That is, a node may have its coordinates defined with respect to coordinate system A, but its displacement coordinate system is associated with coordinate system B.

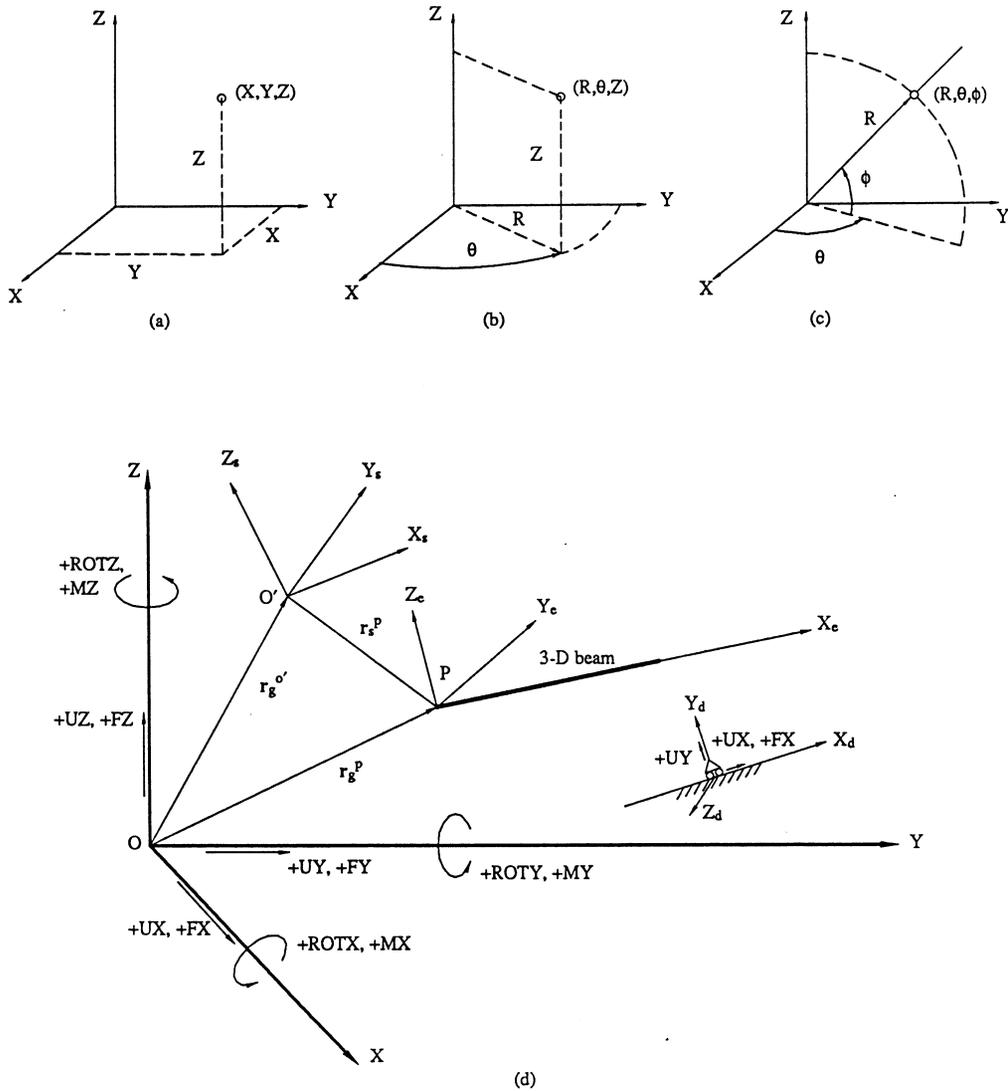


Figure 3.1: Coordinate systems and sign convention for displacements and forces; (a) Cartesian system; (b) Cylindrical system; (c) Spherical system; (d) Global and local systems; (X, Y, Z) global; (X_s, Y_s, Z_s) specified local; (X_e, Y_e, Z_e) element local; (X_d, Y_d, Z_d) local displacement system

3. *Local Displacement Coordinate System*

In some instances, it may be more convenient to refer the components of motion and forces at any node to a local displacement coordinate system (X_d, Y_d, Z_d) , which is different from the default displacement coordinate system (the global Cartesian) as shown in [Figure 3.1\(d\)](#). Possible reasons may be:

- (a) To represent support conditions, or to impose displacement constraints in directions which are not aligned with the global Cartesian axes. For example, consider the roller support condition shown in [Figure 3.1\(d\)](#). This condition may be expressed using one of two options: the local coordinate system (X_d, Y_d, Z_d) may be defined at the support and the displacement component UY (now UY is along Y_d) be set to zero. Alternatively, a multi-point constraint equation of the form $\mathbf{j}_d \cdot \mathbf{u} = 0$ may be used, where \mathbf{j}_d is a unit vector along the Y_d axis and \mathbf{u} is the displacement vector listing the global components of translation at the point. Obviously, the first option is more convenient (and is easier for the program to handle).
- (b) To specify forces or moments whose components are readily available in a preferred coordinate system which is not aligned with the global Cartesian system.
- (c) To obtain the response at any node in any desired directions.
- (d) To select cylindrical or spherical coordinate system as a local displacement coordinate system for any node whose coordinates have been specified in any of these two systems. The procedure is automated such that a local displacement coordinate system, which depends on the location of the node, is internally defined once the displacement coordinate system ID is set equal to the geometry coordinate system ID (= 1 or 2 in this case) in the *NODES data group.

A local displacement coordinate system may be defined using the same options described in item 2 above. It should be noted that once a local coordinate system is specified at any node, then any reference to the components of motion and forces at this node will always be in this local system. This includes specifying forces or moments, boundary conditions, coupled displacements and multi-point constraint equations (defined directly in *MPCEQN and/or generated internally for rigid links). In addition, the force output at that node will be referred to the local system. The force output includes element internal forces, nodal force balance and reactions. The displacement output may be obtained in the local system, in the global Cartesian system, or in both.

4. *Element Local Coordinate System*

Element local coordinate systems are used to define the element and/or the orthotropic material orientation. The element orientation is specified directly by the user for some elements, e.g., the 3-D beam ([Figure 3.1\(d\)](#)) and the 3-D general spring elements, or it is determined internally for the majority of elements, e.g., 2-D spars, 2-D beams, and shells. For springs, spars, and beams, the element real constants (spring constants, areas, cross-sectional properties, etc.), as well as the vertex forces output and the end release codes for 3-D beams are defined in the element local coordinate system. For shell elements, the output of the local components of stress refers to the shell local coordinate system. The direction cosines of the local axes are provided along with the local stress output. An element local coordinate system is associated with an individual element, and in many cases (e.g., shells with isotropic material) is determined internally. Details of the element local coordinate system may be found in the description of each individual element in [Chapter 4](#).

3.2.2 Degrees of Freedom

In the displacement-based finite element method, the components of the displacement vector at the nodes are the primary field variables (degrees of freedom) determined by the analysis. For heat transfer analysis, the temperatures at the nodes are the primary unknowns. Intrinsic variables such as stresses and heat flow are secondary unknowns which are derived from the primary field variables.

The displacement vector at a node point has a maximum of six components (three translations and three rotations). For input and output purposes, a displacement degree of freedom at a node point is identified by the node number and a label. As shown in [Figure 3.1\(d\)](#), the labels are (UX, UY, UZ, ROTX, ROTY, ROTZ).

In NISA, the number of degrees of freedom at a node point is determined from the union of the degrees of freedom (per node) of the elements connected to it. For example, if a node is connected only to a 3-D solid element, the degrees of freedom at this node are UX, UY, UZ. On the other hand, if this node is also connected to a 3-D beam element with six degrees of freedom per node, it will have six degrees of freedom. Nodes which are defined in the nodal coordinates data (*NODES data group) but do not appear in the element connectivity data (*ELEMENTS data group) are automatically purged. This rule is not enforced for the master nodes of rigid links, see [Section 3.10.2](#). The number of degrees of freedom in the model is

provided in the NISA printout and is itemized as total, unconstrained (active), constrained and dependent.

3.2.3 Sign Convention

Displacements and Forces

As mentioned earlier, the global Cartesian system is the *default* displacement coordinate system used to express the components of motion and forces except for the nodes where a local displacement coordinate system is specified. As shown in [Figure 3.1\(d\)](#), the translations (UX, UY, UZ) and the forces (FX, FY, FZ) are positive along the positive X, Y and Z axes, whereas the rotations (ROTX, ROTY, ROTZ) and the moments (MX, MY, MZ) are about the X, Y and Z axes and are considered positive in the right-hand sense.

Stresses

NISA provides the components of stress tensor in the global and/or local XYZ Cartesian coordinate system. Local components may be provided for shell elements or for orthotropic materials. In order to establish the directions of stress components, consider an infinitesimal cube at the point of interest. The positive stresses act on the positive faces of the cube in the positive direction as shown in [Figure 3.2](#). The components of the stress tensor are the direct stresses S_{xx} , S_{yy} , S_{zz} , and the shear stresses S_{xy} , S_{yz} , S_{xz} ($= S_{yx}$, S_{zy} , S_{zx} , respectively). The first subscript indicates the face on which the stress is acting and the second subscript gives the direction. For example, S_{xx} is the direct stress acting on the +X face and in the positive X direction.

Stress Intensities

The following stress intensities are computed from the stress components. They provide useful information for interpreting the behavior of the material according to the various theories of failure:

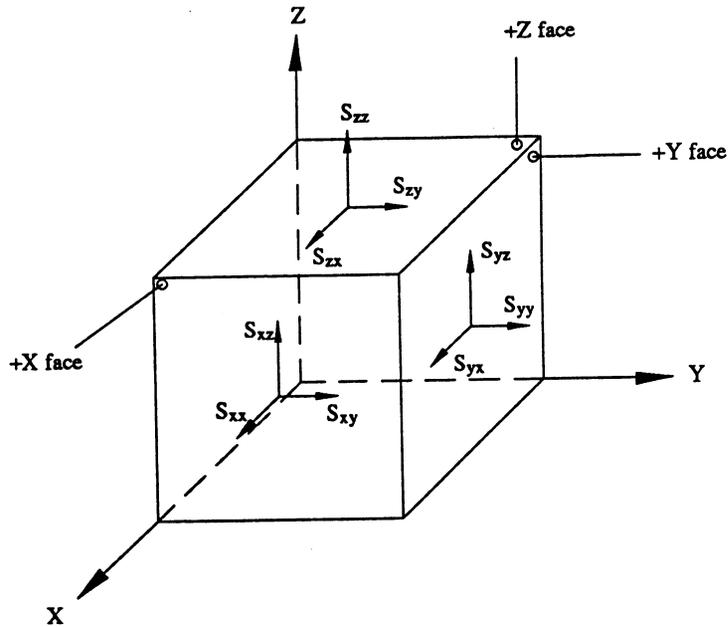


Figure 3.2: Sign convention for stresses

Maximum Shear stress,

$$S_{\max} = \text{maximum of } \left[\frac{1}{2}|S_1 - S_2|, \frac{1}{2}|S_2 - S_3|, \frac{1}{2}|S_3 - S_1| \right]$$

von Mises equivalent stress,

$$S_{\text{eq}} = \frac{1}{\sqrt{2}} [(S_1 - S_2)^2 + (S_2 - S_3)^2 + (S_3 - S_1)^2]^{1/2}$$

Octahedral shear stress,

$$S_{\text{oct}} = \frac{\sqrt{2}}{3} S_{\text{eq}}$$

in which S_1 , S_2 and S_3 are the principal stresses.

3.3 Linear Static Analysis (STATIC)

Linear static analysis is concerned with the linear behavior of elastic continua under prescribed boundary conditions and statically applied loads. This includes the calculation of displacements, strains, stresses, reactions, and energy in the continua. In general, the applied loads include prescribed forces and pressures. Other loading types may also be included, namely body forces (e.g., gravity and centrifugal loads) and thermal loading (temperature specification). The kinematic conditions may include specified displacements, coupled displacements, multi-point constraints and rigid link specifications, (see [Section 3.10](#)). The basic equation for linear static analysis may be written in the form,

$$\mathbf{K}\mathbf{u} = \mathbf{p} \tag{3.1}$$

where, \mathbf{K} is the linear stiffness matrix of the structure, \mathbf{u} is the nodal displacement vector and \mathbf{p} is the load vector.

An extensive element library is available in NISA which enables the user to model various structural problems. This includes spar, spring and beam elements, two-dimensional plane stress or strain elements, axisymmetric solid elements, layered composite shell and solid elements, shell elements, axisymmetric solid and shell elements with non-axisymmetric loading and three-dimensional solid elements (see [Chapter 4](#) for details).

[Figure 3.3](#) shows a flow chart for a typical NISA static analysis run. The procedure along with pertinent capabilities may be summarized in the following steps:

- ❑ A NISA input deck is prepared. DISPLAY-PRE data generation program may be used for interactive model generation.
- ❑ Data processing is performed by the program including various levels of data checking (see [Section 3.12](#) for details) and the geometry and the basic data are written to file 26
- ❑ Wavefront optimization is performed (unless suppressed) to minimize the wave front and resequence the elements. This process is transparent to the user and all input and output specifications are in the user's element identification numbers (see [Section 3.13](#) for details).
- ❑ Element stiffness matrices are calculated based on the geometry, element types and the material models given. Both isotropic and orthotropic material models are available in NISA with temperature dependent material properties. For orthotropic mate-

rials, the material principal directions may be defined in global, element or nodal format (see [Section 3.9](#) for details). Layered composite material models are also available in NISA and may be used with 3-D laminated composite solid and shell elements. The element stiffness matrices are written to file 24.

- ❑ A loop on load cases starts with the generation of element load vectors which are written to file 28. Element loads may include pressure, thermal and body force loadings (see [Section 3.11](#) for details).
- ❑ Displacement solutions are obtained using the frontal solution techniques (see [Section 3.13](#) for details) and the decomposed stiffness matrix is written to files 30, 61 to 69 and is used for subsequent load cases if there is no change in the specified displacement boundary conditions.
- ❑ Stress recovery and computation of internal forces, reactions, etc. are finally initiated and the calculated results are written to the output file and to file 27 (postprocessing data file).
- ❑ Graphical representation of the results may be obtained using the DISPLAY-POST interactive program.

Basic Output

The basic output from linear static analysis includes:

1. Applied load vector at selected nodes.
2. Displacement components at selected nodes.
3. Reactions at selected constrained nodes.
4. Element internal forces and strain energy for selected elements.
5. Nodal force balance (equilibrium check).

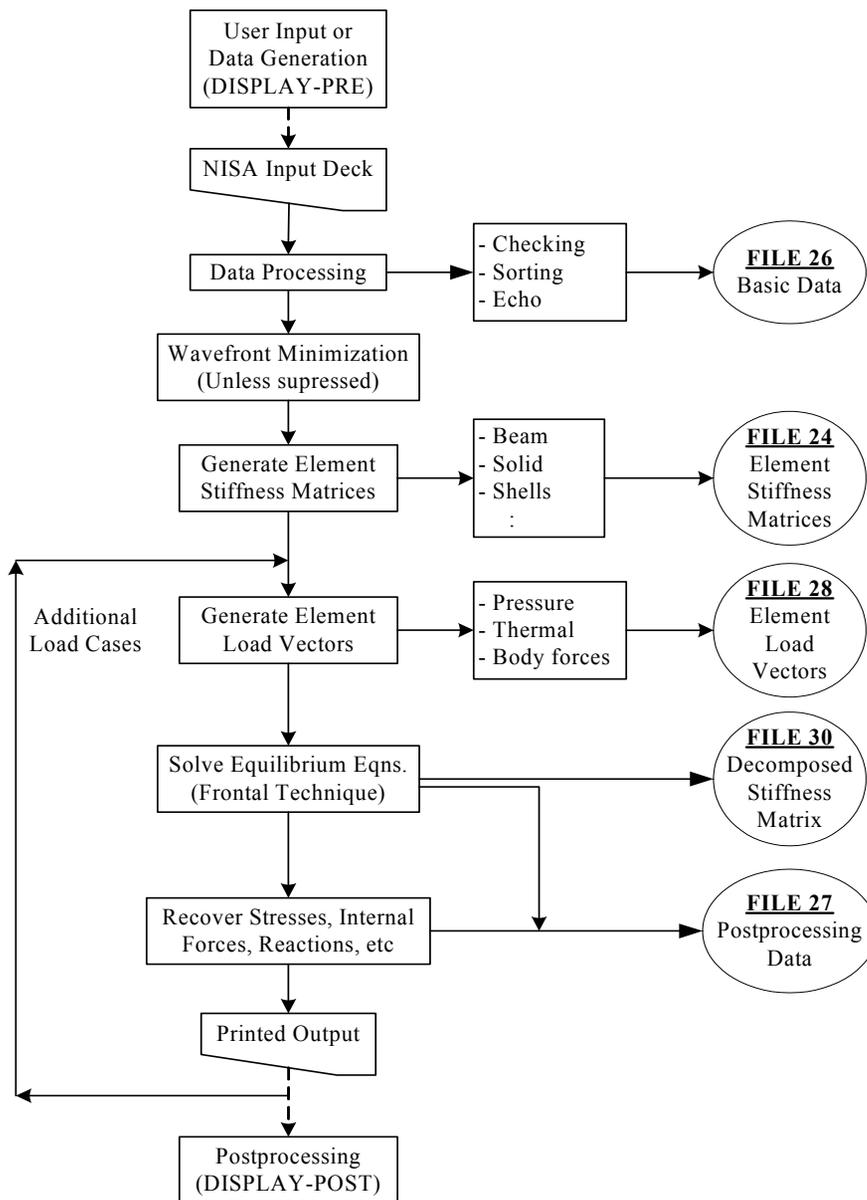


Figure 3.3: NISA Static analysis flow chart

6. Element stresses (stress resultants for line elements) and stress intensities for selected elements.
7. Averaged nodal stresses and stress intensities for selected nodes.
8. Stress filtering and sorting.

Graphical representation of the results, e.g., deformed shape and stress contour plots may be obtained using the DISPLAY program. Table form description of the above output items along with the data groups requesting them, as well as the postprocessing procedures and features are given in [Section 3.14](#).

3.3.1 Moving Loads in Static Analysis

In civil engineering application, it may require to analyze the effects of loads moving along a structure such as a bridge or railroad. To help users to achieve this task easily, NISA provides the moving load path (*MPATH) data group and the moving load (*MLOAD) data group to generate multiple load cases to simulate moving load analysis. *MPATH uses nodes in the model to define a continuous path along the edge of the element. The order of the nodes given in *MPATH data group will define the direction of moving load. The moving load can be concentrated load or distributed load. Every moving load set will have a leading position which indicates the current reference location. The leading position is always at the location of a node of a path. All moving loads in the moving load set should be behind the leading position with a distance, which can be zero. The leading position can move one node at a time or several nodes at a time along the path depending on the users' choice which can be specified in *MLOAD data. Based on the data given in *MPATH and *MLOAD, NISA will generate load cases for the moving load analysis.

3.4 Nonlinear Static Analysis (NLSTATIC)

3.4.1 Types of Nonlinearities and Formulation

Nonlinear static analysis (NLSTATIC) deals with nonlinear behavior of structures under static loading. The nonlinearities considered in the program may be material, geometric, combined (material and geometric) nonlinearity, or boundary nonlinearity (gap or contact). In material nonlinearity, the material constitutive or stress-strain relations may be dependent on stresses, strains and/or displacements. Applications for material nonlinearity are evident, for example, in plasticity, creep, and viscoplasticity problems. Three main material nonlinear behaviors are available in this program; these are: the elastoplastic material behavior, creep, and the hyperelastic or rubber-like material behavior. In the elastoplastic material behavior various types of yield criteria and hardening rules are available. In creep model, both general or ORNL (Oak Ridge National Laboratory) material laws can be used. In the hyperelastic or rubber-like material models, various types of strain energy functions with finite compressibility or near incompressibility behavior are available. Detailed description of these models and their theoretical basis is given in [Section 2.7](#) and [Section 3.9](#).

The second type of nonlinear behavior available in the program is the geometric nonlinearity. In this case the classical theory of infinitesimal strains does not hold, and the strains are obtained from the displacements via a nonlinear differential operator. This type of nonlinearity may involve large displacements, large rotations and finite strains. Applications for geometric nonlinearity are evident, for example, in large deflection and buckling of shells, where large displacements and rotations are important. Common applications for large or finite strains are, for example, metal forming problems and large deformation of rubber-like materials.

The third type of nonlinear behavior available in the program is boundary nonlinearity. In this case, the material and strains remain linear. The only nonlinear behavior comes from changing of boundary.

In geometric nonlinearity, or combined geometric and material nonlinearities, a distinction has to be drawn between the original (undeformed) and the deformed configurations of the structure, and the equilibrium or energy balance equations must be written for the deformed configuration. Consequently, various formulations, and stress and strain measures arise depending on the chosen reference configuration to describe the deformation of the body. In

the program two referential formulations are available; these are: the total and the updated Lagrangian formulation. The total Lagrangian formulation uses a fixed configuration (the undeformed configuration) as a reference to describe the motion of the body. In the updated Lagrangian formulation, the reference configuration is always updated. The second Piola-Kirchhoff stress tensor and the Green-Lagrange strain tensor (conjugate pair) may be printed in either formulation. The user may also choose to print the more physical Cauchy stress tensor, which represents true stresses, in either formulation. For details of the formulations, see [Chapter 2](#) and [3.1-3.4].

Both formulations are equivalent mathematically and they include all the nonlinear kinematic effects. If appropriate constitutive relations are used, and the same approximations are introduced, identical results (except for round-off differences) should be obtained. The advantage of one formulation over the other may be in numerical efficiency, ease of handling particular nonlinear boundary conditions, ease of specifying material constitutive relations, etc. Whenever applicable, the user will be advised of the preference of a particular formulation, e.g., for specific elements and/or analysis options.

In contact problem, penalty method and augmented Lagrangian multiplier method are available for the analysis. In penalty method, the contact force is the product of amount of penetration and the penalty number. The value of penalty number controls the penetration or depth of the penetration between two contact surfaces. A larger penalty number will result in less penetration at equilibrium and also more accurate results. However, this method is sensitive to the value of penalty number. Improper penalty number may result in incorrect results due to too much penetration from a small penalty number or diverge results due to too much change in contact forces from a large penalty number. In augmented Lagrangian method, the contact forces are updated incrementally until the system is balanced. A smaller value of penalty number is used than the penalty method and the incremental contact force is computed from the product of penetration and the penalty number. Therefore, it is numerically more stable than penalty method and it needs more iterations or computation time to obtain the solution. If a problem has relatively uniform contact forces over the contact area, penalty method can provide a faster solution by using a proper penalty. On the other hand, if a problem has a large variation of contact forces, augmented Lagrangian method may be a better choice otherwise the solution may not converge. In the Lagrangian Multiplier method the impenetrability constraints are satisfied in an exact manner. The main advantage in the Lagrangian multiplier method is that the user does not have to specify any penalty parameter which is a problem defined parameter and requires user's discretion. Moreover, the user can specify small regions around the slave nodes with a pinball radius

NISA Capabilities

Nonlinear Static Analysis (NLSTATIC)

parameter so that the search process is limited to only this region and the whole analysis process becomes faster.

User can also view the convergence criteria as graphs by running the `ndp.exe` file from the installation folder during the complete run of the analysis. This helps in visualizing the convergence pattern in order to proceed with or stop at any point of time during the analysis.

3.4.2 Load Stepping

The entire load history for a typical nonlinear analysis may be divided into *events*. Generally, an event should be defined when new load types are introduced, when an existing load changes its sign, or when solution strategy changes. An event may have one or more load steps (or increments). Step size (load or time increments) may be determined automatically or via user definition. In each event, the user may define parameters for step (or time increment) size, number of steps, equilibrium checks, tolerances for convergence, iterative procedure and other control parameters.

Various types of loading may be applied in each event. This may include concentrated nodal forces (follower and nonfollower), pressure loads (follower and nonfollower), body forces, nodal temperatures, and non-zero specified displacement boundary condition. Detailed description of the loading types is given in [Section 3.11](#). Each type of load is associated with a time amplitude curve. The value of the applied load at a given time is determined from the load value specified in the corresponding data group, and the referenced time amplitude curve. The time amplitude curve allows a general description of the load history, see *TIMEAMP for details.

3.4.3 Solution Schemes

Conventional and Modified Newton-Raphson Methods

In solving the nonlinear equilibrium equations, the program employs an incremental-iterative scheme, for which at a time t , the equilibrium equations may be written in the form,

$$\mathbf{K}_t \Delta \mathbf{u}^{(i)} = {}^{t+\Delta t} \mathbf{p} - {}^{t+\Delta t} \mathbf{f}^{(i-1)} \quad (3.2)$$

and,

$${}^{t+\Delta t} \mathbf{u}^{(i)} = {}^{t+\Delta t} \mathbf{u}^{(i-1)} + \Delta \mathbf{u}^{(i)} \quad (3.3)$$

where,

\mathbf{K}_t	is the tangent stiffness matrix at time t (linear and nonlinear),
$\Delta \mathbf{u}^{(i)}$	is the iterative increment of nodal displacements in iteration (i) ,
${}_{t+\Delta t} \mathbf{u}^{(i)}$, ${}_{t+\Delta t} \mathbf{u}^{(i-1)}$	are the vectors of nodal displacements at time $t + \Delta t$ corresponding to iterations (i) and $(i-1)$, respectively.
${}_{t+\Delta t} \mathbf{p}$	is the vector of externally applied mechanical nodal forces at time $t + \Delta t$ and pseudo forces due to thermal and creep effects.
${}_{t+\Delta t} \mathbf{f}^{(i-1)}$	is the vector of internal nodal forces, equivalent to element stresses, at time $t + \Delta t$ and iteration $(i-1)$

The incremental solution is performed in a step-by-step manner until the full specified loads are applied. In each increment, the above iterative scheme is performed until convergence is achieved or maximum iterations are reached. During each increment, the tangent stiffness matrix \mathbf{K}_t may be updated for each iteration (Newton-Raphson method), or kept constant in all iterations of the pertinent increment (modified Newton-Raphson method). In the case of the modified Newton-Raphson method, the user has the option of updating the tangent stiffness matrix at the first few iterations, or every specified number of iterations. Although the use of the modified Newton-Raphson may be more economical in some specific material nonlinearity applications, the utilization of the method in general for material and geometrical nonlinearities is not always successful. Also, in material nonlinearity problems, the application of the method has been rather limited to problems in which no unloading occurs. Both the conventional and the modified Newton-Raphson methods are compared graphically in [Figure 3.4](#).

Performance of the Newton-Raphson iterative method can be improved significantly by using line search technique. Line search is done to find a scalar multiplier for the iterative displacement. The optimum value for this scalar is found such that it minimizes the residual forces. The search for the optimum value involves recalculation of stresses and residual forces without reforming stiffness or solving the equilibrium equations. Hence, line search is computationally inexpensive and generally it improves the performance of the Newton-Raphson method.

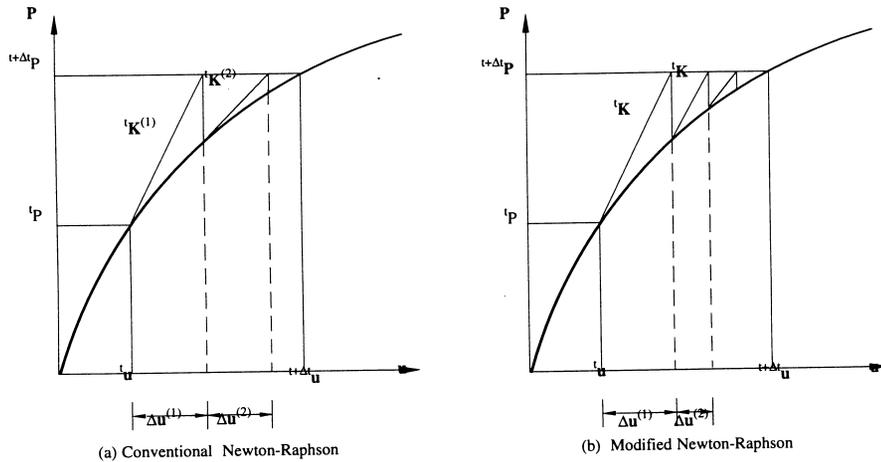


Figure 3.4: Comparison of Newton-Raphson schemes

Pure Incremental Procedure

A pure incremental analysis, i.e., with no iterations, may be invoked by specifying EQUILIBRIUM = ON with very high tolerances in the event data group, the *EVENT. A special formulation ([3.4]) is also available in NISA, for pure incremental analysis, which may be activated by specifying EQUILIBRIUM = OFF in the *EVENT data group. Usually, the special formulation gives better results when compared with the conventional one. This special feature is important in practical nonlinear analyses where the number of increments and iterations are unknowns. The user may find it advantageous to use the special pure incremental feature as a first trial in order to get acquainted with the problem and determine appropriate parameters such as the number of increments, tolerances, etc.

Convergence Criteria

Three convergence criteria are available in the program, these are: displacement, force, and energy criterion. In the displacement criterion, a load step will be assumed converged when the ratio of the maximum absolute iterative displacement to the maximum absolute displacement at the first iteration is less than or equal to a specified tolerance value. The force criterion assumes a load step converged when the ratio of the Euclidean norm of the residual force vector to the Euclidean norm of the incremental force vector is less than the specified tolerance. Finally, in the energy criterion, a load step will be assumed converged when the ratio of the iterative energy to the energy at the first iteration is less than the specified tolerance.

Tolerances for each criteria may be specified independently on the *EVENT data group. If all tolerances are specified, a step will be assumed converged when any *one* criterion is satisfied. If the maximum number of iteration limit is reached, before the satisfaction of any criteria, the step is not converged and the program stops. It should be noted that loose tolerance values may lead to inaccurate results, whereas tight tolerance values may be uneconomical from the computational point of view. In general, the use of only one criterion (specially the force one) may not be adequate for general applications. For example in some applications of elastoplastic problems with zero or very small strain hardening modulus, the force criterion may be easily satisfied while the displacement one is in significant error.

Faster Iteration Using Line Search or Step-length Parameter

Line search is a powerful and efficient methodology for solving nonlinear structural problems. In mathematical terms they are continuation methods capable of solving problems with strong nonlinearities and passing extremes or limit points. Line searches (with very few extra residual calculations) can significantly improve the performance of the modified Newton-Raphson method. A line search algorithm based on secant-type interpolations (or extrapolations) as proposed by Crisfield [3.21,3.22] and implementation described by Patel [3.23] is adopted in NISA.

During an iteration step the total displacement $\mathbf{u}^{(i)}$ is updated using Equation (3.3). In the line search technique, Equation (3.3) is rewritten as,

$$\mathbf{u}^{(i)} = \mathbf{u}^{(i-1)} + \eta^{(i,k)} \Delta \mathbf{u}^{(i)} \quad (3.4)$$

The total displacement is updated using the previous total displacement, current iterative displacement and a scalar step length parameter $\eta^{(i,k)}$. The superscripts (i,k) stand for the line steps ($k = 0,1,2,3,\dots$) in the current iteration (i). The iterative vector $\Delta \mathbf{u}^{(i)}$ may be found using any technique such as the Newton-Raphson or modified Newton-Raphson. If a line search is not to be used (k is limited to 1) then $\eta^{(i,k)}$ is taken as unity. In some cases, it will be advantageous to apply a line search to find the optimum value of the scalar $\eta^{(i,k)}$.

The optimum value of $\eta^{(i,k)}$ is the value that makes the inner product of the out-of-balance force vector $\mathbf{s}^{(i)}$ and $\eta^{(i,k)} \Delta \mathbf{u}^{(i)}$ zero. The inner product is denoted by $\epsilon^{(k)}$

$$\boldsymbol{\varepsilon}^{(k)} = \mathbf{s}^{(i,k)} \cdot \eta^{(i,k)} \Delta \mathbf{u}^{(i)} = 0 \quad (3.5)$$

For a given $\Delta \mathbf{u}^{(i)}$, satisfying the above equation involves a one-dimensional search for $\eta^{(i,k)}$. Each trial value of $\eta^{(i,k)}$ involves a calculation of the out-of-balance force vector $\mathbf{s}^{(i,k)}$. This is done by finding the incremental strain from the updated displacement $\mathbf{t} \mathbf{u}^{(i)}$. Next the total stress is calculated using elastic or thermoelastoplastic equations. Finally $\mathbf{s}^{(i,k)}$, using Equation (3.2), (where $\mathbf{s} = \mathbf{p} - \mathbf{f}$), is calculated.

In the search for the optimum $\eta^{(i,k)}$, first $\boldsymbol{\varepsilon}^{(0)}$ is calculated using $\Delta \mathbf{u}^{(i)}$, which in turn was calculated using the residual load at the end of iteration (i - 1), denoted as $\mathbf{s}^{(i-1, k=0)}$.

$$\boldsymbol{\varepsilon}^{(0)} = \Delta \mathbf{u}^{(i)} \cdot \mathbf{s}^{(i-1,0)} \quad (3.6)$$

For the first line search step ($k = 1$) (or in case of no line search) $\eta^{(i,1)} = 1.0$, $\mathbf{s}^{(i,1)}$ is calculated as explained earlier. Now $\boldsymbol{\varepsilon}^{(1)}$ or in general $\boldsymbol{\varepsilon}^{(k)}$ can be found as:

$$\boldsymbol{\varepsilon}^{(k)} = \Delta \mathbf{u}^{(i)} \cdot \mathbf{s}^{(i,k)} \quad (3.7)$$

If the condition $\boldsymbol{\varepsilon}^{(k)} = 0$ is not satisfied, then $\eta^{(i,k-1)}$ was not optimum. A new trial value for $\eta^{(i,k+1)}$ is found by linear interpolation using $\boldsymbol{\varepsilon}^{(0)}$, $\boldsymbol{\varepsilon}^{(k)}$ and $\eta^{(i,k)}$. Next $\eta^{(i,k+1)}$ is used to find the next trial $\mathbf{t} \mathbf{u}^{(i,k+1)}$ and so on. $\boldsymbol{\varepsilon}^{(k)} = 0$ cannot be satisfied exactly. Instead, it is practical to satisfy.

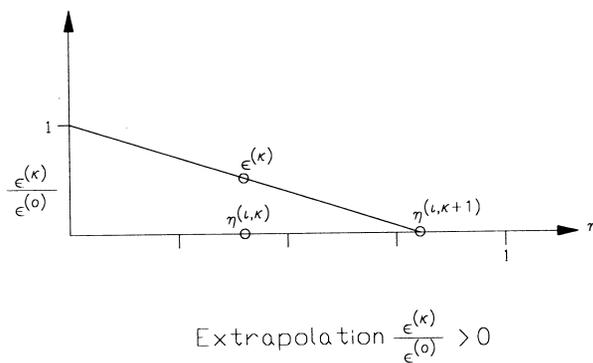
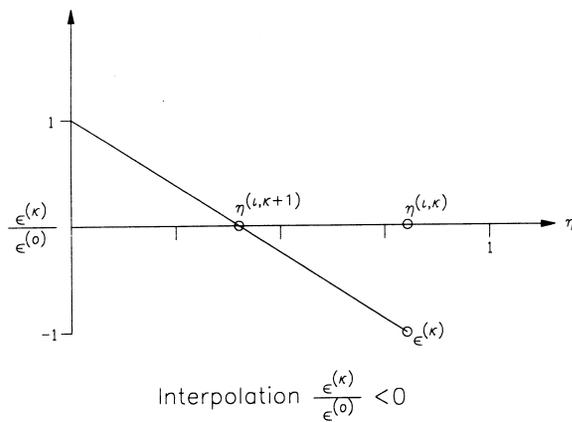


Figure 3.5: Interpolation and extrapolation techniques in line search

$$\frac{\epsilon^{(k)}}{\epsilon^{(0)}} \leq \Psi' \tag{3.8}$$

where Ψ' is a line search tolerance.

The simplest interpolation procedure to find $\eta^{(i,k+1)}$ is to interpolate (or extrapolate) between the estimates of $\varepsilon^{(k)}$ and $\varepsilon^{(0)}$

$$\frac{\eta^{(i,k+1)}}{\eta^{(i,k)}} = \frac{\varepsilon^{(0)}}{\varepsilon^{(0)} - \varepsilon^{(k)}} \quad (3.9)$$

Better interpolation may be achieved using bracketing procedures. In particular, one may interpolate (extrapolate) between the first negative value of $\varepsilon^{(k)}/\varepsilon^{(0)}$ and the nearest (with smaller $\eta^{(i,k)}$) that is positive. If $\eta^{(i,k+1)}$ involves extrapolation, it can be dangerous, so a maximum ratio of ten is stipulated. These interpolation and extrapolation techniques are shown in [Figure 3.5](#) If $\varepsilon^{(0)}$ is negative, the displacement, $\Delta \mathbf{u}^{(i)}$, is going downhill in an energy sense. To find stable equilibrium, $\eta^{(i,1)} = -1.0$ instead of $\eta^{(i,1)} = 1.0$ is taken.

Arc Length Method

To follow the structural response beyond the critical point presents a challenging task to the analyst. The major difficulty is to overcome the singularity of the incremental tangent stiffness matrix when the structure reaches its stability limit. In addition, snap-through and snap-back buckling phenomena pose some of the most difficult problems in nonlinear structural analysis. A method called “arc length automatic stepping”, can be applied to overcome the problems of stiffness singularity and post-buckling.

The general goal of arc length procedure is the control of iteration in the numerical solution of complex nonlinear problems. The main idea of this method is based on the concept of constraining the length of the incremental displacement. The incremental displacement length for each successive iteration is constrained by the length of the previous iteration. Correspondingly, the load is adjusted in order to satisfy the global equilibrium requirement of the system.

Procedures of this nature require a reformulation of the traditional mathematical problem. The standard equilibrium equation for proportional loading is $\lambda \mathbf{r} - \mathbf{f} = 0$, where \mathbf{r} is a reference load vector and \mathbf{f} is vector of internal force. The proportional loading factor λ in incremental form yields a general formulation as,

$$\begin{aligned} \mathbf{K}^{(i)} \delta^{(i)} &= \lambda^{(i+1)} \mathbf{r}_{t+\Delta t} - \mathbf{f}_{t+\Delta t}^{(i)} \quad i = 0, 1, 2, \dots \\ &= \mathbf{p}_{t+\Delta t}^{(i+1)} - \mathbf{f}_{t+\Delta t}^{(i)} \end{aligned} \quad (3.10)$$

Note that,

$$\begin{aligned} \lambda^{(i+1)} &= \lambda^{(i)} + \Delta \lambda^{(i)} \\ \mathbf{f}_{t+\Delta t}^{(0)} &= \mathbf{p} \end{aligned} \quad (3.11)$$

We define,

- $\mathbf{K}^{(i)}$ the tangent stiffness matrix at i th iteration
- $\Delta \mathbf{u}^{(i)}$ Incremental displacement vector from the beginning of the step to the end of current iteration cycle (i th iteration)
- $\delta^{(i)}$ Incremental displacement vector of current iteration
- $\lambda^{(i)}$ A load parameter (scale) for the i th iteration

We can write,

$$\begin{aligned} \Delta \mathbf{u}^{(i+1)} &= \Delta \mathbf{u}^{(i)} + \delta^{(i)} \quad i = 0, 1, 2, \dots \\ \mathbf{u}_{t+\Delta t}^{(i+1)} &= \mathbf{u}_t + \Delta \mathbf{u}^{(i+1)} \end{aligned} \quad (3.12)$$

and $\Delta \mathbf{u}^{(0)} = 0$. In order to find $\delta^{(i)}$, we rewrite Equation (3.10) into two parts

$$\mathbf{K}^{(i)} \bar{\delta}^{(i)} = \lambda^{(i)} \mathbf{r}_{t+\Delta t} - \mathbf{f}_{t+\Delta t}^{(i)} \quad (3.13)$$

$$\mathbf{K}^{(i)} (\delta_T)^{(i)} = \mathbf{r} \quad (3.14)$$

therefore,

$$\delta^{(i)} = \bar{\delta}^{(i)} + \Delta\lambda^{(i)}(\delta_T)^{(i)} \quad (3.15)$$

where the displacement $(\delta_T)^{(i)}$ due to a unit load factor multiplied by the incremental variation in the load level $\lambda^{(i)}$, and the displacement $\bar{\delta}^{(i)}$ update for a conventional “load controlled” Newton procedure due to the unbalanced load.

It is seen that in order to completely define $\delta^{(i)}$, one must find $\Delta\lambda^{(i)}$.

Thus, consider a constraint arc length,

$$\begin{aligned} (\Delta l^{(i+1)})^2 &= (\Delta l^{(i)})^2 \\ &= \left(\Delta \mathbf{u}^{(i+1)T} \Delta \mathbf{u}^{(i+1)} \right) + (\lambda^{(i+1)} - \lambda^{(0)})^2 \\ &= \left(\Delta \mathbf{u}^{T(i)} \Delta \mathbf{u}^{(i)} \right) + (\lambda^{(i)} - \lambda^{(0)})^2 \end{aligned} \quad (3.16)$$

Based on above equations, $\Delta\lambda^{(i)}$ and $\delta^{(i)}$ can be solved.

This procedure can be applied with either a modified or full Newton-Raphson iteration scheme. The relationship between these components in the load-displacement space is displayed as shown in [Figure 3.6](#).

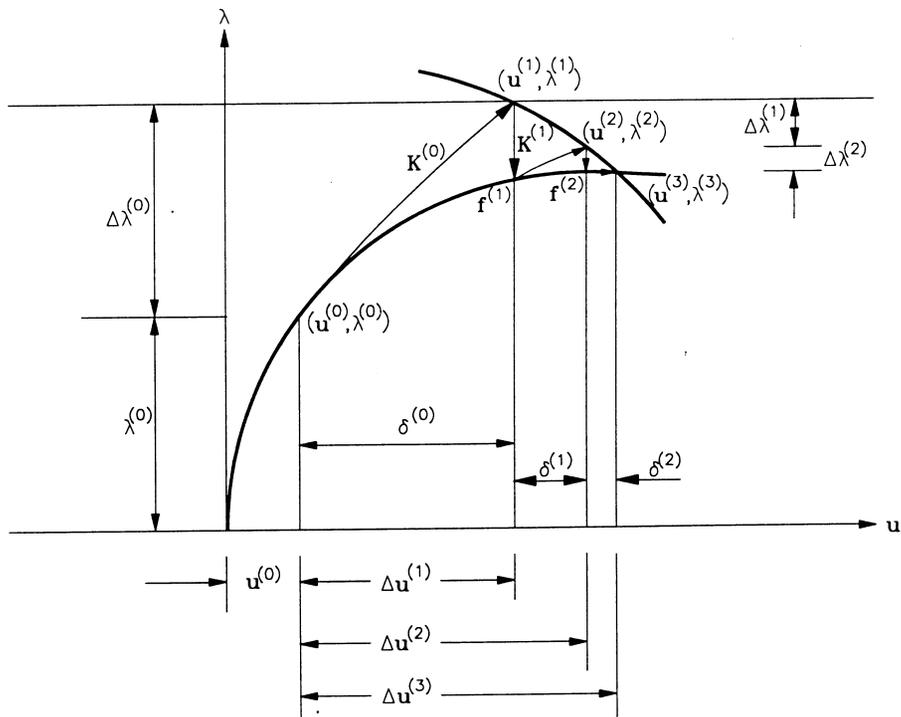


Figure 3.6: Arc Length Solution Algorithm

3.4.4 Basic Analysis Output

The basic output from a typical nonlinear static analysis run is briefly described in what follows. Tabulated description of output options and the data groups requesting them as well as the postprocessing features are given in [Section 3.14](#). The following is a list of the basic output available for selected load steps in nonlinear static analysis:

- Displacement components at selected nodes.
- Reactions at selected constrained nodes.
- Element stresses and stress intensities for selected elements.
- Averaged nodal stresses and stress intensities for selected nodes.
- Stress filtering and sorting.
- Effective plastic strains for selected elements.
- Normal contact pressure for contact surfaces.
- Penetration and contact status for contact elements

Graphical representation of the results, e.g., deformed shapes, multiple deformed shapes at various steps, stress and strain contours, xy plots, etc. may be obtained using the DISPLAY program.

3.5 Linear and Nonlinear Dynamic Analysis

3.5.1 Overview of Analysis Types and Capabilities:

Various Analysis Types:

Linear Dynamic analysis involves computation of responses of a linear system to time dependent loads as detailed in Section 2.4. Depending on the assumptions made for the characteristics of the stiffness, mass and damping matrices and the load vector in [Equation \(2.90\)](#), different analysis types are obtained. When the inertia effects are not significant and when the damping is zero, the responses are pseudostatic and can be obtained by STATIC analysis ([Section 3.3](#)). When the inertia effects are appreciable, the responses can be obtained by the different dynamic analysis types available in NISA as follows:

1. EIGENVALUE Analysis which involves the analysis of undamped free vibration of a structure. In the absence of damping and applied loads, the equations of motion ([Equation \(2.90\)](#)) reduce to a generalized eigenvalue problem ([Equation \(2.93\)](#)) which can be solved by employing the various algorithms available in NISA. This analysis type is described in [Section 3.5.2](#).
2. TRANSIENT Dynamic Analysis which involves the computation of the response of the system when the loads are arbitrary but known functions of time. In NISA, transient dynamic analysis can be performed by the use of modal transient dynamic analysis (modal TRANSIENT Analysis) or by the use of direct integration of the equations of motion (LTRANSIENT and NLTRANSIENT analyses). The former requires an EIGENVALUE analysis to be done before computation of transient response and is detailed in [Section 3.5.3](#), [Section 3.5.4](#). The direct linear transient analysis is explained in [Section 3.5.8](#) and the nonlinear transient analysis is discussed in [Section 3.5.10](#)
3. RANDOM Vibration Analysis which computes the statistical measures, such as power spectral density (PSD) functions and RMS values of the responses of the structure given the PSDs of stationary or nonstationary input loads. This analysis is based on the modal superposition method and is detailed in [Section 3.5.3](#) and [Section 3.5.5](#)
4. FREQUENCY Response Analysis which involves computation of steady state response of structures subjected to harmonic loads. In NISA, this analysis can be performed by either using modal superposition (modal FREQUENCY) or by solving the equations directly in frequency domain (DIRECT FREQUENCY RESPONSE ANALYSIS). The former requires an EIGENVALUE analysis to be done before computation

of frequency response and is detailed in [Section 3.5.3](#) and [Section 3.5.6](#). The direct frequency response analysis is explained in [Section 3.5.9](#).

5. SHOCK Spectrum Analysis which estimates the maximum responses of a structure under a support excitation defined in terms of its shock (response) spectrum. This analysis is explained in [Section 3.5.3](#) and [Section 3.5.7](#).

Mass Formulation

In NISA, the inertia properties of the system can be modeled using point mass elements (See [Section 4.22](#) - [Section 4.30](#)) and/or by specifying a nonzero material density. The latter results in the automatic computation of element mass matrices. In NISA, this can be done by using either consistent or lumped mass formulation. Consistent mass matrix is so called because its computation is consistent with the way in which the stiffness matrix is computed in the finite element formulation. This matrix is banded (coupled) and usually gives more accurate results than the lumped mass matrix. The lumped mass matrix, on the other hand, is diagonal and in a few cases gives better results. Since all the eigenvalue extraction algorithms and direct integration procedure take advantage of sparsity of the lumped mass matrix, the computational time is less than that with the consistent mass matrix. For example, the eigenvalue extraction using subspace iteration is about 30 percent faster with the lumped mass formulation. However, this advantage is reduced in eigenvalue extraction if kinematic constraints like rigid elements or multipoint constraint equations, which introduce coupling in the global mass matrix, are used. The type of mass matrix formulation is specified by the executive command 'MASS FORMULATION' (see [Section 5.3](#)).

Damping

Damping is caused by inherent nonconservative forces in the system leading to dissipation of energy. These damping forces are not necessarily linear functions of displacement or velocity. The resistance of air to moving bodies, for example, is found to be approximately proportional to the square of the velocity of the body. In most actual systems, it is difficult to derive such expressions for damping forces. The differential equations of motion that result from using nonlinear expressions for damping forces may be difficult to handle. Hence it is customary to replace the damping forces in the system by equivalent damping either proportional to velocity (viscous) or proportional to displacement (structural). A detailed description on the nature of damping can be found in Hurty and Rubinstein [3.6].

Viscous damping is aerodynamic in nature and is present when the structure vibrates in a fluid (e.g., air). The damping force is proportional to and directed against, the velocity. In

NISA, viscous damping may be used in all types of dynamic analyses. The damping is specified by one or more of the following methods:

- (i) Discrete viscous damper elements (dash pots) can be used only in direct transient analysis (see [Section 4.36](#)). These are damping counterparts of spring elements and represent discrete idealizations of viscous damping in the structure. These elements can be specified in addition to proportional viscous damping described below.
- (ii) The global damping matrix can be proportional to the global stiffness and mass matrices (proportional viscous damping) as shown in [Section 2.4.4](#). This is also known as Raleigh's damping. Proportional viscous damping is available in both the modal dynamic (*DAMPING) and direct transient (*PDAMPING) analyses. For direct transient analysis, the concept of proportional damping may be extended to element level. In this case, the element damping matrix is proportional to the element stiffness and mass matrices. Since the proportionality constants can vary from element to element, the global damping matrix resulting from the assemblage of such element damping matrices may not be proportional to the global stiffness and mass matrices.
- (iii) Modal viscous damping involves directly specifying damping ratios for all the participating modes. The modal damping ratio is the ratio of actual damping to critical damping in that mode of vibration. As the name implies, this is available only in the modal dynamic analysis (*DAMPING). Currently, only underdamped systems can be analyzed. The damping ratios for all the modes should be less than 1.0.

Structural damping is the result of internal friction within the material and at the connections between elements of the system. The damping force is proportional to the elastic force but is opposite to the velocity. In NISA, Structural damping can be used only in Random Vibration and Frequency Response analyses and is specified as modal damping ratios for all the participating modes (*DAMPING).

3.5.2 Eigenvalue Analysis (EIGENVALUE)

Eigenvalue analysis deals with undamped free vibration of a structure. Bifurcation buckling analysis also involves eigenvalue extraction and is discussed in [Section 3.6](#).

Eigenvalue analysis, thus, does not represent response due to any loading, but yields the natural frequencies (eigenvalues) and corresponding mode shapes (eigenvectors) of the structure when there is no dissipation of energy due to damping. A structure with a non-zero initial condition (initial displacement or velocity) corresponding to any of the mode shapes

will exhibit simple harmonic motion at the corresponding natural frequency. The amplitudes of the free vibration will depend on the initial conditions and in the absence of damping, the vibration will continue without any decay.

Though the above description reflects the true nature of eigenvalue analysis, more often this analysis is carried out to get the dynamic characteristics of the structure in terms of its natural frequencies and mode shapes, which may later be used for efficient computation of the response in the presence of dynamic loads and damping in modal dynamic analysis (also known as normal mode or modal superposition).

Eigenvalue analysis involves a much higher computational effort than static analysis. For large problems it is recommended that a static analysis be done to check the model before performing an eigenvalue analysis.

NISA currently has the following five methods for eigenvalue extraction (see executive command 'EIGEN EXTRACTION' in [Section 5.3](#)):

1. Conventional Subspace iteration.
2. Accelerated Subspace iteration.
3. Inverse iteration using Sturm sequence count.
4. Lanczos Method.
5. Shifted Block Lanczos Method.
6. Guyan Reduction followed by Householder QR iteration.

Different methods of eigen extraction have been provided since there is no method that is efficient in all situations for all problems. These methods employ either transformation or tracking procedures.

Both the conventional and the accelerated subspace algorithms use simultaneous inverse iterations with a set of vectors until the eigenvalues have converged to a user specified tolerance. Each iteration is a reduction using a vector subspace and the reduced eigen problem is solved using the Householder-QR method. The conventional subspace method is an iterative transformation technique. The accelerated subspace iteration employs shifting to accelerate the convergence of roots far away from the lower end of the eigenvalue spectrum, and hence can be termed a tracking and transformation method.

The inverse iteration algorithm first isolates roots by successive bisection and Sturm sequence count. The eigenvalues and the corresponding vectors are then computed by inverse iteration in each isolated eigenvalue region. The inverse iteration is a tracking method.

The Lanczos method uses a recurrence relation to reduce the eigen problem to a block tridiagonal form and the QR method is employed to derive the eigenvalues of the block tridiagonal matrix. Size of the block can be specified by user, else it will be computed rationally by the algorithm. The Lanczos steps are terminated when the required number of eigenvalues have converged to a user specified tolerance. The Block Lanczos algorithm is a non-iterative transformation method.

In Shifted Block Lanczos method, Block Lanczos algorithm is called multiple times each one with a new shift. The new shift is chosen based on the eigenvalues already converged at the end of the specified number of iterations. The orthogonality is preserved at every shift.

Guyan Reduction involves reducing the eigen problem to a few master degrees of freedom, selected either by the user or automatically by the program, and solving the system by Householder transformation and QR iterations. Thus, this can be termed a transformation method.

Both consistent (or coupled) and lumped (or diagonal) mass matrices can be used for eigenvalue extraction in NISA. Rigid body modes (zero frequency modes) that are present in an unconstrained or a partially constrained structure can be handled. In addition, a Sturm sequence check may be requested to obtain all the natural frequencies below a user specified frequency limit or to report the number of frequencies below the limit while using the subspace iteration or the Lanczos algorithm. In inverse iteration, Sturm sequence check is always done at both upper and lower cutoff frequency limits and only the frequencies in the bounded region are computed.

The output from eigenvalue analysis consists of:

1. Natural frequencies and the computational tolerances on eigenvalues.
2. Printout of selected eigenvectors (mode shapes) which are normalized with respect to the mass matrix. The printout may be requested for a subset of the nodal points.

3. Printout of modal participation factors, modal masses, and cumulative modal masses under a ground motion (support excitation) in all the six global directions (three translations and three rotations).
4. Selective computation/printout of element modal internal forces, element modal strain energies, percentage strain energies and strain energy densities.
5. Selective computation of modal reaction coefficients and printout of modal reactions.
6. Selective computation/printout of element modal stresses and averaged nodal stresses.

The above items are described in a tabular form along with the data groups needed to request their computation in [Section 3.14](#). It should be noted that if a particular response quantity, such as stress, is to be computed in a subsequent modal dynamic analysis, the corresponding response quantity, in this case modal stress, must be computed for all the modes to be included in the subsequent analysis.

Choice of Eigenvalue Extraction Method and Tolerance

The choice of the eigenvalue extraction method depends on the nature and size of the problem, the number of eigenvalues required, and whether it is to be employed for obtaining the natural frequencies or the buckling load factors for a structure. A comparison of the eigenvalue extraction methods available in NISA is given in [Table 3.1](#).

Generally, the Lanczos and the accelerated subspace methods are faster than other methods. For most large size problems, the Lanczos algorithm has the edge in speed. Both these algorithms benefit when the roots are well separated. If the eigenvalues are far apart, the accelerated subspace employs aggressive frequency shifts and closes in on the roots to expedite their convergence. The Lanczos method can be slower when eigenvalues appear in a cluster of close or multiple roots. This is because the nature of the Lanczos method is such that convergence is not strictly monotonic towards all the eigenvalue in a cluster. The algorithm sets in on one eigenvalue but is slow to converge on others of the same cluster until some higher eigenvalues have been produced. Thus, if the last few eigenvalues requested by the user happen to lie in such a cluster, additional Lanczos steps may be required in order to find all the roots. A Sturm sequence check is employed to ensure that close or multiple roots are not missed out.

Table 3.1: Comparison of the eigenvalue extraction methods available in NISA

METHOD	TYPE	BEST PERFORMANCE IN TERMS OF CPU TIME	RESTRICTIONS	PARAMETERS UNDER USER CONTROL
ACCELERATED SUBSPACE	Tracking and Transformation	Roots well separated. Large systems of size 200-10,000 dof. Many eigenvalues, greater than 100.	None	Tolerance Initial Shift Size of Subspace Maximum Iterations
CONVENTIONAL SUBSPACE	Transformation	Close roots. Small systems of size less than 200 dof. Few eigenvalues, less than 10.	None	Tolerance Initial Shift Maximum Iterations
INVERSE ITERATION	Tracking	Few eigenvalues required in a given range.	May not extract zero eigenvalue	Tolerance Lower Cutoff Frequency Upper Cutoff Frequency Maximum Iterations
LANCZOS	Transformation	Roots well separated. Very large systems of size greater than 10,000 dof. Intermediate number of eigenvalues, between 10-100.	Not recommended for buckling analysis when negative load factors are expected	Tolerance Initial Shift Minimum Steps Maximum Iterations
GUYAN	Transformation	Large systems. Intermediate number of eigenvalues or eigenvectors representing only certain behavior of the structure is required.	None	Master degrees of freedom
* These parameters are specified using the *EIGCNTL data group.				

The conventional subspace algorithm is more efficient when the roots are closely spaced and when only the first few eigenvalues (say less than 10) are needed. The Lanczos algorithm is recommended for extracting an intermediate number of eigenvalues (between 10-100) and the accelerated subspace iteration would be faster if more than 100 eigenvalues is needed. The conventional subspace is best suited for small models of size less than 200 dof, and the accelerated subspace and the Lanczos method are most efficient for large systems.

When eigenvalues in a specified region are required and when this region is far away from the lower end of the eigenvalue spectrum, inverse iteration may be more cost effective than the other methods. This will be the case when the number of roots below the lower limit is large compared to the number of roots within the lower and upper limits of the region of interest. For example, when one is interested in natural frequencies between 30 and 50 Hz for a structure having twenty modes between 0 and 30 Hz and only five between 30 and 50 Hz, inverse iteration should be used.

Guyan Reduction followed by Householder-QR iteration is recommended when intermediate number of eigenvalues are required in large to very large systems. This method involves static condensation of the problem to a few master degrees of freedom selected by the user or automatically selected by the program. Thus, this method is inherently approximate and must be used with caution. But, by proper selection of master degrees of freedom, the user may be able to get a fairly accurate estimate of the eigenvalues.

For buckling analysis, inverse iteration or either of the subspace iteration algorithms can be employed. Lanczos method is not recommended for buckling analysis when negative load factors are expected. While using subspace iteration algorithm in buckling analysis, it should be noted that only the roots closer to zero (either positive or negative eigenvalues) are extracted. If only all the positive (or only all the negative) eigenvalues are desired, inverse iteration with suitable lower and upper limits should be adopted.

Choice of Parameters for Eigenvalue Analysis

The default convergence tolerance in NISA is 10^{-5} . This tight tolerance is used to achieve high accuracy and is not commensurate with the errors associated with finite element approximation. A less tighter (higher) tolerance can be applied especially in a large problem to minimize the computational time. A value of 10^{-3} to 10^{-4} is suggested. Reducing the convergence tolerance, on the other hand, increases the accuracy of solution and the computational effort.

While using accelerated subspace iteration, the important parameter that can have a significant effect on the computational time is the dimension of subspace (i.e., the number of iteration vectors). Increasing the number of vectors improves the convergence rate and hence reduces the number of iterations required for convergence. However, the number of operations per iteration increases rapidly as the number of vectors becomes larger. Using less number of vectors increases the number of iterations but the computational effort per iteration is reduced. The optimum value for the dimension of subspace to minimize the total computational effort is a function of the spacing of eigenvalues, the number of eigenvalues to be evaluated, and the number of operations involved in each decomposition indicated by the wavefront solution parameters. NISA uses a default value which is probably close to the optimum for most of the problems (see [Section 7.1.2](#)). The user, however, has the option of specifying an alternate value for this parameter. In addition, the user can prevent an inordinate computational effort by specifying the maximum number of iterations to be used, the default value for which is set at 20.

In the Lanczos method, the recursion is employed at each step to derive the elements of the tridiagonal matrix. However, the eigenvalues of this tridiagonal matrix are not calculated at each step. The Lanczos algorithm is first run through a minimum number of steps before eigenvalue solution is attempted using QR iteration. This is the estimated number of steps needed for convergence of all the desired roots, and this value may be set by the user. After the minimum number of steps are performed, the steps will continue with eigenvalue evaluation performed at each step while checking for convergence of the roots. If the estimated number of steps is much less than that actually needed, eigenvalue solution is performed at all the following steps till convergence. If, on the other hand, a conservative (higher) estimate is used, eigenvalues are calculated only once, well after convergence has been achieved. This may result in a higher or lower computational effort depending on how many additional steps were done. The cost of QR iterations itself increases rapidly with the increasing steps. NISA establishes a default value for the minimum steps based on an empirical formula (see [Section 7.1.2](#)). Although this default value is close to the optimum, it can be unconservative for some cases where a large number of eigenvalues are required. In addition to this parameter, the user can also specify a maximum number of steps to be taken.

Guyan reduction is very useful when the user wishes to extract eigenvectors corresponding to a certain behavior of a large structure. For example, if the user is interested in doing a modal dynamic analysis of a plate subjected to out-of-plane forces. The user can, in this case, select the out-of-plane degrees of freedom (UZ, ROTX, and ROTY) as the master degrees of freedom. Master degrees of freedom may be specified by the user, selected

automatically by the program or obtained by a combination of both (see [Section 7.1.2](#) and [Section 7.3.12](#)).

The user may be able to obtain sufficiently accurate results by choosing the degrees of freedom that correspond to the lowest modes as masters. For those structures where bending modes are the lowest modes of interest, the user can neglect the in-plane degrees of freedom and rotational degrees of freedom. The number of master degrees of freedom chosen should be at least two to three times the number of eigenvalues sought.

The automatic selection of master degrees of freedom in NISA, is based on the ratio of the diagonal elements of the global mass and stiffness matrices. Those degrees of freedom with the least value for this ratio are selected as the masters. This method has been found to give good estimates for the lowest eigenvalues.

Rigid Body Modes

All the eigenvalue extraction algorithms in NISA except the inverse iteration can handle rigid body (zero frequency) modes conveniently. These modes arise if the structure is unconstrained (free-free) or partially constrained like the case when internal mechanisms are present. This results in a singular stiffness matrix. If a singularity is detected during decomposition of the stiffness matrix, NISA aborts decomposition, introduces a negative shift thus removing the singularity and restarts the decomposition process. This may involve considerable computational effort. Hence, for structures where rigid body modes are suspected to be present, the user has the option of specifying a negative initial shift. If shifting does not eliminate the singularity as in the case when ‘massless mechanisms’ are present, NISA aborts after three successive attempts to remove the singularity by perturbing the shift. While using the accelerated subspace iteration algorithm, the user also should ensure that the dimension of subspace is larger than the number of rigid body modes expected. If the default size of the subspace is lower, then a larger value has to be specified.

3.5.3 Modal Dynamic Analysis

Modal dynamic analysis (Normal mode analysis) is an efficient method of forced vibration analysis. The normal mode method is based on the fact that the equations of motion are decoupled by expressing the responses in terms of the modal responses. Thus modal dynamic analysis involves forming the modal loads, solving for the modal responses (generalized coordinates) in each mode of vibration and computing the physical responses such as displacements, velocities and stresses through modal superposition.

The following modal dynamic analysis types are available:

1. Transient Dynamic Analysis (TRANSIENT)
2. Random Vibration Analysis (RANDOM)
3. Frequency Response Analysis (FREQUENCY)
4. Shock (response) Spectrum Analysis (SHOCK)

The choice of a particular analysis type depends on the load description that is available to the user and the form of response output that is desired.

In NISA, the modal dynamic analysis is a two-step procedure. The first step is an eigenvalue analysis to determine the dynamic characteristics of the structure in terms of its natural frequencies, mode shapes, participation factors, modal stresses, etc. This information is stored in NISA files 26 and 27 which should be saved. The second step is the modal dynamic analysis using any of the analysis types mentioned above.

In what follows general topics common to all modal dynamic analysis types are discussed. Detailed description of each analysis type follows, beginning with transient dynamic analysis in [Section 3.5.4](#). The output pertinent to each analysis type is included in the corresponding sections for ready reference and also presented in a tabular form along with the postprocessing features in [Section 3.14](#).

Mode Selection

The efficiency of modal dynamic analysis stems from the fact that fairly accurate results can be obtained for a structure under a dynamic loading by using few modes in the analysis. By exercising care in selecting modes to be used in the modal superposition process, the user can obtain acceptable results using a small fraction of the total number of modes in the model, which is equal to the total number of degrees of freedom. The guidelines suggested herein can be used while selecting the modes for modal dynamic analysis. Modes are selected using the *MODESELECTION data group, [Section 8.5.1](#).

One factor that affects the participation of a particular mode in the response is the spatial orientation and distribution of the loads. A mode with predominant Y-displacements, for example, will be excited by dynamic loads mostly oriented in the Y-direction. A mode with no appreciable displacement corresponding to a particular degree of freedom, on the other hand, will show little effect due to loads applied to that degree of freedom. For example, if a mode has close to zero displacement in Z-direction at a node, it will not respond to a load

applied at the node in the Z-direction even if it is predominantly a 'Z-direction mode'. This mode, thus, can be excluded from the analysis performed to obtain the responses due to the load described above. The user can inspect the modes obtained from an eigenvalue analysis and employ only those which may contribute to the responses for the loading at hand during the modal dynamic analysis.

Modal participation factors, modal masses and cumulative modal masses are printed out during NISA eigenvalue analysis for all the modes extracted and for all the six global directions. If a modal dynamic analysis is to be performed for obtaining responses due to ground motion, these values may be used to aid selection of modes. Most design practices require modes with modal masses above a specific percentage, say 5 percent, of the total mass be included in the modal superposition till the cumulative modal mass reaches a specified percentage, say 85 percent, of the total mass.

Another factor that affects the contribution of a specific mode to the total dynamic response is the frequency content of the loading. In frequency domain analyses like random vibration analysis and frequency response analysis, the frequency content can be readily ascertained from the frequency domain description of the load, such as PSD function in random vibration analysis, amplitude spectrum in frequency response analysis and response spectrum in shock spectrum analysis. Modes in the frequency regions with higher values of the load description function will contribute more than, and hence should be preferred to, the modes in the other frequency regions.

In the case of transient dynamic analysis, the frequency content can be determined by following the variation of the load with respect to time. A sudden (impact or blast) load is rich in higher frequencies and hence will excite higher modes (modes with higher frequencies and hence lower periods). A gradual load, on the other hand, will elicit response from the lower modes.

Ground Motion

All types of modal dynamic analysis can be used to evaluate response due to a ground motion (base or support excitation) input. Ground motion can be specified directly as part of the input loads instead of simulating ground motion by using arbitrarily large coefficients in stiffness/mass matrices to convert applied forces into ground motion.

Ground motion input includes the following features:

1. Translational ground motion along the three global axes.
2. Rotational (rocking) ground motion with components specified about axes parallel to the three global axes. The global coordinates of the point about which there is ground rotation can also be specified.
3. The ground motion in each direction may be independently specified in terms of displacements, velocities or accelerations. The displacement and velocity options are not currently available in transient dynamic and random vibration analysis.

In the uniform base excitation, all support points are assumed to follow the same ground motion in a particular direction. The responses (displacements, velocities and accelerations) obtained from NISA are relative to the ground as discussed in [Section 2.4.2](#). However, in multiple support excitation analysis, only total response quantities can be obtained. The option for multiple support excitation is applicable if independent support motions exist or if absolute responses are required.

Absolute Response by Large Mass Concept (For uniform base excitation)

Absolute responses (displacements, velocities and accelerations) under ground motion can be obtained from NISA using the ‘large mass concept’ described herein. This method, though lacking in theoretical rigor, gives acceptable results and is preferred to the ‘exact’ method which involves more matrix manipulations and, hence, more computational time.

The procedure to obtain the absolute responses in modal dynamic analysis is as follows:

1. In the eigenvalue analysis run, the constraint at the support point in the direction of excitation is released, and a very large point mass (or rotational mass if rotational ground motion is to be simulated) is added to the released degree of freedom. If there are more than one node with the displacements in the direction of excitation constrained, then all these displacements can be released and coupled together (see [Section 3.10](#)) to reduce the number of dynamic degrees of freedom. In such cases, the large mass needs to be added at one of the nodes only. The accuracy of the solution can be improved by increasing the value of the large mass relative to the total mass of a structure. A value of 10^6 times the total mass of the structure in the case of translational ground motion, or 10^6 times the mass moment of inertia of the entire structure about the corresponding axis in case of rotational ground motion, is suggested. This value is found to yield sufficiently accurate results. This may result in one or more rigid body

modes in the eigenvalue analysis and precautions as described in [Section 3.5.2](#) may be taken.

2. In the modal dynamic analysis run, a force (or moment in the case of rotational ground acceleration) equal in magnitude to the value of the large mass times the ground acceleration is applied at the released degree of freedom. If all the rigid body modes and the elastic modes are included in the analysis, absolute (total) responses can be obtained. If relative responses are desired, only the elastic modes should be included in the modal dynamic analysis. It should be noted that response quantities other than displacements, velocities and accelerations, such as stresses and reaction forces, are unaffected by the rigid body modes and, hence, can be calculated without including these modes.

Missing Mass Correction

While accuracy of the modal summation largely depends on the number of modes considered, it is appropriate to truncate the modes which are beyond the bandwidth of the signal. However, a correction can be made to account for the truncated modes by taking the static response of these modes. Typically, seismic excitations will have dominant frequencies up to 33 Hz and modes beyond this cut-off frequency may be ignored in the modal summation. However, a provision is made available to specify the cut-off frequency for other types of support excitations.

3.5.4 Modal Transient Dynamic Analysis (TRANSIENT)

Transient dynamic analysis may be used to determine the response of structures subjected to arbitrary time-varying loads. The variation of loads can be represented by piecewise linear curves of time vs. amplitude. As described in [Section 2.4.6](#), for piecewise linear loading, the modal response is evaluated exactly in closed form. The solution for the next time step is derived for the applied loading by treating the responses from the previous time step as the initial conditions. Rigid body motions can also be handled. This is done by employing modified closed form expressions which take into account the zero frequencies of the rigid body modes.

The capabilities and special features available in transient dynamic analysis are as follows:

Input

1. Combination of any number of time dependent support excitations (*GROUND), concentrated nodal forces (*DCFORCE) and pressure loads (*DPRESSURE) can be specified.

2. Combination of any number of time dependent differential support excitations (*MSEXCITATION), concentrated nodal forces (*DCFORCE) and pressure loads (*DPRESSURE) can be specified.
3. The variation of the loads with respect to time can be expressed by means of time functions (*TIMEFUNCTION) and arrival times (*ARRIVALTIME).
4. Different time increments can be specified for different time regions (*TSTEP), in order to reduce computational time without loss of accuracy.
5. Non-zero initial conditions (displacements and/or velocities) can be specified (*INITIAL) at the nodes of the structure.

Output

1. Selective printout and/or plot of time histories of nodal displacements, velocities, accelerations, stresses and reaction forces over a user specified time range.
2. Selective printout of maximum values and times of maxima of nodal displacements, velocities, accelerations, stresses and reaction forces over a user specified time.
3. Printout and/or plot of snapshot (i.e., freeze) of the above responses at a given time or at an instant in time when another specified response peaks.
4. Selective printout of floor response spectra at specified node locations for maximum, average and average plus standard deviation.

Selection of Time Step Size

An overall user defined analysis parameter is the time step size for time integration. Since the integration is performed in closed form, the criterion for selecting the time step size is based on accuracy and not on the stability of the integration algorithm, as is the case for some numerical procedures. In order to obtain the exact modal response for a piecewise linear load function, the step size should be such that all “kinks” in the time function curve are covered. Also, to capture the correct response history, the step size must be small enough to resolve all the participating modes. Usually one tenth of the period of the highest selected mode should be small enough.

The exact integration procedure of the modal equations also allows different time step sizes to be specified for different time regions. This can be useful in some situations when very small time steps are needed only in certain time intervals of interest. For example, the response of a structure may have significant contributions from vibration modes of very high frequencies. However, the responses in these modes diminish quickly because of damping effects. In this case, small time step sizes may be employed in the initial duration of the analysis. Larger step sizes may be used in the subsequent time regions, where the

effect of higher modes are negligible, thus making the analysis more cost-effective. Time step size is specified using the executive command 'DELTATIME', [Section 8.3](#), and the *TSTEP data group, [Section 8.5.3](#).

Time Function and Arrival Time

The time dependent loads applied on a structure are described by time functions given in a piecewise linear fashion (*TIMEFUNCTION). The loading may be a combination of any number of concentrated nodal forces, pressure loads, or components of ground excitation described by their respective load magnitudes and time functions which indicate their pattern of variation with time. Hence, more than one input loading may be referenced to the same time function.

The specification of time dependent loads may be simplified by having an arrival time (*ARRIVALTIME) associated with each load in addition to the time function. For instance, if the time functions that define the loading differ from each other merely by an initial delay in time, then only one time function needs to be specified. The loads would refer to this single time function but would be associated with different arrival times.

Transient Dynamic Analysis under Moving Loads

Dynamic loads on engineering structures such as bridges, runways, roads and railway tracks arise mainly from moving loads. To achieve fast, efficient and reliable transportation it is necessary to understand dynamic response of structures under moving loads. In the case of moving loads, the position of load changes with time. For finite element analysis of structures subjected to moving loads, it is necessary to transform moving loads to appropriate nodal force time histories. In NISA field consistent approach has been used to convert a moving load into suitable nodal forces corresponding to a moving load. The information regarding moving loads are specified through *MVELOAD and *MVEPATH cards. Information regarding magnitude, direction, velocity and number of loads are described in terms of node IDs over which load traverses. These nodes should be described in the order in which they are traversed by the load. The order in which nodes are specified in *MVEPATH decides the direction in which load is moving. Along the path of the load all the nodes should be specified and not a single node should be left out. If there are multiple loads traveling along the same path then magnitude and distance between these loads can be described in *MVELOAD. The position of the loads should be described in terms of their distance from the first load.

It is to be noted that in NISA transient dynamic analysis under moving loads can be performed even in the presence of other dynamic loads.

3.5.5 Random Vibration Analysis (RANDOM)

Unlike other methods of analysis described here, random vibration analysis is nondeterministic in which the input and the output are described in probabilistic fashion. The loading to a structure, in reality, is not known uniquely and at best can only be predicted. This uncertainty in the loading is more pronounced if it is dynamic in nature. One way of representing the possible dynamic load is by a family or ensemble of histories. Then the structure may be analyzed for each of the histories and the responses processed to find the important statistics. Alternatively, the same results can be obtained more efficiently by introducing the statistics in the beginning, and performing a single random vibration analysis.

The theory of random processes which is an outgrowth of probability theory forms the basis for the rigorous treatment of random vibration. A random process can be adequately characterized by an average amplitude and by a decomposition in frequency. The average amplitude most commonly employed is the RMS or root mean square value. The frequency decomposition is indicated by a power spectral density (PSD).

Natural phenomena are inherently nonstationary random processes. While a stationary random process is characterized by a PSD which represents its special distribution of average energy content for all time, a nonstationary random process is characterized by an evolutionary PSD. The evolutionary PSD, if available, shows the evolution with time, of the average energy content in frequency domain. Large class of physical processes are modeled as uniformly modulated nonstationary random processes [3.24] which are more amendable for mathematical treatment with less rigor but at the same time with sufficient accuracy.

The capabilities and special features available in random vibration analysis are as follows:

Input

1. Stationary or nonstationary concentrated nodal forces (*DCFORCE), pressure loading (*DPRESSURE) and/or support excitations (*GROUND) can be specified in terms of their auto-and cross-spectral density functions (PSDs).

2. Stationary or nonstationary concentrated nodal forces (*DCFORCE), pressure loading (*DPRESSURE) and/or differential support excitations (*MSEXCITATION) can be specified in terms of their auto-and cross-spectral density functions (PSDs).
3. The excitation can be specified as groups of fully and/or partially correlated processes (*CORRELATION).
4. An auto- or a cross-spectral density can be specified in the following formats (*PSDFUNCTION):
 - (i) Frequency vs. real PSD values
 - (ii) Percentage octave band vs. power level in decibels
 - (iii) Constant value at all frequencies (white noise)
 - (iv) Frequency vs. real and imaginary components of a complex PSD function

Analysis

1. Integration and response output frequency points may be:
 - (i) User specified
 - (ii) Automatically generated with a higher density of points closer to natural frequencies
2. Different analysis options can be specified to achieve trade-off between cost and solution accuracy:
 - (i) Closed form exact analysis if the input consists of flat (white noise) PSDs
 - (ii) The analysis may be simplified by reducing cross-modal coupling based on a user specified cut-off ratio and the proximity of modes
 - (iii) Numerical integration by Simpson or trapezoidal rules

Output

1. Printout of covariance matrices of modal displacements, velocities, and accelerations.
2. Plot and/or printout of response PSDs, PSD maxima, mean crossing rates and extreme value characteristics of nodal displacements, velocities, accelerations, stresses, beam vertex forces, and reactions
3. Printout of response PSDs, and cross PSDs in file 18.
4. Contour plot and/or printout of RMS values of nodal displacements, velocities, accelerations, stresses, beam-end forces, and reactions.
5. For nonstationary input the root mean square response (RMS) quantities, being functions of time, can be plotted as time histories using DISPLAY III.

6. Snapshot facility is made available for time varying RMS output quantities.
7. Plot and/or printout of response PSD outputs are presently made available only at the end time in case of nonstationary input.

Random Load Description and Correlation of Processes

A random dynamic load may be described by a power spectral density (PSD). In [Section 2.4.8](#) a PSD function was formally defined as the Fourier transform of the autocorrelation function of a random process. The mean square value of a stationary process is the area under the curve of power spectral density vs. frequency. The units of the PSD are accordingly those of (unit of excitation)²/(unit of frequency). Thus, scaling the excitation by a factor α , amounts to multiplying the ordinates of the PSD by α^2 .

PSDs are commonly defined as two-sided functions with a frequency scale from $-\infty$ to ∞ , or as one-sided functions with a frequency scale from 0 to ∞ . Also, the frequencies may be expressed in either cycles/unit-time (e.g., Hz) or in angular frequency (e.g., rad/sec). NISA requires the input of the PSDs in the form of one-sided functions with frequencies expressed in cycles/unit-time. Hence, the ordinates of a two-sided PSD must be scaled up by a factor of two. Further, if the abscissa of the PSD is in angular frequency, the PSD ordinates have to be multiplied by 2π . Thus, a two-sided PSD with frequencies in rad/sec should be multiplied by 4π to convert it to a one-sided PSD with frequencies in Hz.

When a structural system is subjected to more than one load process, in addition to the description of each process given by their PSDs, a measure of correlation between the processes has to be provided. Often the loads are either assumed to be “uncorrelated” or “fully correlated.” However, in NISA a cross-PSD may be used to characterize the degree of correlation between any two random processes. Thus, a “cross-PSD matrix” of dimension $NP \times NP$ is used to describe a loading consisting of NP processes. This PSD matrix, $\mathbf{S}_L(f)$, is Hermitian, with auto-PSDs along the diagonal and cross-PSDs away from the diagonal. An off-diagonal entry (i, j) may be zero indicating that processes i and j are uncorrelated. Consider for example a structure subjected to loads $F_1(t)$ and $F_2(t)$. The PSD matrix for this loading is given by,

$$\mathbf{S}_L(f) = \begin{bmatrix} S_{11}(f) & S_{12}(f) \\ S_{21}(f) & S_{22}(f) \end{bmatrix} \quad (3.17)$$

in which $S_{11}(f)$ is the auto-PSD for process $F_1(t)$, $S_{22}(f)$ is the auto-PSD for process $F_2(t)$, and $S_{12}(f)$ and $S_{21}(f)$ are the cross-PSDs for $F_1(t)$ and $F_2(t)$. Since the matrix is Hermitian, $S_{12}(f)$ is the complex conjugate of $S_{21}(f)$, and only the upper triangle entries of the matrix need to be specified.

If processes $F_1(t)$ and $F_2(t)$ are scaled by factors α_1 and α_2 respectively, the ordinates of $S_{11}(f)$, $S_{22}(f)$, and $S_{12}(f)$ are in turned scaled by α_1^2 , α_2^2 and $\alpha_1\alpha_2$. It must be mentioned that cross-PSDs have to be described in complex form for their characterization of the correlation between the processes to be complete.

Nonstationary Random Vibration Analysis

Nonstationary loads are to be specified as uniformly modulated random processes in the form of $e(t)l(t)$. $e(t)$ is the envelope function (also known as modulating function) which is deterministic. $l(t)$ is a stationary random process whose PSD is to be specified as one-sided function in terms of frequency in cycles per unit time.

Each stationary random process $l(t)$ can be associated with an envelope function. Three types of input forms are provided to describe this envelope function. The input can be a constant value or in the form of tabulated values of time and amplitude. It can also be described in an analytical form of the type $c\{e^{-\alpha t} - e^{-\beta t}\}$ where α and β are parameters controlling the shape of the function. c is a scale factor.

Users can feed envelope function as a constant value to obtain the nonstationary response under a stationary load.

The tabular and analytical forms are particularly useful for obtaining nonstationary response under the special category of the uniformly modulated nonstationary input loads.

Percentage Octave Band vs. Power Level Input

Random loading may also be specified in the form of percentage octave band decibel levels, especially for acoustic pressure input. The forcing power spectral densities are obtained, internally, by converting the percentage-octave-band vs. power level input within the narrow band. The percentage of an octave p for which the power level is specified is given by,

$$F_i/F_{i+1} = 2^p \quad (3.18)$$

in which F_i is the center frequency of the i th band. By assuming that the power at the center frequency of the i th band is the average power in the band, we have the narrow band power level in units of (force)²/Hz as follows,

$$S(f) = \frac{Q_r^2 \times 10^{(0.1\omega_i)}}{F_i [2^{0.5p} - 2^{-0.5p}]} \quad (3.19)$$

in which W_i is the power level in decibels for the i th band and Q_r is a reference force level. For acoustic problems the reference pressure level is 2.90×10^{-9} lb/in². The above two equations are employed to convert the data given as F_i and W_i to power spectral density vs. frequency format to be employed for analysis.

Integration Procedures and Cross-Modal Correlation

An important result from random vibration analysis is the variance or the RMS value of a response quantity of interest. As shown in [Section 2.4.8](#), the RMS value is calculated by integrating the PSD of the response. If the input PSD function is of a general shape these integrals have to be performed numerically. In order to traverse the peaks and troughs of a response PSD during integration, an automatic frequency point generation feature is provided. An increased number of frequency points are generated in the vicinity of the natural frequencies. The lower the damping for a certain mode of vibration, the steeper the PSD gradients will be around that natural frequency, requiring a higher density of frequency points in that region. The criterion for frequency point selection is described in [Section 8.2.6](#).

The Simpson and trapezoidal procedures are available in NISA for numerical integration. For most problems the Simpson integration gives the best results. However, the trapezoidal integration is recommended if the selected natural frequencies are far apart (this is likely when rigid body modes are present). If the input PSD is flat (i.e., white noise) the integration for RMS values in NISA is performed more efficiently using closed form expressions. The exact integration option may be invoked even if the input consists of a general shaped PSD, in which case an “equivalent flat PSD” approximation of the input PSD would be calculated internally, and employed for the analysis.

The modal responses are calculated as given in [Section 2.4.8](#). The modal PSD, and covariance matrices are of order NMODE which is the number of the participating modes selected for the analysis. The modal matrices are symmetric, and thus, at most

$NMODE \times (NMODE + 1)/2$ number of entries may need to be computed. An off-diagonal entry in the modal matrix represents the degree of modal correlation between the respective modes. Typically, the contribution from the matrix entries become negligible when moving away from the diagonal, especially if the natural frequencies of the modes are far apart. A feature is provided in NISA (executive command ‘ACUTOFF’, [Section 8.3](#)) to control the entries of the modal matrix that will be considered in the analysis. This is done by defining a cut-off ratio which will be employed to derive a “modal-bandwidth” based on the proximity of the modes. Only those off-diagonal entries within the bandwidth satisfying the ratio requirement are actually calculated. This approximation can amount to significant savings in the analysis with negligible loss in accuracy.

Reliability Estimation

As mentioned above, the loading for a random vibration analysis and the resulting response are both statistically described. Since the analysis is nondeterministic there is no guarantee that a certain response will never exceed a given design level. However, by employing statistical response parameters such as the RMS value and the zero mean crossing rate (MCR) generated in NISA, the probability of exceeding the design level may be determined. In addition, the zero mean crossing rate (also known as the apparent frequency), shape factor, extreme value of the response quantity α (EXTDP) with probability equal to p (DPROB) in the time interval T (TSEISMIC) and expected value of the number of crossings (LVLCROSS) of a level defined by α (EXTDP) with a positive slope in the time interval T (TSEISMIC) for all responses can be obtained. This can form the basis for estimating the reliability or safety of a structure.

A stationary random process was described in [Section 2.4.8](#) using measures such as PSD and auto-correlation function, but no mention was made about an associated probability distribution. Often a process is assumed to be normal or Gaussian and of zero mean. For a zero mean process the RMS value of the response is equal to its standard deviation σ_y . The zero MCR $\eta_0^{(+)}$ is the rate of positive crossings of the level $y = 0$ averaged over the samples of the process $y(t)$.

Estimating exceedence probabilities based on σ_y , and η_0^{\dagger} fall under the category of “first passage problems”. The positive crossing rate at a level $y = a$ for a Gaussian process with zero mean is given by

$$\eta_a^+ = \eta_o^+ e^{\frac{-a^2}{2\sigma_y^2}} \quad (3.20)$$

Using further analysis and assumptions, the probability function for the time T to the first such crossing may be shown to be,

$$p(T) = 1 - e^{-\eta_a^+ T} \quad (3.21)$$

Shape factor of a random process is the measure of bandwidth of the process and is given by,

$$k = \sqrt{2\pi(1 - \lambda_1^2 / (\lambda_0\lambda_2))} \quad (3.22)$$

Extreme value of the response quantity α (EXTDP) with probability equal to p (DPROB) in the time interval T (TSEISMIC) is given by,

$$\alpha = \sqrt{-\ln\left\{-\ln(p)\frac{2\pi\sigma_{\dot{y}}}{T\sigma_y}\right\}}2\sigma_y \quad (3.23)$$

Expected value of the number of crossings (LVLCROSS) of a level defined by (EXTDP) with a positive slope in the time interval T (TSEISMIC).

$$N(\alpha;T) = \frac{T\sigma_{\dot{y}}}{2\pi\sigma_y} \exp\left\{\frac{-\alpha^2}{2\sigma_y^2}\right\} \quad (3.24)$$

The probability $p(T)$ and $N(\alpha;T)$ may be used for design in two ways:

1. To estimate the time when $y(t)$ reaches a certain failure level $y = a$ for the very first time.
2. To estimate the time to failure due to an accumulation of damage. Each excursion of $y(t)$ does a small but definite amount of damage depending on the amplitude $y = a$, and failure occurs when these damage increments reached a fixed total.

An example of the second type of failure is the fracture of metal elements due to fatigue. A more detailed presentation on structural reliability may be found in Lin [3.7] or Augusti [3.8].

3.5.6 Frequency Response Analysis (FREQUENCY)

Frequency response analysis is an efficient method for finding the steady state response of a structure subjected to harmonic loading. This analysis feature in NISA is not the traditional frequency domain transient dynamic analysis (see Clough [3.5]) in which a Fourier transform is performed on the time history of the load, followed by analysis in the frequency domain and an inverse Fourier transform to recover the time history of the response. In NISA frequency response analysis, sinusoidal harmonic load or loads are represented by their amplitude and phase at the “exciting” frequency or frequencies, and the response of the system in terms of amplitude and phase, is evaluated at these frequencies. Thus, the load description, the analysis, and the response quantities are all in the frequency domain.

The capabilities and special features available in frequency response analysis are as follows:

Input

1. Frequency dependent harmonic concentrated nodal forces (*DCFORCE), pressure loading (*DPRESSURE), and ground excitations (*GROUND) may be specified.
2. Frequency dependent harmonic concentrated nodal forces (*DCFORCE), pressure loading (*DPRESSURE), and ground excitations (*MSEXCITATION) may be specified.
3. Unit harmonic nodal forces, displacements, velocities and accelerations may be specified at nodes for point-to-point transfer function calculations (*DRIVER).
4. Amplitude and phase spectra may be expressed as functions of the exciting frequencies which may be in cycles/unit-time, radians/unit-time, or period (*SPECTRUM).

Output

1. Amplitudes and phase angles of modal displacements, velocities and accelerations.
2. Printout and/or plot of amplitudes and phase angles of selected nodal displacements, velocities, accelerations, stresses, reactions, and beam vertex forces.

Amplitude and Phase Input and Output

A harmonic forcing function may be expressed as,

$$f_1(t) = F_1 \sin(\omega t + \phi_1) \quad (3.25)$$

in which F_1 is the amplitude, ω the frequency, and ϕ_1 the phase lag. The values F_1 and ϕ_1 may be represented by points at frequency ω on an amplitude and a phase spectrum, respectively. The response from two or more harmonic loads having the same exciting frequency ω and different amplitudes F_1, F_2 , etc., and phase angles ϕ_1, ϕ_2 etc., is evaluated by superposition. The loading having different exciting frequencies or harmonics is treated separately. If a number of exciting frequencies are employed (i.e., perform a “frequency sweep” of the structure) then the response may be obtained as smooth amplitude and phase spectra. The response in general is given by,

$$u_1(t) = U_1 \sin(\omega t + \psi_1) \quad (3.26)$$

in which U_1 is the amplitude and ψ_1 the absolute phase lag. Thus, the phase lag of this response $u_1(t)$ with respect to the load $f_1(t)$ is $(\psi_1 - \phi_1)$.

If the exciting frequency is in the vicinity of a natural frequency of the system the response will be greatly magnified. A feature is provided in NISA to automatically generate a list of exciting frequencies with a higher density of points surrounding the natural frequencies. This feature, described in [Section 8.2.6](#), ensures that the possible peak responses for a system are not missed out, and also provides smooth plots of the response vs. frequency curves. The magnification of the response is a maximum when the natural frequency and the exciting frequency coincide (i.e., resonance occurs), and for zero damping this value is infinite. However, the system response for zero damping is finite at an exciting frequency which does not coincide with one of the natural frequencies, although such a result is only of academic interest since the response of the system cannot reach steady state if no damping is present.

Transfer Function Evaluation (*DRIVER)

The transfer function is, in effect, a dynamic equivalent of the familiar flexibility coefficient for a structure as we know it in static analysis, i.e., it is the response at one DOF in the structure due to a unit input at another DOF. The point-to-point transfer functions may be

evaluated in NISA at all the DOFs if a unit harmonic input function of force, displacement, velocity, or acceleration is imposed at a particular “driver” DOF of the structure. A phase angle may be associated with a driver input.

The transfer function derivation for a unit force is straightforward, since it is the response from a frequency response analysis with a unit concentrated force assigned along the driver DOF. However, the transfer functions for unit displacement, velocity, or acceleration is evaluated by scaling the amplitude, of the response obtained from a concentrated force analyses, such that the desired driver response is unity. The concentrated force in the latter case is also applied along the DOF for which the displacement, velocity, or acceleration transfer function is required.

A frequency response analysis for deriving transfer functions ignores other forms of excitation such as ground motion and pressure since it is, in fact, not a load-response analysis. A transfer function is by definition a property of the structure and is usually derived as a calibration parameter or as a prelude to other analyses. For example, some early random vibration analysis approaches required separate point-to-point transfer function calculations as a first step.

3.5.7 Shock Spectrum Analysis (SHOCK)

The shock spectrum analysis, also known as response spectrum analysis, may be used to estimate the maximum response of a multi degree of freedom structure subjected to an arbitrarily oriented foundation shock spectra input ([3.5]). The input for shock spectrum analysis differs from the other modal dynamic analysis types. Here, instead of directly specifying the ground motion input, the user specifies the maximum responses that the input will produce on single degree of freedom systems having various natural frequencies and damping ratios. Thus, the maximum responses are given in the form of a spectrum known as response or shock spectrum. The inherent assumption in this analysis is that the excitation is the same at all support points. Maximum relative nodal displacements, velocities, accelerations and stresses are determined by combining the modal and directional maxima using the superposition rules chosen by the user. The maximum response values obtained from the shock spectrum analysis are thus all positive and do not represent the state of the structure at the same instant in time.

The capabilities and special features available in shock spectrum analysis are as follows:

Input

1. The spectra input can be a single spectrum curve or a family of multiple damping spectra and is usually furnished by the user. It can also be one of the prestored spectra from the spectrum library. The spectra input is specified using the *GROUND data group (Section 8.7.2) or *MSEXCITATION data group (Section 8.6.9-for multiple support excitation) which may refer to the *SPECTRUM or the *MDSPECTRUM data groups (Section 8.6.5 and Section 8.6.6, respectively).

Output

1. Maximum responses of nodal displacements, velocities, accelerations, stresses and reaction forces, elemental centroidal stresses and stress resultants, base shear forces can be printed and/or stored for postprocessing. The following combination rules are available:
 - (a) ABS or SRSS combination procedures may be used to combine the responses across directions (executive command 'DIRECTION', Section 8.3).
 - (b) ABS, SRSS, CQC and PRMS/NRL, GRP, TPM and DSM combination rules may be used to combine the responses of different modes (*RESPONSE, Section 8.8.7).

Shock Spectrum Description

The user can specify displacement, velocity or acceleration shock spectra over a frequency range for translational and/or rotational ground motion. If a velocity or acceleration spectrum is specified, it is converted into a displacement spectrum as shown in Section 2.4.9. This involves the assumption of spectral pseudo velocity and pseudo acceleration, and the displacement spectrum thus obtained is only approximate. The input spectrum can be expressed as a function of frequency or period. The modal maxima are computed from the shock spectra by interpolation. Linear, semi-logarithmic, or logarithmic interpolation can be employed for this purpose.

The shock spectrum can be a single spectrum curve independent of damping or a family of spectrum curves corresponding to different damping values. Linear interpolation is used for interpolating with respect to damping across two spectrum curves.

Spectrum Library

Currently, NISA has the following prestored spectra which can be employed by the user:

1. The 1973 horizontal velocity spectra recommended by U.S. Nuclear Regulatory Commission for the seismic design of nuclear power plants. This curve is defined in terms of velocity in in/sec vs. frequency in Hz for various values of damping ratios between 0.005 and 0.10 as shown in [Figure 3.8](#).
2. The 1973 vertical velocity spectra recommended by U.S. Nuclear Regulatory Commission for the seismic design of nuclear power plants. This curve is defined in terms of velocity in in/sec vs. frequency in Hz for various values of damping ratios between 0.005 and 0.10 as shown in [Figure 3.7](#).
3. Elastic ground acceleration response spectrum as defined by EUROCODE8 (EURO8) – Design provisions for earthquake resistance of structures – Part I-1: General rules – Seismic actions and general requirements for structures. This prestored spectrum can be defined by specifying a few parameters as described in Prestored spectra definition cards (see *PRSPECTRA – page 8.6-18).
4. Average acceleration spectra as defined in Indian Standard IS – 1893 (1984). This curve is defined in terms of acceleration in ‘g’ units vs. natural period in seconds for various values of damping ratios 0.0 to 0.2 as shown [Figure 3.9](#).
5. Average acceleration spectra as defined in Indian Standard IS – 1893 (2002). This curve is defined in terms of acceleration in ‘g’ units vs. natural period in seconds for 5% of damping for rocky and other soils sites as shown [Figure 3.10](#).
6. Acceleration response spectrum as defined in AIJ Recommendations for Loads on Buildings by Architectural Institute of Japan. This prestored spectrum can be defined by specifying certain parameters as described in *PRSPECTRA cards.

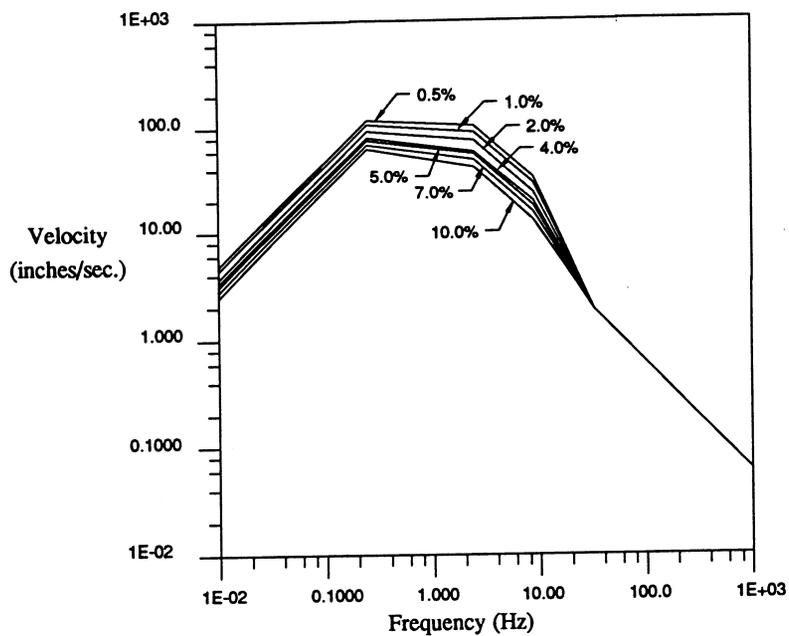


Figure 3.7: U. S. Nuclear Regulatory Commission (NRC) vertical velocity spectra for different damping ratios

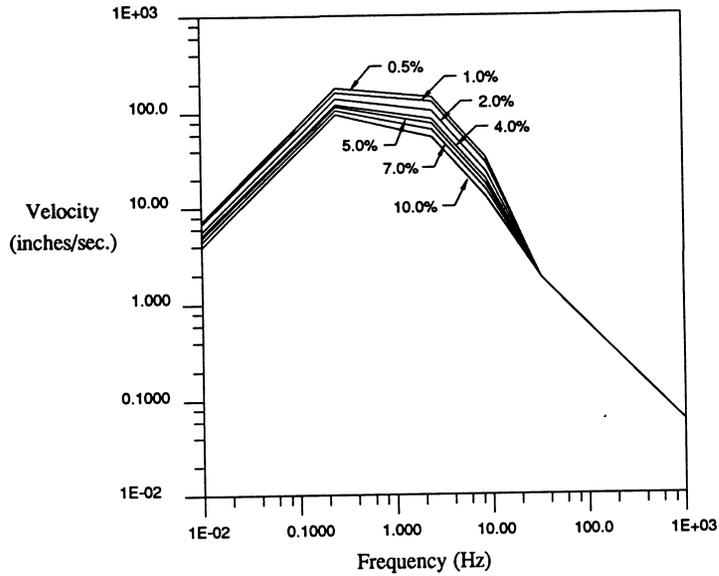


Figure 3.8: U. S. Nuclear Regulatory Commission (NRC) horizontal velocity spectra for different damping ratios

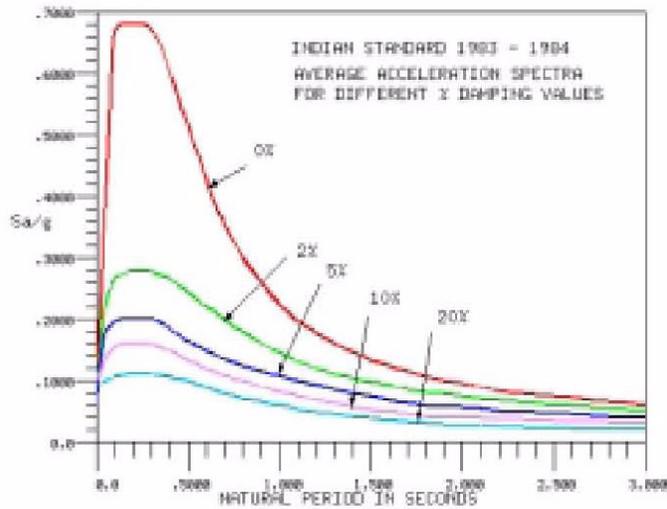


Figure 3.9: Indian Standard 1893 - 1984 average acceleration spectra for different percentage damping values

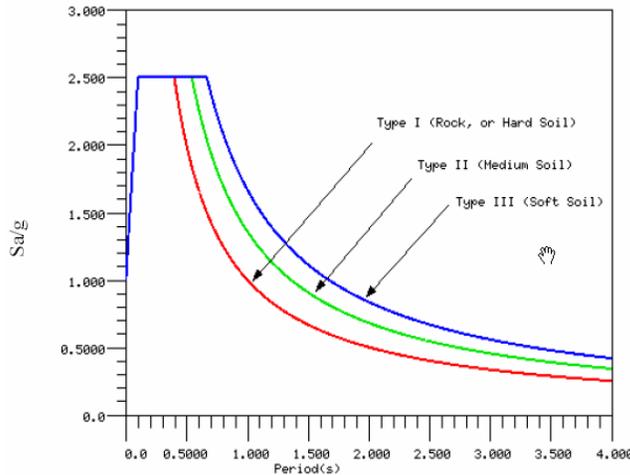


Figure 3.10: Response spectra for rock and other soil sites for 5 percent damping

Combination Rules

The description of the ground motion by means of response spectra, though giving the absolute maximum response, does not give any indication regarding the sign of response or the time at which the maximum response occurs. Thus, an approximation has to be made while summing the response due to shock input in different directions and due to different modes. Several procedures for superposing the directional maxima and modal maxima have been suggested.

Spatial combination across different directions can be performed using one of the following superposition rules:

1. PEAK/ABS (Absolute sum of the directional responses) - This method is very conservative and is usually employed in cases where the ground motion in two directions are statistically correlated. This may be the case when they arise out of the same source as, for example, in the case of seismic excitation.
2. RMS/SRSS (Square root of the sum of the squares of the directional responses) - This method can be used when the ground excitation in the two directions are statistically uncorrelated.

The combination of modal maxima can be done by employing one or more of the following superposition rules:

1. PEAK/ABS (Absolute sum of the modal responses) - This method is conservative and is used if the natural frequencies are closely spaced (within 10 percent of each other) and/or damping is high.
2. RMS/SRSS (Square root of the sum of the squares of the modal responses) - This method is applicable if the modal responses are statistically independent, which is the case when the natural frequencies are far apart and/or damping is low.
3. CQC (Complete quadratic combination) - This method is based on random vibration concepts and employs the modal damping ratios for the summation. It encompasses the SRSS and ABS procedures and usually gives better results than the two.
4. PRMS/NRL (Peak Root Mean Square response) - In this procedure, the absolute maximum of the modal responses is added to the RMS/SRSS of the remaining modal responses. Some design specifications, e.g., Naval structure shock design criteria, require the use of PRMS method.
5. GRP (Grouping Method) - This method is used when modes are closely spaced. The closely spaced modes are divided into groups that include all modes having frequencies lying between lowest frequency in the group and a frequency of 10% higher.
6. TPM (Ten Percent Method) - In this method, pair-wise grouping of modes is adopted. Two modes are closely spaced if the higher frequency falls within 10% of the lower one, otherwise the mode is considered independent.
7. DSM (Double Sum Method) - This method takes into account the modal damping and the duration of the seismic event

CQC rule with missing mass correction

In seismic analysis, CQC rule is recommended for application to structures with closely spaced modes. The rule is derived from fundamental principles of random vibration. In the original CQC combination rule, the average peak response estimate is given by,

$$\langle \max |R_k(t)| \rangle = \left[\sum_{i=1}^N \sum_{j=1}^N \phi_{ki} \phi_{kj} p_i p_j p_{ij} S(\omega_i, \zeta_i) S(\omega_j, \zeta_j) \right]^{1/2} \quad (3.27)$$

p_i and p_j are the modal participation factors in mode i and j . p_{ij} is the cross correlation coefficient between the modal responses for modes i and j . $S(\omega_i, \zeta_i)$ and $S(\omega_j, \zeta_j)$ are the mean displacement response spectrum values for modes i and j .

p_{ij} is given by,

$$p_{ij} = \frac{\lambda_{ij}}{\sqrt{\lambda_{ii}\lambda_{jj}}} \tag{3.28}$$

where, λ_{ij} is the cross-modal spectral moment and is defined by,

$$\lambda_{ij} = \int_0^\infty H_i(\omega)H_j(-\omega)G_{\ddot{u}\ddot{u}}(\omega)d\omega \tag{3.29}$$

From Equation 3.27, it can be observed that in the absence of the cross correlation coefficient p_{ij} the combination rule degenerates to SRSS rule.

In the original CQC rule, a white noise model of the ground excitation is assumed and p_{ij} is found to be,

$$p_{ij} = \frac{8\sqrt{\xi_r\xi_s}(\xi_r + \gamma\xi_s)\gamma^{3/2}}{(1 - \gamma^2) + 4\xi_{-r}\xi_s(1 + \gamma^2) + 4\gamma^2(\xi_r^2 + \xi_s^2)} \tag{3.30}$$

This approximation is valid where the ground motion is a wide band random process with smoothly varying PSD over a frequency range that covers all the significant modes of vibration of the structure. Hence the response spectrum analysis is satisfactory if the earthquake response is computed by model’s superposition of all modes having frequencies less than a cut-off frequency (zero period acceleration frequency). However the effect of missing mass due to the ignored modes – of high frequency – may be significant when these modes have significant contribution or when the input motion has a narrow band-width. For applications of CQC rule in such cases, the following improvements are to be included to account for the quasi-static contribution of the high-frequency modes that are ignored in the modal dynamic analysis.

$$\langle \max |R_k(t)| \rangle = \left[\sum_{i=1}^n \sum_{j=1}^n \phi_{ki} \phi_{kj} p_i p_j \rho_{ij} q_i q_j + ST_k^2 \ddot{u}_{max}^2 + 2 \sum_{i=1}^n \phi_{ki} p_i \rho_{is} q_i ST_k \ddot{u}_{max} \right]^{1/2} \tag{3.31}$$

$ST_k \ddot{u}_{max}$ is the pseudo static response component of the ignored modes. \ddot{u}_{max} is the mean peak ground acceleration. ρ_{is} is the cross correlation coefficient between the i^{th} modal response and the ground excitation and is given by 2 options mentioned below.

Option 1

$$\rho_{is} = \sqrt{\frac{\omega_i^4 \lambda_{ii}}{\langle \ddot{u}^2 \rangle} \left(1 - \frac{\lambda_{2,ii}}{\omega_i^2 \lambda_{ii}} \right)} \quad (3.32)$$

$\langle \ddot{u}^2 \rangle$ is the mean square value of the ground acceleration. $\lambda_{2,ii}$ is the cross modal spectral moment and is given by,

$$\lambda_{2,ii} = \int_0^\infty \omega^2 H_i(\omega) H_i(-\omega) G_{\ddot{u}\ddot{u}}(\omega) d\omega \quad (3.33)$$

Option 2

If the cross-correlation coefficients ρ_{js} are not available, the following data is to be supplied by user. This data facilitates computation of ρ_{js}

- Mean peak ground acceleration ($\langle \ddot{u}^2 \rangle$ in [Equation 3.6](#))
- Input ground PSD (stationary and one -sided) ($G_{\ddot{u}\ddot{u}}(\omega)$ in equation (3.7))

If Kanai-Tajimi PSD - filter parameters ζ_f and ω_f

For other types - PSD in Tabular form

Computational needs in the case of option 2.

To find the Cross Correlation coefficients ρ_{js} , using equation (3.6), it is required to compute λ_{ii} , $\lambda_{2,ii}$ and mean ground acceleration $\langle \ddot{u}^2 \rangle$. These quantities are given by,

$$\lambda_{ii} = \int_0^\infty H_i(\omega) H_i(-\omega) G_{\ddot{u}\ddot{u}}(\omega) d\omega$$

$$\lambda_{2,ii} = \int_0^\infty \omega^2 H_i(\omega) H_i(-\omega) G_{\ddot{u}\ddot{u}}(\omega) d\omega$$

$$\langle \ddot{u}^2 \rangle = \int_0^{\infty} G_{\ddot{u}\ddot{u}}(\omega) d\omega$$

It is to be noted that the extreme case of ρ_{js} equal to zero is valid in case the input ground excitation is wide banded and the modal damping is low. On the other hand, in the case of a narrowband input and modal frequencies much higher than the input frequencies, ρ_{js} can be assumed to be unity.

3.5.8 Linear Direct Transient Analysis (LTRANSIENT)

Linear direct transient analysis may be used to analyze the linear behavior of structures under dynamic loads. In general, the applied loads may include prescribed forces and pressures. Other loads such as gravity, Coriolis force, and centrifugal force can also be included. The kinematic conditions may include specified displacements, coupled displacements, multipoint constraints and rigid link specifications, but the structure need not be constrained. Rigid body motions as well as the elastic responses can be handled.

The capabilities and special features available in linear direct transient analysis are as follows:

Input

1. Combination of any number of time dependent concentrated nodal forces (*CFORCE) and body forces (*TBODYFORCE) can be specified.
2. The variation of loads with respect to time can be expressed by means of time amplitude curves (*TIMEAMP).
3. Different time increments can be specified for different events (*EVENT).
4. Non-zero initial conditions (displacements and/or velocities and/or accelerations) can be specified (*INITIAL) at the nodes of the structure.

Output

1. Selective printout and/or plot of time histories of nodal displacements, velocities, accelerations, stresses and reaction forces over a user specified time range.
2. Printout and/or plot of the above responses for selected steps can be specified.

Selection of Time Integration Parameters

The time step size for direct time integration is a user defined analysis parameter. The criterion for selecting the time step size is based on accuracy and stability of the integration algorithm. In Newmark family, two parameters β and γ determine the stability and accuracy characteristics of the algorithm. Depending on β and γ , the algorithm can be unconditionally stable or conditionally stable.

For unconditional stability, we should have,

$$2\beta \geq \gamma \geq \frac{1}{2} \quad (3.34)$$

For conditional stability, we have,

$$\gamma \geq \frac{1}{2} \quad (3.35)$$

$$\omega \Delta t \leq \Omega_{crit} \text{ (Stability condition),} \quad (3.36)$$

$$\beta < \frac{\gamma}{2} \quad (3.37)$$

where, Ω_{crit} is the critical sampling frequency

$$\Omega_{crit} = \frac{\xi \left(\gamma - \frac{1}{2} \right) + \left[\frac{\gamma}{2} - \beta + \xi^2 \left(\gamma - \frac{1}{2} \right)^2 \right]^{\frac{1}{2}}}{\left(\frac{\gamma}{2} - \beta \right)} \quad (3.38)$$

The stability condition [Equation \(3.36\)](#) must be satisfied for each mode in the system. Since the effect of viscous damping is to increase the critical time step of conditionally stable Newmark methods, time step size estimated from the undamped case can serve as a conservative criterion,

$$\Omega_{crit} = \left(\frac{\gamma}{2} - \beta\right)^{\frac{1}{2}} \quad (3.39)$$

The default values for β and γ in the NISA program, are 0.3025 and 0.6, respectively. These values will give an unconditionally stable algorithm with some numerical damping. It is desirable to have some form of numerical damping to remove the participation of high-frequency modal components which may be artificially introduced by discretization of the structure and may not be part of the actual physical responses. In terms of the Newmark method, $\gamma > 1/2$ is necessary to introduce high-frequency dissipation. For a fixed γ , the ideal value for β is given by,

$$\beta = \frac{\left(\gamma + \frac{1}{2}\right)^2}{4} \quad (3.40)$$

Time Amplitudes for Loads

The time dependent loads applied on a structure are described by time amplitude curves given in a piecewise linear fashion (*TIMEAMP). The loading may be a combination of concentrated nodal forces and/or body forces described by their respective load magnitudes and time amplitude curves which indicate their variation with time. Hence, more than one input loading may be referenced to the same time amplitude curve. There is no restriction for the number of loads that can be specified, but the number of time amplitude curves is limited to 128.

Coriolis Force

Coriolis force as well as other body forces such as translational acceleration and centrifugal forces are handled in the *TBODYFORCE data group. The output responses from the analysis are relative values. In NISA, the presence of Coriolis force will direct the solution process to go through several iterations automatically for each time step. This is required because the Coriolis force is treated as an additional load component.

Variation of Time Step Size

In linear direct transient analysis, the time step size can be changed from event to event, but it is a fixed value within each event. By maintaining a constant time step size, the formation

and factorization of a new effective stiffness matrix can be avoided. From the point of view of cost effectiveness, this saves substantial computational time.

Initial Accelerations

By default, the initial acceleration state of the system is assumed to be zero by the program. Theoretically, the initial acceleration \mathbf{a}_0 should be obtained by solution of the following equation,

$$\mathbf{M}\mathbf{a}_0 = \mathbf{p}_0 - \mathbf{K}\mathbf{d}_0 - \mathbf{C}\mathbf{v}_0 \quad (3.41)$$

where \mathbf{M} , \mathbf{C} and \mathbf{K} are the global mass, damping and stiffness matrices, \mathbf{d}_0 , \mathbf{v}_0 and \mathbf{p}_0 are the initial displacement, velocity and external load vector prescribed by the user. In practice, however, the mass matrix \mathbf{M} is often singular, and the evaluation of the acceleration at $t = 0$ is not possible.

To overcome this difficulty and also to avoid the “ringing” of massless degrees of freedom subjected to step loads, the initial acceleration is taken as zero by default. This is equivalent to *resetting* the initial external load vector prescribed by the user to,

$$\mathbf{p}_0 = \mathbf{K}\mathbf{d}_0 + \mathbf{C}\mathbf{v}_0 \quad (3.42)$$

However, NISA allows the user the flexibility of being able to input the initial acceleration state, \mathbf{a}_0 , if desired. This is achieved by specifying the initial acceleration values of various nodes in the *INITIAL data group. In such a case, the external load vector at $t = 0$, namely, \mathbf{p}_0 is computed consistent with the initial state, \mathbf{d}_0 , \mathbf{v}_0 and \mathbf{a}_0 , as,

$$\mathbf{p}_0 = \mathbf{K}\mathbf{d}_0 + \mathbf{C}\mathbf{v}_0 + \mathbf{M}\mathbf{a}_0 \quad (3.43)$$

For analysis with two or more events, the transition from one event to another is carried out such that the solution is continuous in displacement, velocity and acceleration at the instant of transition. This may involve resetting the external load vector for the second event at the time of transition in a manner similar to [Equation \(3.43\)](#).

For Restart 5 runs, the initial state (including acceleration) is read from File 37, i.e., the same as the last time step of the previous run. Any or all initial condition values specified by the user in the *INITIAL data group are ignored.

3.5.9 Direct Frequency Response Analysis (DFREQ)

Direct frequency response analysis may be used to analyze the steady state response of the linear structures under frequency dependent dynamic loads. In general, the applied loads may include prescribed forces and pressures. The kinematic conditions may include specified displacements, coupled displacements, multipoint constraints and rigid link specifications, but the structure has to be constrained to arrest rigid body motions.

The capabilities and special features available in direct frequency response analysis are as follows:

Input

1. Combination of any number of time dependent concentrated nodal forces (*CFORCE) and pressures (*PRESSURE) can be specified.
2. The variation of loads with respect to frequency can be expressed by means of frequency functions (*FREQFUNCTION).
3. Different frequency increments can be specified for different frequency controls (*FREQCNTL).
4. The same frequency function (*FREQFUNCTION) card can be used to specify a frequency dependent support excitation.
5. Frequency dependent stiffness and damping characteristics for spring and damper elements respectively can be specified by a frequency curve. Currently NKTP=21, NKTP=17, NKTP=48 have this feature enabled.

Output

1. Selective printout and/or plot of frequency functions of nodal displacements, velocities, accelerations, stresses and reaction forces over a user specified frequency range.
2. Printout and/or plot of the above responses for selected frequency points can be specified.

Specification of Loads in Frequency Domain

The frequency dependent loads applied on a structure are described by frequency functions given in a piecewise linear fashion (*FREQFUNCTION). The loading may be a combination of concentrated nodal forces and /or pressures described by their respective frequency functions which indicate their variation with frequency. Hence, more than one input loading may be referenced to the same frequency function. There is no restriction for

the number of loads that can be specified, but the number of frequency function curves is limited to 128.

Variation of Frequency Step Size

In direct frequency response analysis, the frequency step size can be changed from one frequency control to another, but it is a fixed value within each frequency control.

3.5.10 Nonlinear Direct Transient Analysis (NLTRANSIENT)

Nonlinear direct transient analysis can be used to examine the nonlinear behavior of structures under dynamic loads. The applied loads may include prescribed forces and pressure. Other loads such as gravity, Coriolis force and centrifugal force can also be included. The nonlinearities considered in nonlinear direct analysis may be material, geometric or combined (material and geometric). In solving the nonlinear equilibrium equation, the program employs an incremental-iterative scheme. Both conventional and modified Newton-Raphson methods are available. As in nonlinear static analysis, three convergence criteria are available in the program which are: displacement, force and energy. Tolerances for each criteria may be specified independently on the *EVENT data group. If all tolerances are specified, a step will be assumed converged when any one criterion is satisfied. Similar to modal dynamic analyses and linear direct transient analysis, both consistent and lumped mass formulation can be used. Viscous damping can be specified in terms of dashpot elements or proportional damping coefficients. Although the program was not primarily developed to solve kinematic problems, it can handle rigid body motions.

As in nonlinear static analysis, two equivalent referential formulations are available, namely, the total and the updated Lagrangian formulations.

The load stepping feature is the same as in nonlinear static/linear transient analyses. The entire load history may be divided into events, an event generally being one where new load types are introduced. Step sizes may be determined automatically by the program or via user definitions. In each event, the user has the option of specifying time increment sizes, equilibrium checks, tolerances, iterative procedure to be used, and other control parameters through the use of the *EVENT data group.

For a description of selection of time interpretation parameters, time amplitudes for loads (*TIMEAMP), Coriolis force, and initial accelerations, please see [Section 3.5.8](#).

In nonlinear direct transient analysis, time step size can not only be changed from event to event, but also within an event using either INCREMENTS = AUTO, n or INCREMENTS = USER, n or INCREMENTS = ADAPTIVE, n in *EVENT data.

The adaptive time stepping scheme (INCREMENTS = ADAPTIVE, n) is based on the control of the estimated relative error between explicit Euler and Newmark methods as described below.

Original Nonlinear ODE:

$$M\ddot{\mathbf{u}} + C\dot{\mathbf{u}} + K\mathbf{u} + Q(\mathbf{u}, \dot{\mathbf{u}}, t) = F(t) \quad (3.44)$$

Where $Q(\mathbf{u}, \dot{\mathbf{u}}, t)$ is the nonlinear part of the vector field.

Knowing the response at the i^{th} instant t_i , response at t_{i+1} may be computed by the explicit Euler method as,

$$\mathbf{u}_{i+1}^e = \mathbf{u}_i^e + h_i \dot{\mathbf{u}}_i^e, \quad \dot{\mathbf{u}}_{i+1}^e = \dot{\mathbf{u}}_i^e + h_i \ddot{\mathbf{u}}_i^e \quad \text{and} \quad \ddot{\mathbf{u}}_{i+1}^e = V(\mathbf{u}_{i+1}^e, \dot{\mathbf{u}}_{i+1}^e, t_{i+1}) \quad (3.45)$$

Similarly the response at t_{i+1} via the explicit Newmark method is computable as,

$$\mathbf{u}_{i+1}^n = \mathbf{u}_i^n + h_i \dot{\mathbf{u}}_i^n + \frac{h_i^2}{2} \ddot{\mathbf{u}}_i^n, \quad \dot{\mathbf{u}}_{i+1}^n = \dot{\mathbf{u}}_i^n + h_i \ddot{\mathbf{u}}_i^n \quad (3.46)$$

$$\ddot{\mathbf{u}}_{i+1}^n = V(\mathbf{u}_{i+1}^n, \dot{\mathbf{u}}_{i+1}^n, t_{i+1})$$

where h_i is the required time step size.

In equations above, the superscripts ‘e’ and ‘n’ correspond to solutions from explicit Euler and Newmark methods. At each $i+1^{\text{th}}$ step, the relative error between Euler and Newmark explicit solutions is now expressed as,

$$E_{i+1}^l = \left\| \mathbf{u}_{i+1}^n - \mathbf{u}_{i+1}^e \right\| + h_i \left\| \dot{\mathbf{u}}_{i+1}^n - \dot{\mathbf{u}}_{i+1}^e \right\| \quad (3.47)$$

Integrating Equation (3.45) and Equation (3.46) with identical initial conditions, the local error can be estimated to be equal to.

$$E_{i+1}^l(\text{ERRTOL}) \leq \frac{1}{2} \|\ddot{\mathbf{u}}_i\| h_i^2 \quad (3.48)$$

Thus the scheme essentially requires no solution from either Euler or Newmark explicit method. Equation (3.48) is utilized to obtain the adaptive step size, h_i with $\ddot{\mathbf{u}}$ obtained from Newmark- β implicit scheme at each time step. Thus, for an error tolerance (ERRTOL) specified by user, the step size may be obtained as,

$$h_i = \sqrt{\frac{2 \times \text{ERRTOL}}{\|\ddot{\mathbf{u}}_i\|}} \quad (3.49)$$

The input/output features in nonlinear direct transient analysis are as follows:

Input

1. Combination of any number of time dependent concentrated nodal forces, follower forces, pressure loads, follower pressure loads and body forces can be specified.
2. Nonlinear material properties like elastic-plastic behavior can be specified.
3. The variation of loads with respect to time can be expressed by means of time amplitude curves.
4. Different time increments can be specified for different events (*EVENT).
5. Non-zero initial conditions (displacements and/or velocities) can be specified.

Output

1. Selective printout and/or plot of time histories of nodal displacements, velocities, accelerations, stresses and reaction forces over a user specified time range.
2. Printout and/or plot of the above responses for selected steps.

3.6 Buckling Analysis (BUCKLING)

BUCKLING analysis can be used to find critical loads at which a structure becomes elastically unstable. This analysis is properly called ‘bifurcation buckling’ or ‘initial stability’ [3.9] since it involves calculating the points at which the structure’s primary load deflection path, a straight line for a linear static analysis, is bifurcated by a secondary load deflection path. Determination of these bifurcation points requires eigenvalue extraction, since the governing matrix equation for buckling analysis is a general eigenvalue problem of the form,

$$\mathbf{K}\mathbf{u} = -\lambda\mathbf{K}_G\mathbf{u} \quad (3.50)$$

where \mathbf{K} is the linear stiffness matrix, \mathbf{K}_G is the geometric stiffness (or stress stiffening) matrix, λ is the eigenvalue (load factor), and \mathbf{u} is the buckling mode shape. The geometric stiffness matrix for a typical element is a function of the state of stress in the element. A null geometric stiffness matrix is assumed for elements which do not have stress stiffening capability (e.g., point masses and springs).

Buckling analysis in NISA is a two-pass analysis made in one run for a normal run without restart (refer to [Section 3.16](#) for buckling restart). The first pass is a static analysis, which determines the stresses for a given reference set of loads. The second pass is an eigenvalue analysis, which first computes the geometric stiffness matrix (based on the stresses found in the static pass), and then solves for the load factors (eigenvalues) and mode shapes (eigenvectors). The procedure to run buckling analysis is detailed in [Chapter 5](#).

The buckling load factors are multipliers for the reference loads defined in the static pass. For example, if a reference load of 1 lb is applied to the structure, and a buckling load factor of 1600 is obtained, the structure will become elastically unstable at a load of 1600 lbs. If, for the same model, a reference load of 800 lbs is applied, a buckling load factor of 2 will be obtained, resulting in the same critical load of 1600 lbs. Thus, the magnitude of a reference load is not important. However, if more than one load acts on the structure, it is necessary that their relative magnitudes be correct.

The eigenvalue extraction methods for buckling analysis are the same as those available for vibration eigenvalue analysis. Refer to [Section 3.5.2](#) for a discussion on the choice of method. The inverse iteration method or either of the subspace algorithms can be employed to extract the load factors. Lanczos method is not recommended for buckling analysis. The load factors obtained from buckling analysis may be negative, indicating that buckling loads

are obtained when the reference loads are reversed in direction and scaled up by the load factors. The available eigenvalue extraction methods can handle negative eigenvalues (load factors). Negative load factors may be obtained in the inverse iteration method by adopting suitable lower and upper cut-off limits. However, for the subspace iteration methods, it is recommended that a negative initial shift smaller than the lowest eigenvalue (largest negative eigenvalue) be specified in cases where negative load factors are suspected to be present. The buckling analysis also has Sturm sequence check capability, in which all the load factors below a user specified upper limit may be calculated or reported in number.

The output from buckling analysis consists of:

1. For the static pass: All output options available for static analysis, refer to [Section 3.14](#) for details.
2. For the eigenvalue pass:
 - (a) Buckling load factors and the computational tolerances on load factors.
 - (b) Selective printout of eigenvectors (buckling mode shapes).

NISA files 26 and 27 should be saved if post-processing of the results using the DISPLAY program is desired. The plots include mode shapes (displacements) as well as their contours. On certain graphic devices, these mode shapes may be animated.

It should be noted that this type of bifurcation buckling or initial stability analysis does not account for initial imperfections, provides no information about post-buckling behavior and is limited to cases where the initial displacement or large deformation stiffness matrix is negligible. Such cases are very limited in practical application. To be able to neglect the effect of large displacements, bending stresses have to be small compared to membrane stresses and the loading should be non-follower, i.e. does not follow the deformed or buckled geometry.

In order to do a proper post-buckling behavior (e.g., snap-through analysis), a full incremental nonlinear static analysis should be performed taking into account both the initial stress and initial displacement stiffness matrices, and applying the load incrementally. If at any load step, the tangential stiffness matrix becomes singular, static instability is predicted.

3.7 Stress Stiffening

Stress stiffening effect means that the stiffness characteristics of the structure is dependent on the state of stress in the structure, in addition to the material and geometric properties of the model.

Stress stiffening is automatically included in the geometric nonlinear analysis capability (i.e., ANALYSIS = NLSTATIC and NLTYPE = GEOM, see (Section 5.3)). In the geometric nonlinear static analysis, both stress stiffening and large displacement effects are included. In many practical applications, e.g., rotating structures, the stress stiffening effect is more pronounced than the large displacement effect. However, in NISA the two effects are taken into account together in the stiffness formulation. The governing equilibrium equations (Equation (3.2)), repeated here for convenience, are solved incrementally in the form,

$$(\mathbf{K}^{(i)} + \mathbf{K}_G^{(i)})\Delta\mathbf{u}^{(i)} = {}^{t+\Delta t}\mathbf{p} - {}^{t+\Delta t}\mathbf{f}^{(i-1)} \quad (3.51)$$

where,

- $\mathbf{K}^{(i)}$: is the stiffness matrix due to linear and initial displacement effect (and the spin softening effect, see Section 2.3.4)
- $\mathbf{K}_G^{(i)}$: is the stress stiffening (or geometric stiffness) matrix which depends upon the current state of stress
- $\mathbf{u}^{(i)}$: is the iterative increment of nodal displacements
- ${}^{t+\Delta t}\mathbf{p}$: is the vector of externally applied nodal forces
- ${}^{t+\Delta t}\mathbf{f}^{(i-1)}$: is the vector of internal nodal forces

The stress stiffening effect is important for thin flexible structures subjected to loadings causing high tensile or compressive stresses. The combined stiffness of the structure increases (or decreases) as the membrane stresses increases (or decreases). This directly affects the capability of the structure to carry lateral loads, e.g., if a strut is initially tensioned, it will be stiffer in the lateral direction. If the same strut is initially compressed, it will be softer in the lateral direction and in the limiting case the combined stiffness may become singular indicating static instability. Stress stiffening effect may be important also in solid structures with high normal stresses, e.g., turbine blades under centrifugal loading, and in beam structures where the transverse deflections (or vibrations) are affected by the presence of normal force.

It should be noted that for line elements with pre-stress capability (initial tension in 3-D spar and cable elements, NKTP = 14, 45), the stress stiffening effect is taken into account in the first iteration.

3.8 Heat Transfer Analysis

3.8.1 Introduction and Analysis Types

Modes of Heat Transfer

Heat transfer analysis is concerned with the flow or transfer of heat energy as a result of temperature gradient. This phenomenon takes place by three different processes or modes: conduction, convection and radiation. In conduction heat flow, the energy is transferred by direct molecular communication without noticeable displacement of the molecules. This is the predominant mode in solids, but it also occurs in liquids and gases. In convection heat flow, the energy is transmitted primarily through the movements of the molecules. Convection occurs only in fluids, i.e., liquids and gases. In the case of no phase change, convection may be classified as free and forced convection. In free convection, the displacement of the molecules is owing to the effect of thermal gradient on the fluid density. In forced convection, the displacement of the molecules will be due to mechanical or forced agency. The third mode of heat flow is radiation, in which the heat energy is transmitted by electromagnetic waves which can pass through vacuum as well as gases, liquids and some solids.

In the NISA program, problems involving heat transfer due to conduction with convection and radiation boundary conditions can be analyzed. Solidification and melting processes may be analyzed also under the assumptions of no volume change and no convection effect in the analysis, i.e., the energy balance equation does not include the parabolic convective term. Latent heat variation due to change of phase can be included in the analysis. Analogous field problems may also be handled, see [Section 3.8.4](#). In the following, a brief description of various analysis types available in the program are given.

Linear and Nonlinear Steady State Heat Transfer Analysis (SHEAT)

Heat transfer analysis is assumed to be a steady state one, if the material properties and boundary conditions are not time dependent [3.10-3.12]. In the program both linear and nonlinear steady state problems may be analyzed. The problem is considered as linear steady state heat transfer if the following conditions are met:

- No radiation boundary conditions
- No phase change

- ❑ Material properties, convective coefficients, specified heat fluxes and specified internal heat generations are temperature independent.

If any of the above conditions are not satisfied, the problem will be considered as nonlinear steady state heat transfer. In nonlinear steady state heat transfer, an incremental-iterative solution is employed. The user has the option of recalculating the conductivity matrix each iteration (full Newton-Raphson method) or only at the beginning of the increment (modified Newton-Raphson). The load increments may be equal or user defined. Heat flux tolerance may be specified by the user to control the accuracy of the converged solution.

Linear and Nonlinear Transient Heat Transfer Analysis (THEAT)

The problem will be considered a transient one if the material properties and/or boundary conditions are time dependent [3.10-3.12]. In this case, the rate of heat absorption (i.e., the material heat capacity times the temperature time derivative) has to be included in the energy balance equation). Both linear and nonlinear transient analysis may be performed in the program. As in the steady state case, linear transient analysis will be considered if the following conditions are met:

- ❑ No radiation boundary conditions
- ❑ No phase change
- ❑ Material properties, convective coefficients, specified heat fluxes, and specified internal heat generations are temperature independent.

If any of the above conditions are not satisfied, the problem will be considered a nonlinear transient analysis. In such case, an incremental- iterative solution is employed. The user has the option of recalculating the conductivity and the capacity matrices each iteration (full Newton-Raphson method) or only at the beginning of each time increment (modified Newton-Raphson method). The time increments (step sizes) may be equal or user defined with specified bounds on the minimum or the maximum step size. Four common integration schemes are available in the program, these are:

- ❑ The forward difference (explicit Euler) scheme
- ❑ The trapezoidal (Crank-Nicholson) scheme
- ❑ The Galerkin scheme
- ❑ The backward difference (implicit Euler) scheme

All of the above methods except the forward difference one are unconditionally stable. This means that the scheme will be stable irrespective of the value of the time step (no upper bound critical time step size). The controlling factors for choosing the step size, in this case, are the accuracy and the cost. Smaller time steps will give more accurate results but with higher cost. On the other hand, the forward difference (explicit Euler) scheme is an explicit method which requires the time step to be smaller than a critical value for stability considerations, i.e., it is a conditionally stable method. For details of the critical time step, see [Section 2.6](#). It should be noted that all unconditionally stable methods do require triangularization of the conductivity matrix whereas the conditionally stable methods do not, in general, require the factorization process. The use of conditionally stable methods has been limited, however, in practical problems.

Basic Heat Transfer Output:

The basic output from a typical heat transfer run is briefly described here. Table form description of output options and the data groups requesting them as well as postprocessing features are given in [Section 3.14](#). The following is a list of the basic output for heat transfer analysis:

- Nodal temperatures (and nodal temperature differences when applicable)
- Nodal point heat flow for nodes with specified temperatures
- Heat flow on element boundary faces with convection and radiation boundary conditions.

3.8.2 Heat Transfer Material Models

Isotropic Material Model

In this model, all thermal or heat transfer properties are assumed to be directional independent, i.e., they do not change with any orthogonal transformation of axes. In this case only *one* thermal conductivity value is required, along with other applicable thermal property values, such as specific heat, density, etc.

The material properties may be defined as constants (i.e., do not depend on temperature or time) or may be temperature dependent. Temperature dependent material properties may be expressed as polynomial function of temperature (up to fourth order), or may be defined through a temperature dependency input curve.

Orthotropic Material Model

In this model, *three* mutually perpendicular planes of symmetry exist at each point, and the material has different properties in three mutually perpendicular directions at any point. In general, *three* thermal conductivity values are required to define this model, along with other applicable thermal property values, such as specific heat, density, etc. The orthotropic material axes may coincide with the global axes or may be defined with respect to nodes (see *MATDIR1 data group), or with respect to elements (see *MATDIR2 data group).

The material properties may be defined as constants (i.e., do not depend on temperature or time) or may be temperature dependent. Temperature dependent material properties may be expressed as polynomial function of temperature (up to fourth order), or may be defined through a temperature dependency input curve.

3.8.3 Boundary Conditions and Loading

(i) Specified Heat Fluxes (*CFLUX and *DFLUX)

Various types of specified fluxes may be defined in the program. These include: distributed flux, concentrated flux, and zero or no heat flux (insulation) conditions. Distributed heat flux (*DFLUX data group) may be specified on an element face and it can be uniform or non-uniform on that face. Concentrated heat fluxes are input directly at the nodes as nodal values (see *CFLUX data group). If no specified flux value is given on a surface, that surface will have a null flow of flux (into or out of the surface), i.e., the surface will be acting as an insulation boundary.

All specified flux values may be temperature and/or time dependent. Flux values may reference a time-amplitude curve and/or a temperature-scaling factor curve that will define the dependency of the flux value on either parameter. If flux values are temperature dependent, the problem will be nonlinear, whereas if they are time dependent, the problem will be transient.

(ii) Convection Boundary Conditions (*CONVBC)

Pure convection analysis, i.e., taking into account the convective transport term in the field equation, is not available yet in the program. However, in many practical applications, the convection may be handled as a boundary condition on the solid-fluid interface region. NISA provides this capability by allowing the user to specify a convection heat transfer

coefficient between the two surfaces (solid and fluid) and a reference ambient temperature. NISA also provides convection link elements (see [Section 4.46](#)) which can be used for the transport of heat due to convection.

For a given element surface, the convective coefficient may be uniform, non-uniform, temperature dependent, and/or time dependent. This provides flexibility to the user in modeling various types of convective boundary conditions, e.g., due to free convection, forced convection, transient, and nonlinear effects. In specifying the convective coefficients, time-amplitude curve and/or temperature-scaling factor curve may be referenced to give the dependency of the coefficient on time and/or temperature, respectively. If the convective coefficient is temperature-dependent, nonlinear analysis is assumed, whereas if it is time-dependent, transient analysis will be assumed.

(iii) Radiation Boundary Conditions (*RADBC)

Unlike conduction and convection, heat transfer by thermal radiation does not require the existence of any form of media or matter. The electromagnetic radiation waves can pass through vacuum, fluids, as well as some solids. In general, the equation governing the heat exchange by radiation between two surfaces may be written as:

$$Q = \sigma F_a F_\epsilon (\theta_1^4 - \theta_2^4) \quad \text{energy/time} \quad (3.52)$$

where, σ is the Stefan-Boltzmann constant, F_a is a geometric view factor based on the configuration of the two radiating surfaces, F_ϵ is an effective emissivity factor (for black bodies, $F_\epsilon = 1.0$), and θ_1, θ_2 are the absolute temperatures of the radiating surfaces.

NISA enables the user to handle radiation energy incident on (or emitted from) a surface as a boundary condition specified on that surface. The user specifies the Stefan-Boltzmann constant, σ , and the product of the view factor, F_a , and the emissivity factor, F_ϵ . This product is lumped into one factor for ease of input and given the name of emissivity (refer to [Equation \(3.52\)](#)). The second body temperature, θ_2 , is specified as the ambient temperature.

The lumped emissivity factor ($F_a * F_\epsilon$) may be temperature and/or time dependent, whereas the ambient temperature may be time dependent. It should be noted that radiation is a nonlinear boundary condition causing an iterative solution to be employed. NISA also provides radiation link elements (see [Section 4.47](#)) which can be used for the transport of heat due to radiation.

(iv) Surface Radiation Heat Transfer (*RADSURFACE)

One of the major problems in incorporating surface radiation in a heat transfer analysis is the knowledge of view factors. NISA/HEAT internally calculates the view factors for user specified radiation surfaces taking into account possible obstructions between pairs of surfaces. Emissivity can be specified as a function of temperature. Temperature of surroundings can be specified as a function of time for transient analysis.

(v) Specified Heat Generation Rates (*ELHEATGEN, *NDHEATGEN)

Heat generation values may be specified for elements in the structure to simulate a heat source (or sink). Heat generation values should be given in units of energy/time.volume, and may be specified for elements or nodes. If elemental heat generation values are given (*ELHEATGEN), it will be integrated over the volume of the element, whereas if nodal heat generation values are given (*NDHEATGEN), it will be integrated over the volume of all elements connected to the node. Heat generation values may be temperature and/or time dependent, by referencing a temperature-scaling factor curve, or a time-amplitude curve, respectively.

(vi) Temperature Conditions

*Prescribed Temperatures (*SPTEMP)*

User specified or prescribed temperature values will remain constant throughout the analysis. If transient analysis is considered, the prescribed temperature value may be made time dependent by referencing an appropriate time-amplitude curve. In such a case, the value of the prescribed temperature is determined from the time-amplitude curve at the beginning of each time step and will be kept constant during that step.

*Coupled Temperatures (*CPTEMP)*

Coupling of temperature between two or more nodes is a simple type of linear homogenous constraint indicating that the temperatures of all referenced nodes are the same. Nodes which have coupled temperatures may not appear in a prescribed temperature condition.

*Initial Temperature (*INITEMP)*

Initial temperature values may be specified for the overall structure or parts of it at the beginning of a transient analysis. Zero initial temperature will be assumed for parts which do not have specified initial temperature.

3.8.4 Analogous Field Problems

The quasi-harmonic equation used to solve heat conduction problems may be applied to a wide range of physical applications. As an example, we mention the following few engineering applications:

- Heat conduction
- Electrostatic field problems and distribution of electric or magnetic potential
- Ground water flow and seepage through porous material
- Torsion of prismatic bars
- Transverse deflection of elastic membranes
- Bending of prismatic beams
- Lubrication of pad bearings
- Irrotational flow of an ideal fluid
- Electric conduction

The formulations and finite element discretizations of the above problems are the same as those for heat conduction. The difference will be in the designation of the primary field variable, material constants, and inputs. To aid the user in solving these applications, we list some of the common field problems and their analogous variables and constants in [Table 3.2](#).

Table 3.2: Examples of analogies in analysis of field problems

Field Problem	Primary Variable	Constants	Input	
Heat conduction	Temperature	Thermal conductivities	Heat flux	Internal heat generation
Electrostatic and magnetostatics	Field potential	Permeability	Flux density	Charge density
Ground water flow and seepage	Head or piezometric head	Permittivity or permeability	Prescribed flow conditions	Recharge, pumping or internal flow generation
Torsion	Stress function	Inverse of the shear modulus	Prescribed shear stresses	Twice the angle of twist per unit length
Electric conduction	Voltage	Electric conductivity	Prescribed current	Internal current source
Irrotational flow of ideal fluids	Potential or stream function	Density	Prescribed velocity	Mass production (usually zero)

3.9 Material Models

This section describes only structural material models. For heat transfer material models, refer to [Section 3.8](#).

3.9.1 Linear Elastic Material Model (*MATERIAL)

In this model the components of the material constitutive relations (stress-strain relations) are assumed to be constants, i.e., independent of the stress and strain states or their histories. This ideal elastic solid (Hookean material model) is commonly assumed for the design and analysis of many engineering applications. In a general case, the model requires 21 independent elastic constants to characterize the constitutive relation (assuming symmetry of the stress and strain tensors and the existence of a strain energy function). Two specific cases are available in the program, these are: the isotropic and the orthotropic linear elastic models.

In the isotropic linear elastic model, the elastic constants are directionally independent, i.e., they do not change with any orthogonal transformation of axes. In this case, only two independent constants are required to define the constitutive relations, these are: Young's modulus, EX, and Poisson's ratio, NUXY. In the orthotropic linear elastic model, three mutually perpendicular planes of elastic symmetry exist at each point. This is evident for example in rolled or extruded plates. In this model, generally nine independent constants are required to define the constitutive relations. These are: Young's modulus in three directions EX, EY, EZ, Poisson's ratio in three planes NUXY, NUYZ, NUXZ, and the moduli of rigidities GXY, GYZ, and GXZ. Depending on the type of elements used in the model some of these constants may not be needed. The orthotropic material axes may coincide with the global axes or may be defined with respect to the nodes (see *MATDIR1 data group), or with respect to elements (see *MATDIR2 data group).

All the linear elastic constants may be temperature dependent. The user may express a particular property as a polynomial function of temperature with the polynomial order of up to the fourth order. If thermal (temperature) effects are included, other material properties will be needed, namely coefficients of thermal expansion. Other material properties are needed for specific applications or program options, e.g., material density is needed for body force loading and dynamic analysis.

Linear elastic isotropic or orthotropic models may be used in geometrical nonlinear analysis. In this case, the material constants, as defined by the user, should be with respect to the reference undeformed configuration. In the total Lagrangian formulation, the constitutive relation will then relate the second Piola-Kirchhoff stress tensor to the Green-Lagrange strain tensor. The program will internally perform the appropriate fourth order tensor transformation, for the constitutive tensor, which is required for updated Lagrangian formulation.

It is possible to classify materials in terms of positional rather than directional characteristics, e.g., homogenous, heterogeneous, etc. This is evident for example in composites where the material properties depend on the position. NISA has a layered composite material model that may be used with composite elements such as the 3-D composite solid (NKTP = 7), and the 3-D composite shell element (NKTP = 32).

In this model, the elements are assumed to consist of a number of layers of perfectly bonded orthotropic material. The material principal directions may be different at each layer and can be specified independently. Various options for defining the orthotropic material axes for each composite layer are detailed in the lamination sequence data group, *LAMSQ2. For linear analysis, three failure theories are available for the 3-D composite general shell element, these are: the maximum stress, the modified Hill-Mises, and the Tsai-Wu failure theory. For the 3-D composite solid element, the maximum stress and the Tsai-Wu failure criteria are available. For nonlinear analysis, two yield criteria are available for the 3-D composite shell element, these are: Hill's anisotropic theory and the modified anisotropic theory that takes into effect the differential yield strengths in tension and compression accounting for Bauschinger effect.

3.9.2 Isotropic Elastoplastic Material Model (*PLASTIC)

Stress-Strain Curve

This material model can be used only in nonlinear static analysis. Only the isotropic elastoplastic material model is available in the current version. The user may define the stress-strain curve of such a material by one of two models; these are: the elastic, perfectly plastic model, and the elastic, work hardening model. In the work hardening model, the user has the option of specifying linear hardening or piecewise linear hardening. Another option of specifying a Ramberg-Osgood stress-strain curve is available, which may be more appropriate for Aluminum and other alloy materials. The various options of specifying the stress-strain curve are shown in [Figure 3.11](#).

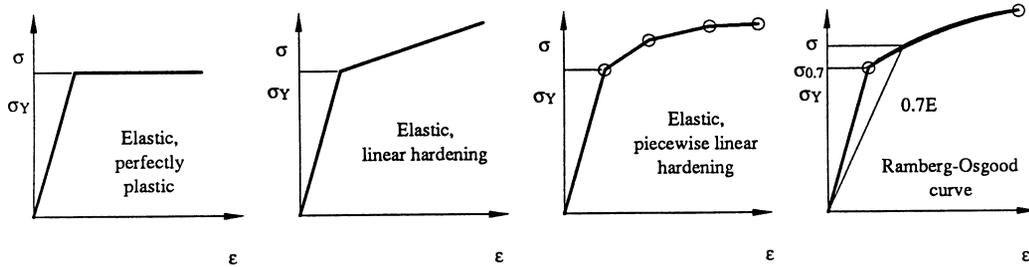


Figure 3.11: Various options of specifying stress-strain curves

Yield Criteria

The yield criterion determines the stress level or stress intensity at the onset of plastic deformations. Various yield criteria or yield conditions are available to represent various material behaviors, [3.13, 3.15]. Four common yield criteria are available in the program. These are: the von Mises, the Tresca, the Mohr-Coulomb, and the Drucker-Prager yield conditions. [Figure 3.12](#) shows a representation of these criteria in the π -plane. The von Mises yield condition is a function of the deviatoric stresses and is independent of the hydrostatic stress (σ_m). It indicates that yielding will occur when the octahedral shear stress or the elastic energy of distortion reaches a critical value. The Tresca condition is a maximum shear criteria which is also independent of the hydrostatic stress. Both Mises and Tresca conditions are suitable for modeling material behavior in which yielding is not appreciably affected by hydrostatic pressure, as in most metals and alloys with moderate hydrostatic pressures. The Mohr-Coulomb and the Drucker-Prager conditions are, on the other hand, dependent on the hydrostatic pressure and may be used for modeling concrete, rock, and soil material behaviors as an example.

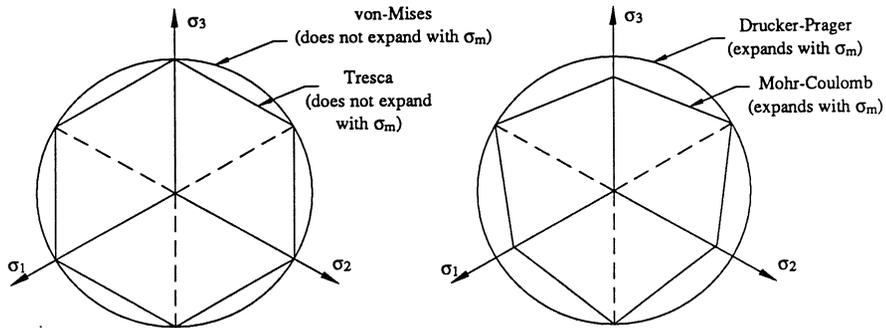


Figure 3.12: π -plane representation of various yield criteria

Work-Hardening Rule

The hardening rule determines how the yield function changes during plastic deformation, [3.13, 3.14]. Besides the no hardening assumption (perfect plasticity), three work-hardening models are available in the program. These are the isotropic, kinematic, and mixed hardening. Figure 3.13 shows a graphical representation for the above models. In the perfect plasticity model, it is simply assumed that the yield surface does not change during plastic deformation. The isotropic hardening model assumes that the yield surface grows in size only while its shape is not changing. This assumption is usually applicable to monotonic proportional loadings with no Bauschinger effect. In kinematic hardening the yield surface is assumed to have a constant size and shape, but moves in the stress space. This is more appropriate for materials with pronounced Bauschinger effect, and cyclic loading applications. A combination of the two models (isotropic and kinematic) is available in the mixed hardening model which usually gives results that are closer to experimental data.

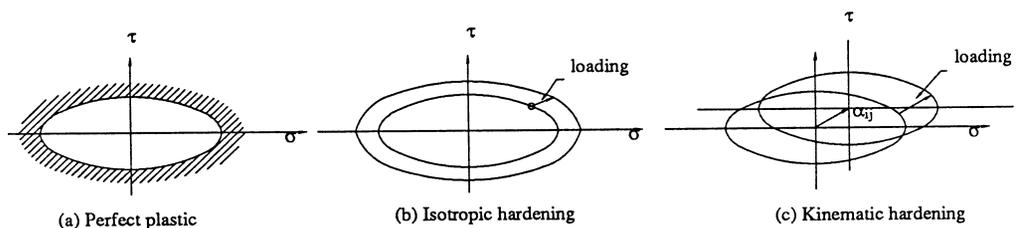


Figure 3.13: Presentation of various hardening assumptions

3.9.3 Hyperelastic Material Model (*HYPEREL)

The hyperelastic or the rubber-like material model may be used in geometric nonlinear analysis with total Lagrangian formulation. A hyperelastic material is an elastic material for which a strain-energy function exists [3.1]. The second Piola-Kirchhoff stress tensor and the constitutive matrix will be the first and the second derivatives, respectively, of the strain-energy function with respect to the Green-Lagrange strain tensor. For initially isotropic materials, the strain energy function is expressed in terms of the three invariants of the strain tensor, I_1 , I_2 and I_3 .

The incompressibility condition is expressed in terms of the third strain invariant, I_3 , and is handled as an average constraint throughout the volume of the element. This method allows a finite compressibility analysis to be performed which may be important in some actual behavior of rubber-like materials. This option is obtained by adjusting the value of Poisson's ratio, ν , to be lower than the near incompressible default value of 0.499. The default value for near incompressibility [$\nu = 0.499$] is usually sufficient for the analysis of an incompressible assumption. Higher Poisson's ratio values, closer to 0.5, may be assumed at the expense of more iterations to perform the analysis, and the possibility of divergence. For all tested verification problems, the results obtained by assuming $\nu = 0.499$, are within 3% of the exact incompressible solutions available in the literature.

Various forms of strain energy functions are available to account for the behavior of various rubber-like materials. These range from the simplest Neo-Hookean form to a generalized Mooney-Rivlin and some exponential forms. The following is a list of all the available strain energy forms in the program (refer to [Chapter 2](#) for details):

1. Neo-Hookean
2. Classical Mooney-Rivlin
3. Generalized Mooney-Rivlin
4. Swanson
5. Blatz-Ko
6. IHT
7. Klosner-Segal
8. Biderman
9. Hart-Smith (exponential)
10. Alexander (exponential)

3.9.4 Anisotropic Elastoplastic Material Model (*APLASTIC)

Stress-Strain curve in Principal Material Directions

This material model is available only for NKTP = 32 in the current version. The user may define the stress-strain curve of such a material by one of two models; these are: the elastic-perfectly plastic model, and the elastic-work hardening model. In the work hardening model, the user has the option of specifying linear hardening or piecewise linear hardening. A Ramberg-Osgood stress-strain curve option is also available. The various options of specifying the stress-strain curve are shown in [Figure 3.11](#). However for anisotropic materials these curves are needed in all the principal material directions (axial and shear). For the modified Hill's theory, all these curves are required for both tension and compression directions.

Yield Criteria:

The yield criterion determines the stress level or the stress intensity level at the onset of plastic deformations. Two common yield criteria are available for anisotropic plasticity in the NISA program. These are:

1. Hill's anisotropic yield criterion [3.13]:

$$2f(\sigma_{ij}) = F(\sigma_{22} - \sigma_{33})^2 + G(\sigma_{33} - \sigma_{11})^2 + H(\sigma_{11} - \sigma_{22})^2 \quad (3.53)$$

$$2L\sigma_{23}^2 + 2M\sigma_{23}^2 + 2N\sigma_{12}^2 = 1$$

where F, G, H, L, M and N are parameters characteristic of the current state of anisotropy, and 1, 2, and 3 are the three material principal directions. Linear terms are not included and therefore, no account for Bauschinger effect is possible.

2. Modified Hill's criterion, [3.20]:

$$3f(\sigma) = M_{ij}\sigma_i\sigma_j - L_i\sigma_i - K = 0 \quad (3.54)$$

where M_{ij} , L_i may represent the variation of the yield stress with orientation as well as strength differentials between tensile and compressive directions, K is a hardening parameter relating to the effective size of the yield surface, and σ_i is the vector form of the stress tensor σ . Linear terms are included in [Equation \(3.54\)](#), $L_i\sigma_i$, and therefore, Bauschinger effect may be accounted for.

Work-Hardening Rule:

The hardening rule determines how the yield function changes during plastic deformation, [3.13, 3.14]. Basic hardening rules for anisotropic plasticity models are the same like those described for the isotropic case. Refer to [Section 3.9.2](#) for details.

3.9.5 Creep Material Model (*CREEP)

The creep material model may be used in material and geometric nonlinear analyses. Both total and updated Lagrangian formulations can be chosen according to the element formulation. Creep material model supports von Mises yield surface and the selection should be given in *PLASTIC data group. If only elastic creep behavior is desired then *PLASTIC data group need not be given. If the *PLASTIC data group is present and the creep analysis is required to be based on elastic behavior, then a very high yield stress value should be given in *PLASTIC data group.

The creep material law can be chosen as a general function of stress, time and temperature. This permits the use of any common forms of creep laws encountered in engineering analysis such as laws due to Norton, McVetty, Soderberg, Dorn etc. Also Oak Ridge National Laboratory (ORNL) creep law can be used as one subset of general law. Detailed forms of these laws are given in [Section 6.5.8](#). Explicit or implicit time integration schemes can be chosen. In implicit scheme, midpoint, Galerkin or backward schemes are available.

3.10 Kinematic Constraints

3.10.1 Definitions

Kinematic constraints are relations among the unknowns that must be satisfied during the solution. The following forms of kinematic constraints are available:

1. Multi-point Constraint Equations (*MPCEQN)

A multi-point constraint (MPC) equation is a linear equation which relates *at least two* displacement degrees of freedom (DOF) and has the form,

$$\sum_{i=1}^n \alpha_i u_i = c \quad (3.55)$$

where n is the number of terms in the equation, α_i are the coefficients, u_i are the selected degrees of freedom (e.g., UX, UY, etc.), and c is the constant term of the equation. The first DOF mentioned in an MPC equation ($i = 1$ in the above equation) is the *dependent* DOF of the equation, the rest are the independent DOF. An example of an MPC equation is,

$$1.0 \text{ UX1} - 0.5 \text{ UX2} - 0.5 \text{ UX3} = 0$$

which may be thought of as enforcing the plane-sections-remain-plane assumption of the elementary beam theory in a problem modeled by plane stress elements, with node 1 at the beam centerline and nodes 2 and 3 at the top and bottom, respectively. MPC equations may be used for several other applications, including distribution of forces and representation of rigid regions.

2. Rigid Links (*RIGLINK)

A rigid link is a set of MPC equations which are generated internally to express the deflections at one end of the rigid link as kinematically equivalent deflections of the other end. For example, consider the rigid link joining node i to node j as shown in [Figure 3.14](#). If the deflections of node i are known, the deflections at node j can readily be computed. That is, node i is the master node and node j is the slave node. Let the position vector of node i and j be \mathbf{r}_i and \mathbf{r}_j . Denoting the displacement vectors at node i and j by \mathbf{u}_i and \mathbf{u}_j , such that,

$$u_i = [d_i^T, \theta_i^T] \quad (3.56)$$

where \mathbf{d}_i denotes translation components and $\theta_i = [\theta_{xi}, \theta_{yi}, \theta_{zi}]^T$ is the rotation vector, we have from rigid-body dynamics ([3.15]), and assuming small rotations,

$$d_j = d_i + \theta_i \times r_{ji} \quad (3.57)$$

$$\theta_j = \theta_i \quad (3.58)$$

where \times indicates vector cross product, and $\mathbf{r}_{ji} = \mathbf{r}_j - \mathbf{r}_i$.

In matrix form we have,

$$\mathbf{u}_j - \mathbf{T}_{ji} \mathbf{u}_i = 0 \quad (3.59)$$

which is a set of MPC equations of the form given in [Equation \(3.55\)](#). Matrix \mathbf{T}_{ji} is the deflection transfer matrix from node i to node j , and depends only on the coordinates of both nodes. Each equation in the above matrix equation involves up to 3 DOF, and if different local displacement coordinate systems are specified at node i and j , the constraint equations are expressed in the local systems and take the form,

$$\mathbf{u}'_j - \mathbf{R}_{ji} \mathbf{u}'_i = 0 \quad (3.60)$$

where the prime indicates local components of displacement, and \mathbf{R}_{ji} is the deflection transfer matrix from the local system at node i and to the local system at node j . Each constraint equation may involve up to 6 DOF in this case.

Up to 6 MPC equations may be generated for a 3-D rigid link, and up to 3 MPC equations for a rigid link in two-dimensions. The user has the option of designating how many and which of these general MPC equations are to be generated (to allow representation of a partial rigid link). A rigid link always has one master node and at least one slave node. The degrees of freedom, assigned to the slave nodes in the rigid link definition, becomes the dependent DOF of the generated MPC equations.

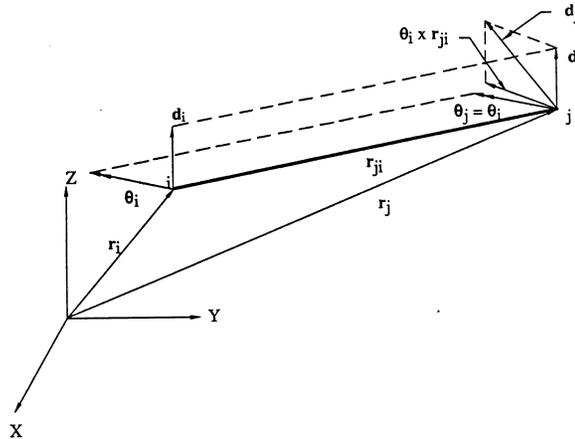


Figure 3.14: Kinematics of a rigid link

The rigid links may be used in several applications, including spot weld, bolted joints, load distribution and transfer, representation of stiff regions, and joining elements of dissimilar degrees of freedom.

The forces in the rigid links are recovered and may be requested in the output.

3. Coupled Displacements (*CPDISP)

The coupling of displacement components between two or more nodes means that these unknown components of displacement will be of the same magnitude and sign at all such nodes. Coupled displacements always refer to the *local* displacement coordinate systems at all such nodes, if any. The first degree of freedom referenced in a coupled displacement set is the independent (master) DOF, the rest are dependent DOF.

The coupled displacement feature represents the simplest form of an MPC equation, and is preferable to use in lieu of an MPC form if the constraint equation is of the form,

$$\mathbf{UX} \text{ (at node i) } - \mathbf{UX} \text{ (at node j) } = \mathbf{0}$$

Coupled displacements may be used in several applications, including representation of hinges, pin or ball joints, merging two parts of a model with different node numbering along the common boundary, and representation of partially rigid connections. In the case of hinge representation, two unique but coincident nodes may be defined, each connected to a

neighboring element. The translational components of displacement are then coupled at the two coincident nodes, whereas the rotations are left uncoupled, which implies that the rotations at the two coincident nodes are independent.

4. Specified Displacements (*SPDISP)

All of the above items are essentially in the form of multi-point constraint equations. Namely, the general MPC form (*MPCEQN), the automated form (*RIGLINK), and the simplest form (*CPDISP). All of these forms involve at least two degrees of freedom per equation, and they are an integral part of the model characteristics. That is, they are load case independent and cannot be changed with loading conditions.

A specified displacement constraint is a kinematic boundary condition that specifies the displacement value at a single degree of freedom at a node point. For static analysis, these specified displacement constraints are load case dependent, and they may be changed from one load case to another.

Specified displacement constraints are mainly used to represent support conditions and eliminate rigid body motion, to impose enforced motions such as support settlements, and to apply symmetric or antisymmetric boundary conditions.

3.10.2 Kinematic Constraint Rules

The following rules must be observed when using kinematic constraints:

1. If a local displacement coordinate system is specified at a node, any kinematic constraint used at this node refers to degrees of freedom in the *local* system, otherwise the reference is always in the global Cartesian system.
2. Nodes referenced in any kinematic constraint must be connected to an element. This requirement is not enforced for the master nodes of rigid links, where these master nodes may not necessarily be connected to elements. However, NISA does not automatically assume six degrees of freedom at a master node of a rigid link, since the active degrees of freedom at a node is always deduced from the connecting elements. In certain situations, the use of a dummy element at the master node (the simplest is a point mass element) may be necessary. For example, consider the special case shown in [Figure 3.15](#). Rigid link (R.L.) No. 1 connects node 1 (slave) and node 2 (master). Only 5 DOF are designated as dependent at node 1, since UY is constrained to zero. As a result, the master node will have 5 DOF (UX, UZ, ROTX, ROTY, ROTZ). Now the

second rigid link connects node 2 (slave) and node 3 (master), and the dependent DOF code is set to zero, which instructs the program to search for the number of DOF at node 2 and make them as dependent DOF. The search process would yield 5 DOF at node 2. Therefore, five MPC equations corresponding to the available DOF at node 2 would be generated. This, however, may not be exactly what the user intends to do. Therefore, if UY at node 2 should be considered as a dependent DOF, it has to exist first. This can be accomplished by using a point mass element (having UY as one of its DOF) at node 2.

3. Multiple definition of a dependent DOF is not permitted. That is, once a particular DOF is designated as dependent in any MPC equation, rigid link, or coupled displacement data, it must not appear as a dependent DOF again. Otherwise, there would be a conflict regarding which equation to be used to eliminate the dependent DOF.

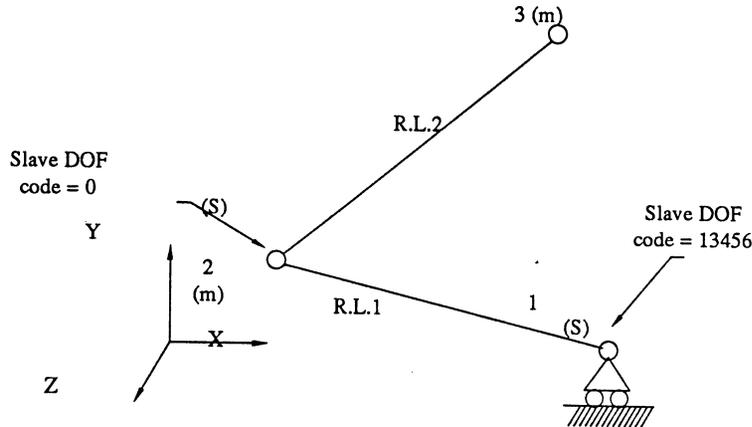


Figure 3.15: Special case where a point mass element should be used at a master node of a rigid link

4. For coupled displacements, a dependent DOF must not be referenced again (as independent or dependent) in any subsequent coupled displacement definition.
5. Dependent DOF must not be constrained in the specified displacement data. Note, however, that designating a node as a slave node in a rigid link does not automatically make all of the DOF at this node as dependent. For example, in [Figure 3.15](#) UY at node 1 is constrained to zero because of the roller support, even though node 1 is a slave node of a rigid link. However, the slave DOF code (13456) at node 1 excludes UY from the dependent DOF list.
6. Independent (master) DOF may be constrained in the specified displacement data (zero or non-zero values are both allowed).

7. An independent DOF can be referenced by any subsequent kinematic constraint as independent or dependent DOF. However, once it becomes a dependent DOF, all the above rules apply to it.
8. The MPC equations generated internally for the rigid links as well as those defined directly in *MPCEQN are resequenced internally to eliminate any sequence dependency problems. The resequencing process does not alter the ordering of MPC equations in cases where an independent DOF in any MPC equation is not a dependent DOF in a previous MPC equation (in the user-defined list) and would not become dependent in any other subsequent equation. The resequencing process does *not* alter the status of a particular DOF from a designated slave DOF to a master, or vice-versa. The resequencing process does not include DOF listed in the coupled displacement data. Dependent DOF defined in *RIGLINK or *MPCEQN must not appear in the coupled displacement data. The resequencing process fails if the MPC equations have inherent sequence dependency that cannot be removed by shuffling the equations. For example, consider the case of redundant constraints shown in [Figure 3.16](#), where a rigid region is represented by three rigid links in a closed loop. The constraints are redundant and no node can be identified as independent. The remedy is to remove *one* of the rigid links.

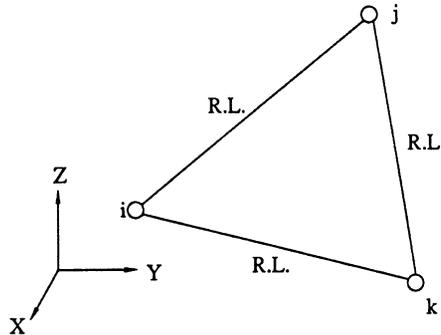


Figure 3.16: Rigid links in a closed loop (not allowed)

3.11 Static Loads

This section discusses the loads that can be applied in linear and nonlinear static analyses. Dynamic loads are discussed in [Section 3.5](#). Loads and boundary conditions for heat transfer analysis are discussed in [Section 3.8](#).

3.11.1 Types of Static Loads

The loads are classified to nodal loads applied directly at the node points, and element loads applied to elements.

Nodal Loads

1. Concentrated Nodal Forces and Moments (*CFORCE):

These loads are applied directly at the node points according to the sign convention shown in [Figure 3.1\(d\)](#). If a local displacement coordinate system is specified at a node point, the force or moment is applied in the local system. Forces applied at nonexistent nodes or degrees of freedom are ignored. If a force and a specified displacement boundary condition are applied at the same degree of freedom, a negative value of the force is added to the reaction at that degree of freedom.

2. Deformation Dependent (follower) Concentrated Nodal Forces (*CFOLLOWER):

Follower concentrated nodal forces may be used in nonlinear static analysis. These forces are deformation dependent. That is, the force direction depends on the deformation, as opposed to the non-follower force which retains the same direction irrespective of the deformation. The difference is depicted in [Figure 3.17](#). Follower forces are implemented in NISA such that a follower force is defined at a node point in the normal direction to a reference face of a reference element connected to the node. A positive value of the follower force indicates a force applied towards the reference face.

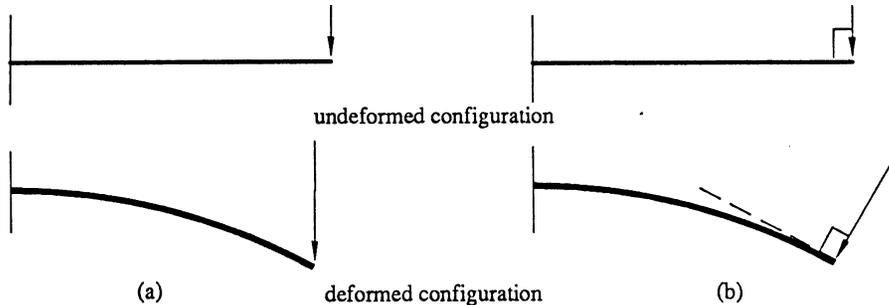


Figure 3.17: Concentrated nodal forces: (a) non-follower (conservative) force, (b) follower force

3. Non-zero Specified Displacement (*SPDISP):

Nonzero specified displacements, e.g., support settlement, represent loading applied to the structure according to the sign convention shown in [Figure 3.1\(d\)](#). They may be applied only to valid degrees of freedom in the model. If a local displacement coordinate system is specified at a node point, the specified displacement is applied in the local system.

Element Loads

All types of element loads are transformed into equivalent nodal forces. Element loads include:

1. Pressure Loads (*PRESSURE):

Uniform or non-uniform pressure may be applied to faces of elements. The pressure acts normal to the face and is positive when it acts towards the face. Refer to [Chapter 4](#) for the element face numbering convention for each element type. In nonlinear static analysis, this type of loading remains normal to the surface in the undeformed configuration and does not take into account the change in the area of the loaded surface.

2. Deformation Dependent (follower) Pressure Loads (*PRESSURE):

In nonlinear static analysis, uniform or non-uniform follower pressure loads may be applied to faces of elements. Follower pressure is a deformation dependent load that remains normal to the face during deformation. Positive pressure acts towards the face. The difference between non-follower and follower pressure is depicted in [Figure 3.18](#).

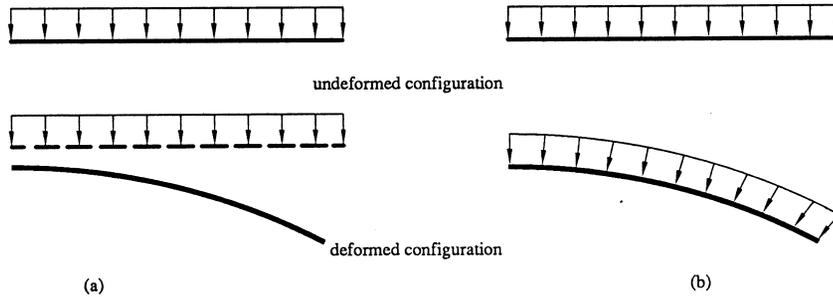


Figure 3.18: Pressure loads: (a) non-follower (conservative) pressure, (b) follower pressure

3. Special Beam Loads (*BEAMLOAD):

Distributed and concentrated loads may be applied to 2-D and 3-D beam elements at arbitrary locations along the beam axis. The applied loads may be concentrated bending moments, concentrated transverse loads, and distributed transverse load with uniform or linear variation. This type of loading is defined in the beam's local coordinate system as shown in [Figure 3.19](#).

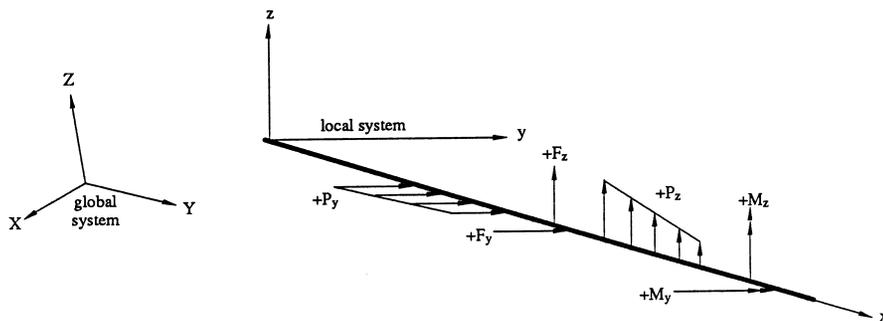


Figure 3.19: Beam loads applied at arbitrary locations along the beam axis

4. Gravity Loads and Loads due to Angular Velocity and Acceleration (*BODYFORCE):

Gravity loads and inertia loads (due to linear acceleration, angular velocity, and angular acceleration) may be applied to any element which has a mass density. The instantaneous total acceleration (Figure 3.20) of a point in space due to uniform linear acceleration field and due to angular velocity ω and angular acceleration ($\alpha = \dot{\omega}$) about an instantaneous axis of rotation passing through a reference point O' (point of rotation), which may be different from the origin of the global Cartesian system, is given by ([3.16]),

$$a_p = a_L + \alpha \times r_{po'} + \omega \times (\omega \times r_{po'}) \quad (3.61)$$

where,

- a_L = the absolute linear acceleration of the reference point O' due to the uniform linear acceleration field (or gravity).
- $\alpha \times r_{po'}$ = the tangential acceleration due to the angular acceleration α .
- $\omega \times (\omega \times r_{po'})$ = the centripetal acceleration due to the angular velocity ω .

It should be noted that the global XYZ system is rigidly attached to the model, and is used as a rotating reference frame. In general, α may not be in the same direction of ω , which implies that the axis of rotation is not fixed in space. If α and ω have the same direction, this case represents angular motion about a fixed axis, and the reference point O' may be any arbitrary point on the fixed axis of rotation. Furthermore, if α is equal to $\mathbf{0}$, this represents a steady rotation about a fixed axis.

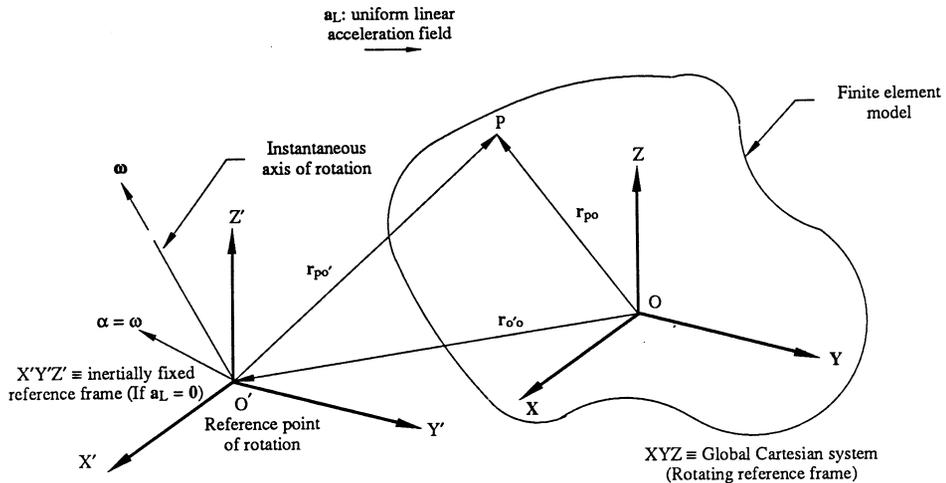


Figure 3.20: Acceleration at a point due to linear and angular motion

5. Thermal Loads (*NDTEMPER, *NDTEMPDIF):

Loads due thermal expansion or contraction may be applied to any element which has a thermal expansion coefficient as a material property. The temperature distribution is specified at the nodal points (*NDTEMPER). The temperature variation within an element is calculated from the nodal values using the isoparametric interpolation. For shell and 3-D beam elements, linear temperature gradient through the thickness may be applied. The temperature gradient loading is specified in terms of temperature difference between the top and bottom surfaces in *NDTEMPDIF data group. The nodal temperature values are also used to compute material properties which are temperature dependent.

The nodal temperature distribution and the linear temperature gradient through thickness can be generated by NISA/HEAT program for heat transfer analysis and can be directly accessed as thermal loads by NISA structural analysis program (see [Section 5.4.7](#) for more explanation).

3.11.2 Multiple Load Cases (*LDCASE) and Load Combination (*LDCOMB)

More than one load case may be specified in a single static analysis run. Each load case (*LDCASE) has a unique load case identification number, and may use any combination of the types of loads described above. The loading and boundary conditions, as well as the output request are specified for each load case separately. The specified displacement boundary conditions may be changed from one load case to another. They default to the boundary conditions of the preceding load case if they are not specified for the current load case. Other forms of kinematic constraints (*MPCEQN, *RIGLINK, and *CPDISP) are model characteristics that cannot be changed in load cases. All other types of loading (*CFORCE, *PRESSURE, *BEAMLOADS, *BODYFORCE, *NDTEMPER, and *NDTEMPDIF) should be defined, as applicable, for each load case.

The stiffness matrix is assembled and decomposed every time a different set of displacement boundary conditions is specified. The decomposed matrix is reused in a subsequent load case whenever the boundary conditions are similar to those defined in the preceding load case.

A set of boundary conditions is considered to be similar to the preceding set if it involves the same constrained degrees of freedom, regardless of the specified displacement values.

Multiple load cases are solved in the sequence in which they are defined in the data deck, and are not rearranged according to their identification numbers. The load case ID numbers are used for reference purposes only. The boundary conditions of a typical load case are internally compared to those of the preceding case. A new decomposition process is initiated if the two sets are different. Therefore, it is recommended that load cases with similar boundary conditions follow one another in the data deck.

The results from different load cases may be scaled and combined together using the load combination capability available in *LDCOMB data group. This process does not involve any decomposition of the stiffness matrix. It only involves combination of the output quantities of the referenced load cases.

3.12 Data Checking and Error Messages

3.12.1 Data Checking

NISA provides an extensive data checking capability. The data checking is automatic and does not require any special commands. As an option, however, the program may be directed to perform a check run only. This option may be activated by using the executive command 'EXECUTION = CHECK' (see [Section 5.3](#)). The check run checks for consistency and completeness of the input data, lists the wavefront element-by-element, and provides the final wavefront statistics (maximum wavefront, root-mean-square wavefront, average wavefront, total number of degrees of freedom, and estimated number of records for writing the decomposed coefficient matrix to file 30).

The wavefront parameters provided in the check run are exact, i.e., they are the same as would be obtained in a normal (execution) run. At least one load case should be specified in a check run. The wavefront minimization is also activated in the check run (this is the default option, see the executive command 'RESEQUENCE' in [Section 5.3](#)). The wavefront history (i.e., element-by-element) may also be obtained in a normal (execution) run if the executive command 'EXECUTION = CGO' is used. The check run does not generate element matrices or provide a solution, and is terminated after the wavefront check. The data checking is automatically performed during a check or a normal run. The program's interpretation of the input data is echoed in the output file for easier verification of the data (see the *ECHO command in [Section 5.2](#)). The data checking ranges from the trivial format errors to the more involved cross reference and improper kinematic constraints checking. Checks on the geometry of the finite element model such as element distortion, shell normals, etc. should be performed using the preprocessor module of the DISPLAY program.

Data checking includes, for example: improper format; duplicate element or node identification numbers; zero, negative or repeated node numbers in element connectivity; repeated definition of constraints; applying specified displacement boundary conditions at nonexistent nodes or degrees of freedom; constraining a dependent degree of freedom; disjoint structure; insufficient memory; etc. Cross reference checks include, for example: missing coordinates for nodes defined in element connectivity; reference to nonexistent material property, real constant, or lamination angle tables; reference to a nonexistent local coordinate system; multiple definitions of a dependent degree of freedom; redundancy in multi-point constraints equations and/or rigid link definitions; etc.

3.12.2 Warning and Fatal Error Messages

Warning and fatal error messages are issued as they are encountered during input reading, sorting and processing, and are written to the output file in an explicit diagnostic form. Warning and fatal error messages may also be produced during the solution phase, but these are discussed in [Section 3.14.3](#).

A warning message is issued when an action (or assumption) is undertaken by the program in regard to a certain inconsistency in the data. For example, to use the first definition of coordinates for a node which is multiply defined, or to ignore displacement boundary conditions specified at a nonexistent degree of freedom. The latter may occur when the specified displacement boundary conditions are conveniently defined for a range of nodes. The executive command 'WARNING = GO/STOP' (see [Section 5.3](#)) may be used to direct the program to continue or stop execution if warning messages are produced during the input processing phase. The default option is to continue execution.

A fatal error message is issued to inform the user that the program cannot interpret the data (e.g., format errors) or that the error is unacceptable (e.g., illegal element type number, reference to a nonexistent material property table, no loads or boundary conditions are applied in a static analysis run, etc.) Fatal errors result in an eventual termination of the execution. Depending on the severity of the fatal errors encountered, the program may continue to interpret as much data as possible before terminating. The input data should be corrected according to the fatal errors as they first appear.

3.12.3 Solution Errors

During the solution of the overall equilibrium equations (a set of simultaneous linear algebraic equations), NISA checks for the positive definiteness of the coefficient matrix and assesses the effect of round-off errors. A positive definite matrix means that all of its eigenvalues are positive. That is, all the diagonal entries in the matrix are initially positive and remain so as they are reduced (and become pivots) during Gaussian elimination. Round-off errors result because of arithmetic operations being performed with finite precision.

If an exact zero pivot is encountered, a fatal error message is issued and the execution is terminated. It is possible, however, that the solution may continue even though the coefficient matrix is singular, resulting in a displacement solution which is excessively high. This is due to arithmetic operations performed with finite precision, i.e., the way the

computer manipulates numbers. To guard against this and against an ill-conditioned coefficient matrix, the following warning message,

****WARNING - HIGH ROUNDOFF OR NEGATIVE PIVOT (CRIT, PIVOT = a, b) AT ELEMENT J****

May be issued during the solution to indicate that the matrix exceeds some nominal criterion for ill-conditioning so that the effect of round-off errors is excessive (CRIT = a), or that the current pivot term is negative (PIVOT = b).

In many cases the solution may be correct in spite of this warning. However, certain checks should be made before accepting the results of the analysis. If the pivot is negative but small in absolute value then it is likely that the original matrix is singular, but that arithmetic operations of finite precision result in a small negative number on the diagonal instead of an exact zero. In this case, one should check all the possibilities to be given later in this section as causes of singularities or high round-off errors. If the negative pivot is large in absolute value, on the order of the characteristic stiffness expected for the element, the following possibilities should be checked:

- (i) The element connectivity is not given in the order shown in the element library ([Chapter 4](#)).
- (ii) Incorrect geometric or material properties. The elastic material matrix must be positive definite. Subtle errors may result from temperature-dependent material properties which become negative, or an incorrect definition of Poisson's ratio for orthotropic materials.

In the case of a well conditioned matrix where all diagonal terms are of the same order of magnitude, it is necessary to solve an enormous number of equations before round-off errors introduce deleterious effects. However, round-off errors become significant for all size problems when the matrix is ill-conditioned. In simple terms, we can define ill-conditioning as resulting from large differences in the magnitude of diagonal terms of the coefficient matrix.

A precise measure of the sensitivity to round-off errors is provided by the so-called matrix conditioning number, defined as the ratio of maximum to minimum eigenvalues of the matrix ([3.17]). As this number approaches the limit of precision allowed by the computer's word length, the accuracy of the solution may be expected to degrade.

This round-off criterion, while precise, is difficult to apply in practice because of the computational expense associated with calculating eigenvalues of a large matrix. Therefore, in NISA the simpler diagonal decay criterion suggested by Irons in formulating the wavefront solution method ([3.18]) is used. The value of CRIT given in the warning message is this diagonal decay criterion, defined as,

$$\text{CRIT} = D_i / (\text{Pivot})_i, \quad i = 1, 2, \dots, \text{total number of equations}$$

$$D_i = [\sum K_{ii}^2 \text{summed each time } K_{ii} \text{ is modified}]^{1/2}$$

The warning message is provided whenever $\text{CRIT} > 10^6$. A large value of CRIT coupled with a pivot term which is almost zero indicates an insufficient number of constraints.

3.12.4 Causes of Singularity and High Round-off Errors

A number of modeling errors may be responsible for producing singularities, high round-off errors or negative pivots, some of these are:

1. Failing to restrain rigid body motion of the structure in static, nonlinear static and buckling analyses. This results in a stiffness matrix which is theoretically singular. Finite precision of the arithmetic operations, however, may allow the computer to solve the equations anyway. When the entire structure is insufficiently restrained, the high round-off warning message will usually occur at the last element. For eigenvalue analysis, unconstrained structures (free-free) are allowed.
2. Mechanism behavior or local singularity, as in the case of two colinear spar elements connected to each other but the displacement(s) is not restrained in the transverse direction(s) at the common node.
3. Error in suppressing spurious normal rotation (ROTZ) in shell elements. In element local coordinate system, the shell element basically has *five* degrees of freedom (three translations and two out-of-plane rotations). To allow modeling of branched shells, or shells of discontinuous curvature, the two rotations in local coordinate system are transformed to three rotations in global coordinates for assembly and solution, thus creating a singular rotational stiffness submatrix. This singularity should be compensated by neighboring elements as in the case of branched shells, or it should be removed by a boundary condition constraint specifying the rotation about the normal as zero. When the executive command 'AUTO CONSTRAINT' is set to ON (this is the default option), NISA handles the logic internally and adds an artificial torsional spring of

nominal stiffness at nodes where the shell has continuous (or nearly continuous) curvature.

4. Improper handling of end release in 3-D beam elements. The static condensation of a released degree of freedom (DOF) is performed in the element local coordinate system. For assembly in a common coordinate system, the stiffness associated with the retained DOFs is transformed to global coordinates. Singularity arises if the stiffness associated with a released DOF is not properly compensated from neighboring elements, otherwise displacement boundary conditions should be used. NISA handles the logic internally if all the beam elements meeting at a common node have all rotations or all translations released.
5. Incorrect specification of material properties or real constants. For example, a frequently encountered error is to neglect the specification of the thicknesses at all of the element nodes for shells or plane stress elements, or to give an incomplete or invalid set of properties for orthotropic materials. These errors lead to a singular stiffness matrix, which may in some cases be solved due to finite computer word length, as mentioned above. A similar result would follow from an error in element connectivity or node location which leaves the element with zero or negative volume or excessive distortion. In a case like this, the offending element is likely to be one which precedes the element giving the warning message.
6. A large difference in characteristic stiffness of parts of a structure may result in a problem which is physically well posed but mathematically ill-conditioned. Characteristic stiffness terms depend on the size, aspect ratio, and material properties of the elements. A structure containing both rubber and steel, which may have elastic moduli differing by several orders of magnitude, necessarily gives rise to stiffness matrices which are less well conditioned than those obtained for a homogeneous structure. Another possibility is large stiffness gradients in the same element. Using solids to model a thin shell, for example, results in a through-the-thickness stiffness which differs from the inplane stiffness in the same proportion as the thickness to side length ratio. If this is very small then numerical ill-conditioning problems may result (this problem is much less severe for the 3-D hybrid solid element, $NKTP = 9$, than it is for the conventional 3-D isoparametric solid element, $NKTP = 4$). A better model will be obtained by modeling with the general shell element ($NKTP = 20$). However, even in this case, in the limit as the thickness to side length ratio becomes very small (say 10⁻⁴), the ratio of transverse shear stiffness to the bending stiffness becomes very large, and ill-conditioning is again suspected ([3.19]). For these very thin plates and shells, the use of the 3-D thin shell element ($NKTP = 40$) which neglects the transverse shear effects, will give better results.
7. In view of the above comment, it is apparent that introducing artificially stiff or flexible components into a structure is a procedure to be done carefully, if at all. Facilities

within NISA to model rigid areas or regions are available. The temptation to model a rigid link as a beam with very high properties should be avoided, kinematic constraints should be used instead.

8. In Eigenvalue or buckling analyses, high round-off errors may occur if the shift point is close to one of the eigenvalues of the system.
9. To summarize, the solution warning message indicates that the system of equations being solved is either not positive definite or is somewhat ill-conditioned, rendering it susceptible to round-off errors. If the magnitude of the round-off criterion, CRIT, is small compared to the precision allowed by the computer's word length, then the message may be considered to be an informative diagnostic, and prudence dictates that engineering judgement be used to verify the adequacy of the finite element model. If the value of CRIT is in the order of or exceeds such precision, the solution should be rejected and the model be searched for serious errors. The comments given in the above section and the modeling guidelines given in Appendix A will provide guidance in isolating any errors.

3.13 High Performance Solvers

NISA II offers a choice of four solvers: the basic frontal (direct, wavefront) solver, the iterative solver, the fast direct sparse matrix solver and Parallel direct sparse solver (PARDISO) from Intel®. Depending on the problems on hand, the latter three solvers can drastically reduce the solution (or elapsed) time and the computing resource (CPU time and disk space) requirements by orders of magnitude. This will help analysts and design engineers to improve their productivity and product quality by arriving at better solutions faster.

3.13.1 Wavefront Solver

NISA uses the wavefront technique for the solution of the overall finite element equilibrium equations, which are in the form of simultaneous linear algebraic equations. For most practical problems, the computer time required for the solution of the system of equations represents a substantial portion of the total computer time of the run. In the wavefront (frontal) method ([3.18]), the solution time is proportional to the square of the wavefront size. Therefore, it is important to optimize the wavefront size. The frontal technique uses Gauss elimination method for the solution of the simultaneous linear equations (see Section 2.9.1 for more details). It alternates between assembly and decomposition of the coefficient matrix, with the assembly proceeding in the order in which the elements are defined. As a result, the entire (global) coefficient matrix is never totally in core. This is unlike the bandwidth techniques in which the element contributions to the overall equations are fully assembled before factorization.

In the frontal technique, the ordering of the elements determines the wavefront size. The node numbering sequence does not have any effect on the wavefront size. Again, this is unlike the banded solvers in which the node numbering is what affects the bandwidth. The instantaneous wavefront size is defined as the number of active equations that are in core when a typical element is assembled to the coefficient matrix in the front. When an equation is fully summed, meaning that the corresponding unknown made its last appearance and will not appear again in any other subsequent element, it is eliminated from the rest of equations, thus making room for a new unknown to occupy the same place. As an element is brought into the front, unknowns making their first appearance occupy vacant or new positions in the coefficient matrix, and unknowns which appeared before are assembled to the existing coefficients. The assembled coefficient matrix, therefore, keeps expanding and contracting as elements are assembled and unknowns making their last appearance are eliminated.

Three quantities are defined to reflect the varying size of the frontwidth. The maximum wavefront (W_{\max}) is the maximum size the instantaneous wavefront can get to,

$$W_{\max} = \max(W_i), \quad i = 1, 2, \dots, M \quad (3.62)$$

where M is the number of elements, and W_i is the instantaneous wavefront given by,

$$W_i = W_{i-1} + F_i - L_{i-1}, \quad i = 1, 2, \dots, M \quad (3.63)$$

where F_i is the number of variables making their first appearances as element (i) is assembled to the front, and L_{i-1} is the number of variables making their last appearances after element number ($i-1$) was assembled to the front. Clearly, W_{M+1} must be zero. The average wavefront size (W_{av}) is defined as,

$$W_{\text{av}} = \frac{1}{M} \sum_{i=1}^M W_i \quad (3.64)$$

The average wavefront size represents the average length of an eliminated equation. The eliminated equations constitute the decomposed coefficient matrix and are written to NISA file 30. Thus, a rough estimate of the number of words in the decomposed matrix is $M * W_{\text{av}}$. The solution time is proportional to the root mean square wavefront (W_{rms}) defined by,

$$W_{\text{rms}} = \left[\frac{1}{M} \sum_{i=1}^M W_i^2 \right]^{1/2} \quad (3.65)$$

The largest problem that can be solved using NISA depends on the memory available at a particular installation. On most main frames and mini computers, the problem size is limited to a maximum wavefront of 2000. The maximum wavefront should not be confused with the total number of degrees of freedom in the problem. The wavefront limit is sufficiently large for most practical problems. It is not unusual to have a problem with 30,000 DOF and a maximum wavefront of 500.

The three measures of the wavefront size are printed in the output file. In addition, the wavefront history (the instantaneous frontwidth as the front advances through the elements) may also be printed, see the executive command 'EXECUTION' in [Section 5.3](#).

As mentioned earlier, the wavefront size depends on the element sequence. A nodal degree of freedom becomes active as the element in which it appears first is assembled into the front, and it becomes eligible for elimination when it makes its last appearance. In order to reduce the wavefront size, the elements should be ordered such that the first and last appearance of a typical node, as implied by the element connectivity, is as close as possible. This is evident from [Equation \(3.63\)](#), where the objective is to make the quantity $(F_i - L_{i-1})$ minimum. In other words, the element should be sequenced such that the wavefront advances through the model continuously from one end to the other in the direction which has the largest number of nodal points.

It is important to note that the element sequence is defined according to the ascending order of the element identification numbers and *not* by the sequence in which they are entered in the data deck.

As an example for the effect of element numbering on the wavefront, consider the mesh shown in [Figure 3.21](#), where two different element numbering schemes are used. It is evident from the figure that the minimum wave front results when the front sweeps across the model in the direction of the largest number of nodes.

Kinematic constraints (multi-point constraint equations, rigid links, and coupled displacements) affect the wavefront size. Although the total number of degrees of freedom may decrease due to the presence of kinematic constraints, the maximum wavefront may actually increase. This is because a constraint equation implicitly connects the elements associated with the degrees of freedom referenced in the equation, although these elements may not have common nodes (i.e., no neighbors). Specified displacement boundary conditions, on the other hand, can only reduce the wavefront, since they correspond to degrees of freedom which are not assembled to the coefficient matrix in the front, to begin with.

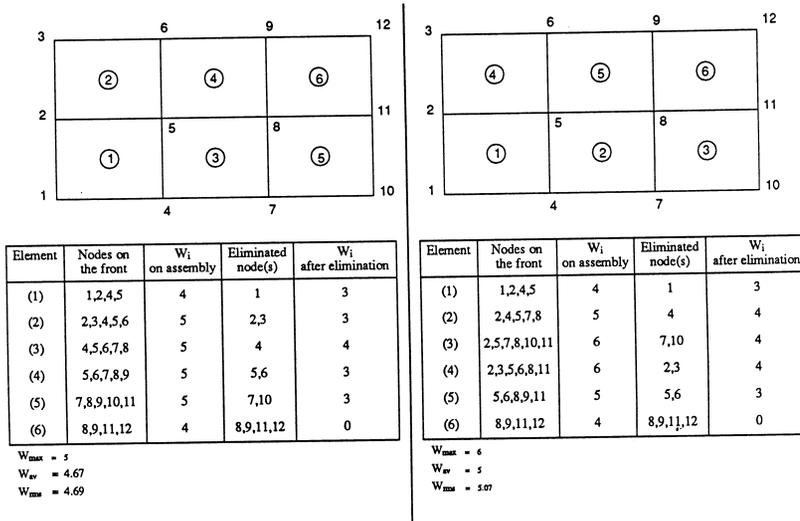


Figure 3.21: Effect of element numbering sequence on the wavefront for a simple mesh (one DOF per node is assumed)

3.13.2 Wavefront Optimizer

One may easily optimize the wavefront size for a simple mesh such as the rectangular mesh shown in Figure 3.21. But it becomes exceedingly difficult and more time consuming to minimize the wavefront for a more complicated mesh. In addition, the elements are generally numbered in an order which is convenient to the user for easier identification of the regions in the model. In many cases, the user element sequence, defined by the ascending order of the element ID numbers, does not yield a cost-effective wavefront size. As mentioned earlier, the solution time is proportional to the square of the frontwidth.

NISA has a built-in wavefront optimizer which does not require additional input from the user, and can be controlled using a single command (see the executive command 'RESEQUENCE' in Section 5.3). The internal element sequence is invisible to the user. All communications between the user and the program, as far as the input specification and the output request are concerned, are in terms of the user's element identification numbers.

The wavefront optimizer is a direct element resequencing algorithm based on topology, and level structure, and it favors minimum front growth criterion (the quantity $F_i - L_{i-1}$ in

Equation (3.63)). Several tie-breakers are used when the criterion is satisfied by several candidates.

The optimizer takes into account the kinematic constraints (multi-point constraint equations, rigid links, and coupled displacements), since these constraints affect the frontwidth (this is the default option, see executive command 'RESEQUENCE'). The specified displacement (or temperature in heat transfer) boundary conditions are not considered in the resequencing process, since these conditions are load case dependent, and can only reduce the frontwidth. The optimizer is only invoked once prior to the solution of all load cases. The case of disjointed structure can also be handled. A disjointed structure means that the model consists of independent regions, which are not connected to each other either directly through element connectivity, or indirectly through constraint equations. A warning message is issued in this case indicating the number of independent regions, and the elements at which disconnections occur.

The wavefront optimizer is cost-effective. The CPU time spent in the resequencing process is only a small fraction of the total CPU time. The latter is, of course, reduced due to reduction of the CPU time in the solution phase. For medium size problems, say less than 5000 DOF, the wavefront optimization takes about 5 to 10% of the total CPU time, and about 1 to 10% for larger size problems, depending on the computer installation.

The algorithm performs five independent trials, each with a different starting element, for the minimization of the frontwidth. The starting elements are chosen such that they are the least connected to the model or the most distant from each other (in the topological sense). A typical trial may be terminated before completion if it becomes apparent that the previous trial has better wavefront statistics. The final internal element sequence is chosen from the completed trial with the least mean square frontwidth (this is the default option, see executive command 'RESEQUENCE').

It should be noted that the wavefront statistics reported in the wavefront optimization module are upper bounds but close estimates for the actual wavefront parameters, for the following reasons:

1. The optimizer works with nodes on the front rather than nodal degrees of freedom on the front. During the solution, some nodal degrees of freedom may be eliminated before all the elements connected to the node are assembled, as in the case when elements of dissimilar DOF are joined together.

2. The specified displacement boundary conditions, which reduce the number of active DOF, consequently the frontwidth, are not considered in the resequencing process.

3.13.3 Sparse Matrix Solver

A direct, sparse matrix solver, was released in the Version 9.0 of NISA II for structure analysis. Similar to the basic frontal solver, it decomposes the global matrix and goes through forward and backward substitution to obtain a solution. Once the global matrix is decomposed, multiple solutions can be obtained very fast from forward and backward substitution.

Compared to the basic frontal solver, the sparse matrix solver stores the global matrix in a very compact form, and the operations are performed only on nonzero values. Therefore, the sparse matrix solver requires far less disk space and solution time than the basic frontal solver, especially for large-size problems.

On the one hand, the sparse matrix solver may be slower than the PCG iterative solver in solving problems for a single solution case. On the other, it has advantages over PCG solver as follows:

1. It can solve a large number of multiple load cases efficiently. In PCG iterative solver, each load case requires about the same solution time.
2. It can handle the negative definite matrix in buckling analysis and eigenvalue analysis. In contrast, PCG iterative solver can only solve the positive definite matrix.

With the help of the sparse matrix equation solver, NISAI can speed up the solution time not only in static and transient dynamics analysis but also in buckling and eigenvalue analysis. It can be applied to all analysis types except for the contact analysis (of nonlinear static).

3.13.4 Iterative Solver

The Iterative Solver in NISA II is a Preconditioned Conjugate Gradient (PCG) solver. The PCG solver pre-conditions the global matrix instead of decomposing the global matrix, and then iterates the solution based on the pre-conditioned global matrix until the solution converges to certain accuracy. The pre-conditioning process takes very little time compared to the PCG iteration process and is based on the underlying physics of the discretization of a

continuous problem, geometry of the elements, and characteristics of the different type of elements.

The PCG iterative solver is currently applicable for the following modules:

- Linear and non-linear NISA/STATIC
- Linear and non-linear Direct Transient NISA/DYNAMICS
- Steady State and Transient NISA/HEAT

The default option is the frontal solver. Users should use SOLVER executive control to activate the PCG iterative solver ([Section 5.3.2](#) for details).

Iterative vs. Direct Solvers

A newer version of PCG iterative solver released in NISA II Version 16.0 is designed to run faster and use less disk space for models created by Automesh. 3D tetrahedron element (NKTP = 4, NORDR = 21) stiffness matrix is generated inside the solver at the time the stiffness matrix is needed. This feature can achieve a big saving in disk space requirement. With this new feature in the solver, the same machine can be used to solve much larger-size problems than the previous version.

1. The iterative solver can be more efficient for large problems, since for well conditioned matrices, the solution time required by the iterative solver very closely approximates a linear function of the problem size. On the other hand, for problems with large wavefronts, the number of operations needed by the direct (frontal) solver to decompose the global matrix is proportional to the square of the wavefront size.
2. Direct (frontal) solvers are very efficient for small to medium sized problems. Once the matrix is decomposed, the backward substitution requires only a fraction of time in Gauss elimination. For linear static problems with multiple load cases (same BCs), the (frontal) solution time on the subsequent load cases is very small. This is also true for eigen-extraction, wherein multiple vectors have to be solved with respect to the same matrix.

However, in the PCG solver, for multiple load cases, the total solution time is a linear multiple of the solution time needed for each load case. Each load case takes about the same solution time because the iterative solver spends time in the iterative process and not in pre-conditioning process.

3. Solution speedups on 3D models will be higher than on 2D models since the wavefronts are smaller in 2D models.

Further, speed gains in shell models will be less than solid modes because of the ill-conditioning of the shell elements. The shell element formulation usually consists of 5 DOF, ignoring the stiffness in the normal direction, thereby causing ill-conditioning of the global matrix. This leads to slower solution speed than those obtained with solid elements, even though all the shell elements may have good aspect ratio.

4. A summary of the differences between Direct (frontal) solver and PCG iterative solver.

PCG iterative solver

1. Pre-conditioned global matrix
2. Faster solution for large size problem
3. Requires less disk space
4. Each solution takes about the same computation time for multiple load cases
5. Same sized problems take different solution times depending on the condition of the coefficient matrix
6. Accuracy of the solution is controlled by the given convergence tolerance

Direct solver

1. Decomposed global matrix
2. Good for small to medium size problem
3. Requires more disk space
4. Subsequent load cases with the same BCs need less computation time than the first load case
5. Same sized problems with the same size of wavefront takes the same computation time to solve
6. Accuracy of the solution is controlled by the number of digits in the computer

3.13.5 Parallel Direct Sparse Solver (PARDISO)

Parallel direct sparse solver (PARDISO) from Intel® Math kernel Library (Ref. 3.24) is integrated into NISA II. Similar to the sparse matrix solver, it decomposes the global matrix and goes through forward and backward substitution to obtain a solution. Once the global matrix is decomposed, multiple solutions can be obtained very fast from forward and backward substitution.

Compared to the basic frontal solver, the parallel direct sparse matrix solver stores the global matrix in a very compact form, and the operations are performed only on nonzero values. Therefore, the sparse matrix solver requires far less disk space and solution time than the basic frontal solver, especially for large-size problems.

With help of the parallel direct sparse solver, NISA II can speed up the solution time on multi core hardware platform. This solver can be used with additional input parameter to make the system use either in-core or out-of-core memory. For very large sized problems when the system RAM is not sufficient the solver becomes drastically slow if it uses the virtual memory of the system. In that case, solver can be made to use the system hard disk space by selecting the out-of-core option in the solver type. However, this will also render the solving speed slower than the in-core solving speed.

Notes:

1. Space Requirements

The disk space requirements for the PCG solver are substantially lower than for the direct (frontal) solver for large size problems.

The PCG solver requires only the non-zero elements in the sparse matrix to perform its iterative process. Therefore, the amount of storage is much less than the amount of space required to keep the decomposed matrix for the direct solver.

(i) Frontal Solver

Estimated memory requirement = $\text{maxpa} * \text{maxpa} * 8 / 1.8$ bytes

Estimated Disk space requirement = $\text{avgwf} * \text{ndof}$

where,

maxpa = maximum wavefront, avgwf = average wavefront, ndof = number of unknowns

(ii) PCG Iterative Solver

Estimated memory requirement = $\text{ndof} * \text{storage-per-dof}$ bytes

where,

storage-per-dof = 600 for structural models, and

storage-per-dof = 300 for scalar problems (i.e., heat transfer)

Estimated disk space requirement = 10% more than the memory requirement.

This formula overestimates for shell and 2D problems, and underestimates for 20-noded hexahedral elements.

A problem with 50,000 unknowns and 1500 average wavefront would then require 33MB disk space with the PCG iterative solver versus 750MB disk space with the frontal (direct) solver.

This represents a substantial savings in disk space for the PCG solver.

(iii) **Large Problems**

It is strongly recommended that users devise their systems to ensure adequate amount of RAM/swap space, as well as disk space. In general, such considerations may not be important, but large problems would require large RAM/swap spaces to cooperate with NISA II in running the PCG iterative solver.

2. In-Core & Out-of-Core

When a problem becomes very large, the system may not have enough memory to keep all the required memory in-core. Virtual memory, which is much slower than the in-core memory, will then be used by the system.

To avoid the inefficient paging of virtual memory, an out-of-core option is built into the iterative solver to maintain the preconditioned global matrix on disk during the Conjugate Gradient iterative process, thereby reducing the memory requirement of this computing intensive process by a factor of two to three. This facility to use the out-of-core memory is available with the Parallel Solver as well.

This option should be activated if the memory requirement is significantly higher than available in-core memory.

3. More RAM for Iterative Solver

The PCG iterative solver runs much faster in in-core memory than out-of-core. Hence, the more the available RAM, the more efficient is the performance of the iterative solver.

The PCG solver uses extra memory, other than the memory reserved for NISA in the common block. A larger version of NISA II implies less RAM is left for the PCG iterative solver to work with. This will force the PCG iterative solver to use the slow memory (i.e., swap space) provided by the virtual memory.

4. Limitations

- (i) The PCG iterative solver can solve positive definite linear equations. Therefore, any matrix with zero pivot or negative pivot will be detected by the solver, and NISA II will stop the execution. A fatal error message is issued to the output file.

In general, negative pivot is caused by highly distorted elements. It is better (than not) to correct the shape of those elements to avoid negative pivots in the model. However, if the distorted elements are not in the critical locations affecting the overall solution, one could carry out the analysis without refining the model.

- (ii) NISA II provides an option for the iterative solver to modify the negative pivots of element matrices to positive values, thus allowing the iterative solver to obtain a solution without interruption.

This option is available only for linear analysis and should be used with caution.

- (iii) The PCG iterative solver can currently be used with all elements in the NISA II element library, *except surface to surface contact element*. The PCG solver supports *gap* and *friction* elements. Further, the *stiffness matrix requires no manipulation* for the PCG solver to be used with such a diverse range of finite elements.
- (iv) The Sparse Matrix solver can be applied to all analysis types except for the contact analysis (of nonlinear static).

3.14 Output Features and Postprocessing

3.14.1 General Description of the Output

This section discusses the information provided in the NISA printout for all analysis types, namely: static, nonlinear static, buckling, eigenvalue, direct linear transient dynamic, direct frequency response and heat transfer analyses as well as modal dynamic analyses. Also described, in a table format, are the main features of the postprocessing module (DISPLAY-POST) of the DISPLAY program.

[Table 3.3](#) gives a general description of the printout for static, nonlinear static, and eigenvalue analyses, and references the pertinent sections in this manual for detailed information about each output item. For buckling analysis the output is similar to static analysis (for the static pass) and is similar to eigenvalue analysis (for the eigenvalue pass) except that only the load factors and mode shapes may be provided. For direct frequency response analyses, the output is similar to direct linear transient analysis except that the real part of the results appears first followed by imaginary part. The output for modal dynamic analyses is given in [Table 3.4](#), whereas the output for heat transfer analysis is given in [Table 3.5](#). A detailed output description for a sample static problem is given in [Appendix B](#).

As shown in the tables, most of the output items are requested through the analysis data ([Chapter 7](#)), some output items are automatic and some output items are requested through the executive commands ([Section 5.3](#)).

For static analysis, most of the output items are requested and controlled by the *LDCASE and the *PRINCNTL data groups ([Section 7.1.1](#) and [Section 7.5.3](#), respectively). The output for eigenvalue analysis is mostly controlled by the *MODEOUT, *EIGOUT and *PRINCNTL data groups ([Section 7.1.8](#), [Section 7.5.1](#) and [Section 7.5.3](#), respectively). For nonlinear static analysis, the *NLOUT and the *PRINCNTL data groups ([Section 7.5.2](#) and [Section 7.5.3](#), respectively) control most of the output. For direct transient dynamic analysis and the direct frequency response analysis, the *NLOUT, *PRINCNTL and *HISTOUT data groups ([Section 7.5.2](#), [Section 7.5.3](#), [Section 7.5.9](#), respectively) control the output. For heat transfer analysis, the *PRINCNTL and *TEMPOUT data groups ([Section 7.5.3](#) and [Section 7.5.8](#)) control most of the output. For modal dynamic analysis, most of the output is controlled by the *RSET data group ([Section 8.8.1](#)) along with other data groups which are analysis dependent. For shock spectrum analysis, however, the *RESPONSE data group ([Section 8.8.7](#)) controls most of the output.

Most of the output items may be suppressed, printed in its entirety or selectively printed for a subset(s) of its members (nodes or elements). See the *SETS and the *PRINCNTL data groups (Section 6.7.1 and Section 7.5.3, respectively) for details. Users are cautioned to review the default options given in the *PRINCNTL data group. These default options are chosen such that the printout of lengthy output items (e.g., element stresses) is suppressed unless explicitly requested by the user.

3.14.2 Postprocessing of the Results

Graphical representation of the analysis results may be obtained interactively through the postprocessing module of the DISPLAY program (DISPLAY-POST), which is a 3-D color graphics program with extensive plotting features for displaying the results. All analysis types are interfaced with the DISPLAY-POST through two binary files: the basic data file (file 26) and the post data file (file 27).

The postprocessing procedure for a typical analysis type is as follows:

1. Run NISA and save files 26 and 27 through the executive commands 'FILE NAME' and 'SAVE FILE' (Section 5.3.1). For heat transfer analysis only file 26 should be saved.
2. For modal dynamic analyses, run the pertinent analysis types, e.g., transient, shock, etc., using files 26 and 27 from the eigenvalue run. These files will be updated and saved automatically.
3. Use DISPLAY-POST program for interactive processing of the results.

Table 3.6 shows some of the postprocessing main features of the DISPLAY-POST program. These features include geometry plotting, colored contours and XY plotting. Complete description of postprocessing is documented, however, in the DISPLAY User's manual.

Table 3.3: Output description for static, nonlinear static, direct transient, direct frequency response and eigenvalue analysis

Output Item	Requested By ¹		
	STATIC	NLSTATIC, DIRECT FREQUENCY ³ , LTRANSIENT, NLTRANSIENT	EIGENVALUE ²
<u>General output</u>			
Annotated echo of input data	*ECHO (5.2)	*ECHO (5.2)	*ECHO (5.2)
Warning and fatal error messages	Automatic (3.12)	Automatic (3.12)	Automatic (3.12)
Problem parameters (number of elements, active nodes, constraints; wavefront statistics)	Automatic (3.13)	Automatic (3.13)	Automatic (3.13)
Resequenced element table	Executive command 'RESEQUENCE' (5.3.2)		
Wavefront history (element-by-element)	Executive command 'EXECUTION' (5.3.2)		
Natural frequencies/load factors and computations tolerances on eigenvalues	—	—	Automatic
Convergence history for eigenvalues CPU time (total and itemized) and elapsed time	Automatic	Automatic	Automatic
Modal participation factors, modal masses and cumulative modal masses	—	—	Automatic(2.4)
Model geometric properties (total volume, total mass and mass moment of inertia)	Executive command 'GEOM' (5.3.2)		
<u>Nodal output</u>			
Applied load vector	*PRINTCNTL (7.5.3)	—	—

Output Item	Requested By ¹		
	STATIC	NLSTATIC, DIRECT FREQUENCY ³ , LTRANSIENT, NLTRANSIENT	EIGENVALUE ²
Nodal displacement vector/mode shape in global Cartesian system (in local displacement systems if specified, or in both)	*LDCASE (7.1.1) *PRINTCNTL (7.5.3)	*NLOUT (7.5.2) *PRINTCNTL (7.5.3)	*EIGCNTL (7.1.2) *MODEOUT (7.1.8) *EIGOUT (7.5.1) *PRINTCNTL (7.5.3)
Nodal force balance	*LDCASE (7.1.1)	—	—
Reactions (in local displacement systems if specified, otherwise in global Cartesian system) and summation of reactions in global system			
Averaged nodal stresses and associated principal stresses, maximum shear, octahedral shear and von Mises equivalent stresses. In nonlinear static analysis, effective stress, yield stress and equivalent plastic strain are also provided. For shell elements, this output is provided for the top, middle and bottom surfaces.	*LDCASE (7.1.1) *PRINTCNTL (7.5.3)	*NLOUT (7.5.2) *PRINTCNTL (7.5.3)	*MODEOUT (7.1.8) *EIGOUT (7.5.1) *PRINTCNTL (7.5.3)
<p>1: refer to section shown between parentheses for details; dash (—) indicates output item not applicable or not available</p> <p>2: modal values are provided</p> <p>3: In direct frequency analysis, stress resultants, like von Mises stresses, which are non linear function of displacements are not available.</p>			

Output Item	Requested By ¹		
	STATIC	NLSTATIC, DIRECT FREQUENCY ³ , LTRANSIENT, NLTRANSIENT	EIGENVALUE ²
Cauchy or second Piola-Kirchhoff stresses, effective stress, yield stress and equivalent plastic strain for NLSTATIC <i>For laminated composite elements</i> <ul style="list-style-type: none"> - Layer stresses in material principal directions (and/or in global directions for composite solid) - Layer failure criteria, maximum stress, Hill-Mises and Tsai-Wu. (Only for linear analysis) - Stress resultants for composite shells - Cauchy or second Piola-Kirchhoff stresses, effective stress, yield stress, and equivalent plastic strain for NLSTATIC. 	— *LDCASE (7.1.1) *PRINTCNTL (7.5.3) *SFDCOMP (7.5.6)	*NLOUT (7.5.2) ³ *PRINTCNTL (7.5.3) *NLOUT (7.5.2) ³ *PRINTCNTL (7.5.3)	— *MODEOUT (7.1.8) *EIGOUT (7.5.1) *PRINTCNTL (7.5.3) *SFDCOMP (7.5.6)
1: refer to section shown between parentheses for details; dash (—) indicates output item not applicable or not available 2: modal values are provided 3: not applicable to linear direct transient analysis 4: not available in nonlinear static or dynamic analysis			

NISA Capabilities

Output Features and Postprocessing

Output Item	Requested By ¹		
	STATIC	NLSTATIC, DIRECT FREQUENCY ³ , LTRANSIENT, NLTRANSIENT	EIGENVALUE ²
<p>For beams, spars and springs</p> <ul style="list-style-type: none"> - Axial force for spars and translational springs - Torsion moment for torsional springs - Stress resultants in element local coordinate system for beams (normal force, shear forces, bending and twisting moments), stresses at specified points on the cross-section for 3-D beam elements (requires specification of stress recovery points in the real constant table, *RCTABLE) 	<p>*LDCASE (7.1.1) *PRINTCNTL (7.5.3)</p>	<p>*NLOUT (7.5.2) *PRINTCNTL (7.5.3)</p>	<p>*MODEOUT (7.1.8) *EIGOUT (7.5.1) *PRINTCNTL (7.5.3)</p>
<p>Element centroidal strains³ (for continuum, shell and laminated composite elements)</p>	<p>*LDCASE (7.1.1) *PRINTCNTL (7.5.3)</p>	—	<p>*MODEOUT (7.1.8) *EIGOUT (7.5.1) *PRINTCNTL (7.5.3)</p>
<p><u>Miscellaneous and sorted output</u></p>			
<p>Forces in rigid links</p>	<p>*LDCASE (7.1.1) *PRINTCNTL (7.5.3)</p>	—	—
<p>Tabulation of nodes with stresses outside specified range (for principal stresses, maximum shear, octahedral shear and von Mises equivalent stress)</p>	<p>*LDCASE (7.1.1) *STRSFILTER (7.5.5)</p>	<p>*NLOUT (7.5.2) *PRINTCNTL (7.5.3)</p>	<p>*MODEOUT (7.1.8) *EIGOUT (7.5.1) *PRINTCNTL (7.5.3)</p>

Output Item	Requested By ¹		
	STATIC	NLSTATIC, DIRECT FREQUENCY ³ , LTRANSIENT, NLTRANSIENT	EIGENVALUE ²
Highest n nodal stresses in descending order of magnitude (for principal stresses, maximum and octahedral shear, and von Mises equivalent stress)	*LDCASE (7.1.1) and executive command 'SORT' (5.3.3)	—	—
Highest n element centroidal stresses in descending order of magnitude (for principal stresses, maximum and octahedral shear, and von Mises equivalent stress)			
Time history			
Displacement, velocity, acceleration, stresses, reaction forces, and beam end forces	—	*HISTOUT (7.5.9)	—

1: refer to section shown between parentheses for details; dash (—) indicates output item not applicable or not available

2: modal values are provided

3: not available in nonlinear static analysis

Table 3.4: Output description for modal dynamic analysis

Output Item	Requested By^a
<p><u>General output</u></p> <p>Annotated echo of input data</p> <p>Warning and fatal error messages</p> <p>Natural frequencies and damping values for the selected modes</p> <p>CPU time (total and itemized) and elapsed time</p>	<p>*ECHO (5.2)</p> <p>Automatic (3.12.2)</p> <p>Automatic (3.5.2)</p> <p>Automatic</p>
<p><u>Transient Dynamic Analysis</u></p> <p>Parameters for time integration (number of time steps, number of time regions, etc.)</p> <p>Output of nodal time history, maximum absolute values and times of maxima over a user specified time region at selected nodes for the following responses:</p> <ul style="list-style-type: none"> - Displacements, velocities and accelerations - Reaction forces - Global stresses for continuum elements - Global top and bottom layer stresses for shell elements - Layer stresses and stress resultants for composite elements <p>Output of time history, maximum absolute values and times for maxima for the following line element stress resultants:</p> <ul style="list-style-type: none"> - Axial forces for spars and translational springs - Torsional moment for torsional springs - Beam normal force, shear forces, bending and twisting moments in local coordinate system 	<p>Automatic (3.5.4)</p> <p>*RSET (8.8.1)</p> <p>*HISTORY (8.8.2)</p> <p>*RSET (8.8.1)</p> <p>*HISTORY (8.8.2)</p>

Output Item	Requested By^a
<p>Snapshot (or freeze) nodal output (3.5.3) at a user specified time or at an instant in time when the user specified response peaks for the following responses:</p> <ul style="list-style-type: none"> - Displacements, velocities and accelerations - Reaction forces - Global stresses for continuum elements - Global top and bottom layer stresses for shell elements - Layer stresses and stress resultants for composite elements 	<p>*RSET (8.8.1) *SNAPSHOT (8.8.3)</p>
<p>Snapshot (or freeze) element response output for the following line element stress resultants at a given time or at an instant in time when the user specified response peaks:</p> <ul style="list-style-type: none"> - Axial forces for spars and translational springs - Torsional moments for torsional springs - Beam normal force, shear forces, bending and twisting moments in local coordinate system 	<p>*RSET (8.8.1) *SNAPSHOT (8.8.3)</p>
<p><u>Random Vibration Analysis</u></p> <p>Analysis parameters (integration scheme, number of frequency points, modal bandwidth, number of correlated groups, etc.)</p> <p>Symmetric banded covariance matrices for modal displacements, velocities and accelerations for the selected modes</p> <p>Output of nodal Power Spectral Density (PSD) functions, mean crossing rates and maximum absolute values of PSD and the frequencies at which they occur for the following responses:</p> <ul style="list-style-type: none"> - Displacements, velocities and accelerations - Global stresses for continuum elements - Global top and bottom layer stresses for shell elements - Layer stresses and stress resultants for composite elements 	<p>Automatic (3.5.4)</p> <p>Automatic (3.5.4)</p> <p>*RSET (8.8.1) *PSDOUT (8.8.4)</p>

NISA Capabilities

Output Features and Postprocessing

Output Item	Requested By ^a
<p>Output of PSD functions, mean crossing rates, maximum absolute values and the frequencies at which the maxima occur for the following stress resultants for line elements:</p> <ul style="list-style-type: none">- Axial forces for spars and translational springs- Torsional moments for torsional springs- Beam normal force, shear forces, bending and twisting moments in local coordinate system.	<p>*RSET (8.8.1) *PSDOUT (8.8.4)</p>
<p>Root mean square values (RMS) of the following nodal response quantities:</p> <ul style="list-style-type: none">- Displacements, velocities and accelerations- Reactions forces- Global stresses for continuum elements- Global top and bottom layer stresses for shell elements- Layer stresses and stress resultants for composites <p>RMS element response output for the following line element stress resultants:</p> <ul style="list-style-type: none">- Axial forces for spars and translational springs- Torsional moment for torsional springs- Beam normal force, shear forces, bending and twisting moments in local coordinate system	<p>*RSET (8.8.1) *RMSOUT (8.8.5)</p> <p>*RSET (8.8.1) *RMSOUT (8.8.5)</p>
<p><u>Frequency Response Analysis</u></p> <p>Number of frequency points selected for analysis</p> <p>Modal response amplitudes and phase angles for displacements, velocities and accelerations</p>	<p>Automatic (3.5.5)</p> <p>Executive Command 'MRESPONSE' (8.3.3)</p>

Output Item	Requested By^a
<p>Amplitudes/phase angles or real/imaginary components of spectra, maximum values of amplitudes and the frequencies at which the maxima occur for the following nodal responses:</p> <ul style="list-style-type: none"> - Displacements, velocities and accelerations - Reaction forces - Global stresses for continuum elements - Global top and bottom layer stresses for shell elements - Layer stresses and stress resultants for composite elements 	<p>*RSET (8.8.1) *SPOUT (8.8.6)</p>
<p>Amplitudes/phase angles or real/imaginary components of spectra for the following line element stress resultants:</p> <ul style="list-style-type: none"> - Axial forces for spars and translational springs - Torsional moment for torsional springs - Beam normal force, shear forces, bending and twisting moments in local coordinate system 	<p>*RSET (8.8.1) *SPOUT (8.8.6)</p>
<p><u>Shock Spectrum Analysis</u></p> <p>Generalized (modal) displacements, velocities and accelerations</p> <p>Maximum values of the following nodal responses based on ABS,SRSS, NRL, CQC, GRP, TPM and DSM methods of combination of modal maxima:</p> <ul style="list-style-type: none"> - Displacements, velocities and accelerations - Reaction forces - Elemental centroidal stresses and stress resultants - Global stresses for continuum elements - Global top and bottom layer stresses for shell elements - Layer stresses and stress resultants for composite elements 	<p>Automatic (3.5.6)</p> <p>*RESPONSE (8.8.7)</p>

NISA Capabilities

Output Features and Postprocessing

Output Item	Requested By^a
Maximum value of the following line element stress resultants based on ABS, SRSS, NRL, CQC, GRP, TPM and DSM combination of modal maxima: <ul style="list-style-type: none">- Axial forces for spars and translational springs- Torsional moment for torsional springs- Beam normal force, shear forces, bending and twisting moments in local coordinate system	*RESPONSE (8.8.7)

a. refer to section shown between parentheses for details

Table 3.5: Output description for heat transfer analysis

Output Item	Requested By¹
<u>General output</u>	
Annotated echo of input data	*ECHO (5.2)
Warning and fatal error messages	Automatic (3.12)
Problem parameters (number of elements, active nodes; wavefront statistics)	Automatic (3.13)
Resequenced element table	Executive command 'RESEQUENCE' (5.3.2)
Wavefront history (element-by-element)	Executive command 'EXECUTION' (5.3.2)
CPU time (total and itemized) and elapsed time	Automatic
<u>Nodal output</u>	
Nodal temperatures	
Nodal point heat flow at nodes with specified temperature	
Temperature difference between top and bottom surfaces at the nodes connected to shell elements due to distributed flux, convection or radiation boundary conditions	*PRINTCNTL (7.5.3) *TEMPOUT (7.5.8) ²
<u>Element output</u>	
Heat flux on element boundary faces on which distributed heat flux, convection or radiation boundary conditions are specified	
1: refer to section shown between parentheses for details 2: for transient heat transfer, the output is provided at the time steps specified in *TEMPOUT data group (7.5.8)	

Table 3.6: Postprocessing options for various analysis types

Option Description	Analysis Type ¹		
	ST, NL, BU, EV, LT, NT, DF	HT	Modal Dynamic
<u>Geometry Plots</u>			
Hidden line removal	x	x	x
Boundary and feature line plots	x	x	x
Plotting by model attributes (e.g., material ID number, real constant number)	x	x	x
View manipulation including rotation, scaling and zooming	x	x	x
View and geometry reset and restore options	x	x	x
Element and node numbering and element shrink	x	x	x
<u>Deformed Geometry Plots</u>			
Deformed geometry (displacement or mode shape) plots	x	—	x
Deformed and undeformed geometry (superimposed)	x	—	—
Multiple deformed geometries (deformed history) plots, for nonlinear static analysis only	x	—	x
Animated deformed shapes (on certain graphic devices)	x	—	x
Deformed geometry combined with other options(e.g., combined with hidden line removal, contour plots, etc.)	x	—	x
Deformed shapes for modal dynamic analysis representing: Snapshot (i.e., at a specific instant) or freeze plots for transient analysis	—	—	x
Maximum values displaced shapes for shock spectrum analysis (displaced shape is actually the maximum response envelope).			

Option Description	Analysis Type ¹		
	ST, NL, BU, EV, LT, NT, DF	HT	Modal Dynamic
<u>Contour Plots</u>			
Displacement (or mode shape for eigenvalue analysis) contours	x	—	x
Temperature contours for heat transfer analysis	-	x	—
Contours for stress components, stress intensities(e.g., von Mises, Tresca), stress resultant and layer stresses for composite elements, and strain energy. For eigenvalue and modal dynamic analyses the plotted contour values represent:			
Modal values for eigenvalue analysis	x	—	x
Modal values for eigenvalue analysis			
Snapshot (i.e., at a specific instant) or freeze values for transient analysis			
RMS values for random vibration analysis			
Maximum values for shock spectrum analysis			
Contour plots for cut sections of 3-D models	x	x	x
Control of color band range in terms of contour values or model regions (by zooming)	x	x	x

NISA Capabilities

Output Features and Postprocessing

Option Description	Analysis Type ¹		
	ST, NL, BU, EV, LT, NT, DF	HT	Modal Dynamic
<p><i>XY Plots</i></p> <p>XY history plots of various output quantities with extensive options for number of lines, symbols, scaling, etc. Example of output quantities are: Displacements, stress components and stress intensities Temperature for heat transfer analysis Amplitude and phase spectra of nodal displacements, velocities, accelerations and stresses for frequency response analysis. These are available in linear, semi-log and log-log formats Displacements, velocities, accelerations, stresses and reaction forces representing time histories for transient analysis and PSDs for random vibration analysis. These are available in linear, semi-log and log-log format.</p>	NL, LT NT and DF only	x	x
<p><u>Beam bending moment diagrams and stress computations</u></p> <p>Bending moment and shear force diagrams for general beam element (NKTP = 12)</p> <p>Beam stress computation, stress contour plots and report file generation</p> <p>Beam stress computation and XY plots for</p> <p>Time histories of beam stresses</p> <p>Amplitude and phase spectra for beam stress components</p> <p><u>Image Enhancement/Miscellaneous Plots</u></p> <p>Multiple regions plots (screen splitting)</p> <p>Smooth contour plots</p> <p>Set color for various options and color band</p>	<p>ST only</p> <p>ST only</p> <p>—</p> <p>—</p> <p>x</p> <p>x</p> <p>x</p>	<p>—</p> <p>—</p> <p>—</p> <p>—</p> <p>x</p> <p>x</p> <p>x</p>	<p>TR only</p> <p>TR and SHO</p> <p>TR only</p> <p>FR only</p> <p>x</p> <p>x</p> <p>x</p>

Option Description	Analysis Type ¹		
	ST, NL, BU, EV, LT, NT, DF	HT	Modal Dynamic
Set color by model attributes (e.g., material ID, real constant table number, etc.)	x	x	x
1: ST: Static	NL: Nonlinear Static	BU: Buckling	
EV: Eigenvalue	HT: Heat Transfer	LT: Linear direct transient	
X: Option available	NT: Nonlinear direct transient	DF: Direct Frequency Response	
—: Option not applicable or not available			

3.14.3 Saving Element Results for Postprocessing

Most of the NISA results related to postprocessing options in DISPLAY-POST are automatically saved to the post-data file (file 27) during a NISA run. In order to minimize the size of the post-data file, the element stress and strain results are saved only when the user requests it. The *POSTCNTL data group controls the saving of these element results to the post-data file for linear static and buckling analysis (see [Section 7.5.11](#) for details of *POSTCNTL data group). [Table 3.7](#) lists features available for saving the element results.

Table 3.7: Description of selective saving of element results on the post-data file.

Saved Items	Requested By	
	STATIC	BUCKLING
Element Gauss point stresses For continuum and shell elements <ul style="list-style-type: none"> - Global stress components For laminated composite elements <ul style="list-style-type: none"> - Layer stresses in material principal directions For beams, spars, pipe and elbow elements <ul style="list-style-type: none"> - Stress resultants in element local coordinate system for beam, pipe and elbow elements - Axial force for spars 	*POSTCNTL	Automatic
Element nodal stresses For continuum and shell elements <ul style="list-style-type: none"> - Global stress components - Elemental centroidal stresses and stress resultants For Laminated composite elements <ul style="list-style-type: none"> - Layer stresses in material principal directions - Layer failure criteria (Tsai-Wu, Hill-Mises and maximum stress) - Element stress resultants - Interlaminar shear stress (available only for NKTP = 32, NORDR = 2, 3, and 11. Computation should be first requested by using *SFDCOMP data group and saving is activated by *POSTCNTL data). For beams, spars, pipe and elbow <ul style="list-style-type: none"> - Stress resultants in local element coordinate system for beam, pipe and elbow elements - Axial force for spars 	*POSTCNTL	*POSTCNTL
Element nodal strains For continuum and shell elements <ul style="list-style-type: none"> - Global strain components For Laminated composite elements <ul style="list-style-type: none"> - Layer strains in material principal directions 	*POSTCNTL	*POSTCNTL

3.15 NISA Files

NISA uses several secondary storage files for reading and writing data as well as storing analysis information on a temporary or permanent basis. A brief description of the files used in NISA for different analysis types is given in [Table 3.8](#) to [Table 3.10](#)

Table 3.8: NISA files used in STATIC, NLSTATIC, EIGENVALUE, LTRANSIENT, NLTRANSIENT and BUCKLING analysis.

File Code ⁺	Type	Contents
—	Formatted, sequential	User's input data
—	Formatted, sequential	Printout
—	Formatted, sequential	Log file, brief messages for the analysis progress
13	Binary, direct access	Scratch
18	Binary, sequential	Scratch
19	Binary, sequential	Scratch
21	Binary, sequential	Scratch
22	Binary, sequential	Reaction coefficients
23	Binary, direct access	Element mass or geometric stiffness matrices
24	Binary, direct access	Element stiffness matrices
25	Binary, direct access	Element damping matrices
26	Binary, direct access	Basic model, analysis data and restart information
27	Binary, direct access	Analysis results (post data)
28	Binary, direct access	Binary, direct access
29	Binary, sequential	Revised element stiffness matrices
30	Binary, direct access	Decomposed stiffness matrix/Global mass or geometric stiffness matrix
31	Binary, sequential	Revised element mass or geometric stiffness matrices
32	Binary, direct access	Element equation numbers
34	Binary, sequential	Scratch
35	Binary, sequential	Element destination vectors
37	Binary, direct access	Global displacements and load vectors. Element stress, strain and other Gauss point information for NLSTATIC and NLTRANSIENT
61 to 69	Binary, direct access	Same as file 30 (extension of file 30)
⁺ file codes designated by dash (—) are system dependent		

Table 3.9: NISA files used in modal dynamic analysis

File Code⁺	Type	Contents
—	Formatted, sequential	User's input data
—	Formatted, sequential	Printout
—	Formatted, sequential	Log file, brief messages for the analysis progress
18	Formatted, sequential	Punch file for snapshot response output
26	Binary, direct access	Binary, and analysis data
27	Binary, direct access	Analysis results (post data)
28	Binary, direct access	Scratch
30	Binary, direct access	Global mass matrix
32	Binary, direct access	Element equation numbers
35	Binary, sequential	Element destination vectors
⁺ file codes designated by dash (—) are system dependent		

Table 3.10: NISA files used in Heat transfer analysis

File Code⁺	Type	Contents
—	Formatted, sequential	User's input data
—	Formatted, sequential	Printout
—	Formatted, sequential	Log file, brief messages for the analysis progress
23	Binary, sequential	Element coefficient matrices
24	Binary, sequential	Alternate element coefficient matrices
26	Binary, direct access	Basic model and analysis data and analysis results (post data)
27	Binary, sequential	Element load vectors
28	Binary, sequential	Alternate element load vectors
29	Binary, sequential	Revised element matrices
30	Binary, direct access	Decomposed coefficient matrix
31	Binary, sequential	Scratch
32	Binary, direct access	Element assembly information
33	Binary, direct access	Element equation numbers
34	Binary, sequential	Temperature time history data
39	Formatted, sequential	Temperature results
⁺ file codes designated by dash (—) are system dependent		

3.16 Restart Capabilities

NISA offers several types of restarts for structural and heat transfer analyses. Restart option is not currently available for direct frequency response analysis. Modal dynamic analyses may also be considered as restart runs from a preceding eigenvalue run, but these are discussed separately in [Section 3.5](#) and [Chapter 8](#). The restart features provide added flexibility to more experienced users and can be utilized to serve various needs in solving large scale problems. The available restart types together with the restart requirements and procedures are discussed in detail in [Section 5.5](#). A brief account of the capabilities is given here.

A restart run, as the name implies, may be made to utilize some data or results that have been processed or computed in the preceding run, usually a fresh (or scratch) run, provided that certain files have been saved and some requirements are met.

When several analyses or runs are to be performed on the same model, especially large models, to analyze different loading conditions for example, it may be worthwhile to consider saving some processed data or computed results from the first run for use in a subsequent restart run. Furthermore, the results of a previous run may be used to simulate certain conditions in the subsequent restart run. This is the case, for example, in analyzing the vibration characteristics of initially stressed structures (e.g., rotating structures).

The available restart types are summarized in [Table 5.11](#) and [Table 5.12](#) for structural and heat transfer analyses, respectively. Also shown in the tables are the applicable analysis types for the previous and current runs, together with file requirements and brief description of the capabilities.

It should be noted that restart from a restart run is allowed, with the understanding that a previous restart run is considered as the ‘preceding’ run for the subsequent restart-from-a-restart run, and therefore it must follow the requirements of restarts given in [Section 5.5](#). [Figure 3.22](#) gives an example of multiple restarts for structural analysis.

The restart capabilities provide the following features (see [Section 5.5](#) for applicability and limitations):

- Restart from a check run. In this case, the restart run proceeds from an already processed model data and optimized wavefront.

- ❑ Restart from a restart run, refer to [Figure 3.22](#) as an example.
- ❑ Post-solution computations for previous static load cases or vibration modes. This may include reactions, internal force, strain energies, stresses and load combinations. This is especially useful for modal dynamic analyses, wherein the participating modes, and the associated modal quantities to be saved in the eigenvalue run, e.g., modal stresses, are not readily decided upon until the results of the eigenvalue run itself in terms of frequencies, modal masses, participating factors, etc., are examined.
- ❑ Analysis of new load cases in static analysis, possibly with new sets of displacement boundary conditions. Decomposition of the stiffness matrix may be bypassed for new load cases, depending on the similarity in boundary conditions with the last static load case of previous run and the restart options used.
- ❑ Buckling restart from a previous static analysis run, which may have included multiple load cases. In addition, buckling load factors for a set of loads (live loads) may be calculated in the presence of another set of loads held constant (dead loads).
- ❑ Vibration eigenvalue analysis following static or nonlinear static analysis. The latter case may be used to analyze the vibration characteristics of initially stressed structures (prestress eigenvalue restart), for example rotating structures or the effect of membrane forces on the transverse vibration of beams, plates and shells. With regard to rotating (or spinning) structures, both the stiffening and the softening effects of the centrifugal forces are included. The centrifugal (or spin) softening effect arises due to the deformation-dependent nature of the centrifugal forces. As such, an eigenvalue analysis of a cantilever beam with square cross-section, spinning about a transverse axis at the fixed end, will yield lower bending frequencies in the plane of rotation (the lead-lag motion in the plane normal to the spin axis) than those in the out-of-plane direction (the flap mode). The difference in the bending frequencies in the two transverse directions is due to the softening effect induced in the plane of rotation. (For this cited example, the spin softening affects axial modes also.)
- ❑ Continuation of a nonlinear static analysis from the last converged and saved load step of a previous run. This restart option can be used to add new loads, to change event and output parameters, and to restart from the last converged load step if current step does not converge.
- ❑ Continuation of a direct transient dynamic analysis from the last saved time step of a previous run, possibly with the introduction of different step size, new event param-

eters and new load types (displacement boundary conditions cannot be changed, however).

- ❑ Continuation of a transient heat transfer analysis from the ending time of a previous run, possibly with new time step size and new load types (specified temperature data cannot be changed).
- ❑ Steady state heat transfer restart using the conductivity matrices saved in the previous run.

An important rule to note in restarts is that the model definition cannot be changed. This includes element connectivities, nodal coordinates, local coordinate systems, material specifications, and kinematic constraint conditions in terms of rigid elements, MPC equations and coupled displacements and temperatures. Some restart types, e.g., restart 3 for linear static analysis, allow change of boundary conditions (specified displacements) to designate a new loading condition. See [Section 5.5](#) for details.

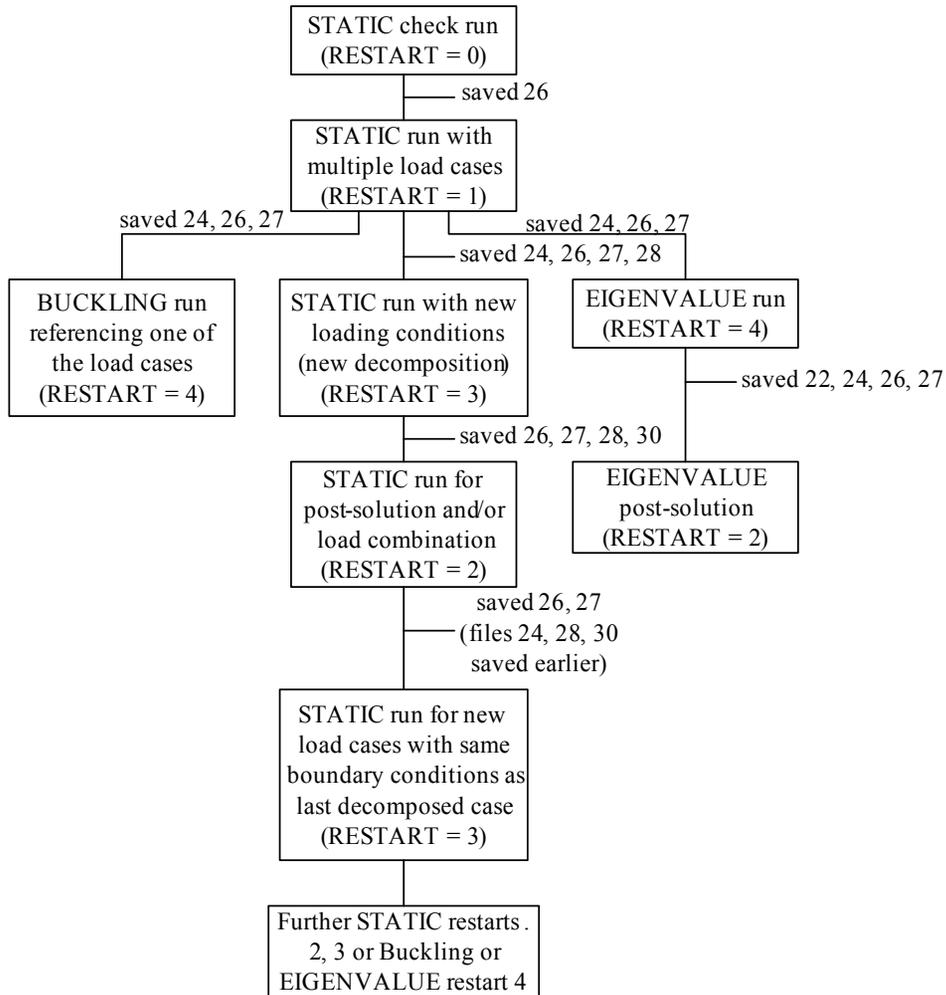


Figure 3.22: An example for multiple structural restarts (files saved include old files automatically saved by program)

3.17 System of Units

NISA is entirely independent of units of the physical quantities specified in the input data. The only requirement is that the units used must be self consistent.

To aid the user in preparing the input, consistent units for some common engineering and physical quantities used in various analysis types are given in [Table 3.11](#) and [Table 3.12](#). Conversion factors for various units are provided in [Table 3.13](#) through [Table 3.18](#).

In these tables, three systems of units are cited, the International System (SI), the metric absolute system (CGS) and the British Gravitational System (BGS).

The SI system is based on the fundamental dimensions of length (L), mass (M) and time (t). Length is measured in meters (m), mass in kilograms (kg) and time in seconds (sec). The program expects, however, the mass and the mass density to be specified in force units (i.e., $F \cdot t^2/L$ and $F \cdot t^2/L^4$, respectively). Therefore, for the user's convenience, [Table 3.11](#) lists force as a fundamental unit, i.e., mass is derived from force through Newton's law $F = Ma$, where a is the gravitational acceleration.

The CGS system (Centimeter, Gram and Seconds) is based on the fundamental dimensions of length (L), mass (M) and time (t). Length is measured in centimeters (cm), mass in grams (gm) and time in seconds (sec). As in the SI units, [Table 3.11](#) lists force as a fundamental unit and mass and mass density are specified in force units (i.e., $F \cdot t^2/L$ and $F \cdot t^2/L^4$, respectively).

The BGS system is based on the fundamental dimensions of length (L), force (F) and time (t). Length is measured in feet (ft), force in pounds (lb_f) and time in seconds (sec).

It should be noted that in [Table 3.12](#), the dimension of heat energy is given the symbol (E) for convenience. The units may be in Joules (J) or British Thermal Units (Btu) which are equivalent to force times length (F.L), see [Table 3.18](#) for conversion factors. Also, time is kept explicit in the units given in [Table 3.12](#) (e.g., J/sec or Btu/sec). Note however, that Watt (W) is defined as the power that produces energy at the rate of one Joule per second, i.e., $W = J/sec$.

To use [Table 3.13](#) through [Table 3.18](#), the units to be converted should be located in the left-hand column whereas the required units should be located in the top row of the table. The conversion factors common to both units should be multiplied by the quantity to be converted. As an example, in [Table 3.13](#) one meter (1m) is equal to 39.37008 inches.

The conversion factors have been carried out to seven significant digits. However, this does not mean the factors are known to this accuracy. Factors given with fewer than seven digits are exact.

To derive conversion factors for quantities not listed in [Table 3.13](#) through [Table 3.19](#), it is always convenient to use “unity factors”. The following two examples explain the use of such factors:

Example 1:

To change the gravitational acceleration from ft/sec² to cm/sec², use the following factors:

$$\begin{aligned}g &= 32.17405 \frac{ft}{sec^2} = 32.17405 \frac{ft}{sec^2} \cdot \left(\frac{sec^2}{cm} \cdot \frac{cm}{sec^2} \right) \\&= 32.17405 \left(\frac{ft}{cm} \cdot \frac{sec^2}{sec^2} \right) \cdot \frac{cm}{sec^2} = 32.17405(30.48) \cdot \frac{cm}{sec^2} \\&= 980.66504 \frac{cm}{sec^2}\end{aligned}$$

See [Table 3.13](#)

Example 2:

To change the Stefan-Boltzmann constant from $W/m^2 \cdot K^4$ to $Btu/hr \cdot ft^2 \cdot ^\circ R^4$, use the following factors:

$$\begin{aligned} \sigma &= 5.67033 \times 10^{-8} \frac{W}{m^2 \cdot K^4} = 5.67033 \times 10^{-8} \frac{J}{\text{sec} \cdot m^2 \cdot K^4} \cdot \left(\frac{hr \cdot ft^2 \cdot ^\circ R^4}{Btu} \cdot \frac{Btu}{hr \cdot ft^2 \cdot ^\circ R^4} \right) \\ &= 5.67033 \times 10^{-8} \left(\frac{J}{Btu} \right) \cdot \left(\frac{hr}{\text{sec}} \right) \cdot \left(\frac{ft^2}{m^2} \right) \cdot \left(\frac{^\circ R^4}{K^4} \right) \cdot \frac{Btu}{hr \cdot ft^2 \cdot ^\circ R^4} \\ &= 5.67033 \times 10^{-8} (9.479735 \times 10^{-4}) (3.6 \times 10^3) (0.3048)^2 \left(\frac{5}{9} \right)^4 \frac{Btu}{hr \cdot ft^2 \cdot ^\circ R^4} \\ &= 0.171257 \times 10^{-8} \frac{Btu}{hr \cdot ft^2 \cdot ^\circ R^4} \end{aligned}$$

See [Table 3.13](#), [Table 3.18](#) and [Table 3.19](#)

Table 3.11: Units for commonly used quantities in static and dynamic analysis⁺

Quantity	Dimensions	Units		
		SI	Metric (CGS)	Imperial (BGS)
<i>Fundamental Quantities</i>				
Force	F	N	dyne	lb _f
Length	L	m	cm	ft
Time	t	sec	sec	sec
Temperature ⁽¹⁾	T	K	K	°R
<u>Derived Quantities</u>				
Acceleration	L/t ²	m/sec ²	cm/sec ²	ft/sec ²
Angle ⁽²⁾	—	rad	rad	rad
Angular acceleration	l/t ²	rad/sec ²	rad/sec ²	rad/sec ²
Angular velocity	l/t	rad/sec	rad/sec	rad/sec
Area moment of inertia	L ⁴	m ⁴	cm ⁴	ft ⁴
Coefficient of thermal expansion ⁽¹⁾	l/T	l/K	l/K	l/°R
Displacement	L	m	cm	ft
Energy	F.L	N.m	dyne.cm	lb _f .ft
Frequency ⁽³⁾	l/t	rad/sec	rad/sec	rad/sec
Mass	F.t ² /L = M	N.sec ² /m = kg	dyne.sec ² /cm = gm _m	lb _f .sec ² /ft = slug
Mass density	F.t ² /L ⁴ = M/L ³	N.sec ² /m ⁴ = kg/m ³	dyne.sec ² /cm ⁴ = gm _m /cm ³	lb _f .sec ² /ft ⁴ = slug/ft ³

Quantity	Dimensions	Units		
		SI	Metric (CGS)	Imperial (BGS)
Mass moment of inertia	$F.t^2.L = M.L^2$	$N.sec^2.m = kg.m^3$	$dyne.sec^2.cm = gm_m.cm^2$	$lb_f.sec^2.ft = slug.ft^2$
Modulus of elasticity	F/L^2	N/m^2 (Pa)	$dyne/cm^2$	lb_f/ft^2
Moment	$F.L$	$N.m$	$dyne/cm$	$lb_f.ft$
Poisson's ratio	—	—	—	—
Pressure	F/L^2	N/m^2 (Pa)	$dyne/cm^2$	lb_f/ft^2
Rotation ⁽²⁾	—	rad	rad	rad
Rotational stiffness	$F.L$	$N.m/rad$	$dyne.cm/rad$	$lb_f.ft/rad$
Strain	—	—	—	—
Stress	F/L^2	N/m^2 (Pa)	$dyne/cm^2$	lb_f/ft^2
Translational stiffness	F/L	N/m	$dyne/cm$	lb_f/ft
Velocity	L/t	m/sec	cm/sec	ft/sec
Work	$F.L$	$N.m$	$dyne.cm$	$lb_f.ft$

[†]applicable notes for [Table 3.11](#) and [Table 3.12](#) are given in the following [Table 3.19](#)

Table 3.12: Unit for commonly used quantities in heat transfer analysis⁺

Quantity	Dimensions	Units		
		SI	Metric(CGS)	Imperial (BGS)
Convective heat transfer coefficient	$E/t.L^2.T$	$J/sec.m^2.K$	$J/sec.cm^2.K$	$Btu/sec.ft^2.°R$
Density ⁽⁴⁾	M/L^3	kg/m^3	gm_m/cm^3	lb_m/ft^3
Heat energy	E (J or F.L)	$J = N.m$	$J = 10^7$ dyne.cm	Btu
Heat flux	$E/t.L^2$	$J/sec.m^2$	$J/sec.cm^2$	$Btu/sec.ft^2$
Heat generation rate	$E/t.L^3$	$J/sec.m^3$	$J/sec.cm^3$	$Btu/sec.ft^3$
Latent heat ⁽⁴⁾	E/M	J/kg	J/gm_m	Btu/lb_m
Specific heat ⁽⁴⁾	$E/M.T$	$J/kg.K$	$J/gm_m.K$	$Btu/lb_m.°R$
Stefan-Boltzmann constant ⁽⁵⁾	$E/t.L^2.T^4$	$J/sec.m^2.K^4$	$J/sec.cm^2.K^4$	$Btu/sec.ft^2.°R^4$
Temperature ⁽¹⁾	T	K	K	°R
Thermal conductivity	$E/t.L.T$	$J/sec.m.K$	$J/sec.cm.K$	$Btu/sec.ft.°R$

⁺ applicable notes are given following [Table 3.19](#)

Table 3.13: Conversion factors for LENGTH

LENGTH	Meter (m)	Centimeter (cm)	Millimeter (mm)	Inch (in)	foot (ft)
Meter m	1.0	$1.0 \times 10^{+2}$	$1.0 \times 10^{+2}$	$3.937008 \times 10^{+1}$	3.280840
Centimeter (cm)	1.0×10^{-2}	1.0	$1.0 \times 10^{+1}$	3.937008×10^{-1}	3.280840×10^{-2}
Millimeter (mm)	1.0×10^{-3}	1.0×10^{-1}	1.0	3.937008×10^{-2}	3.280840×10^{-3}
Inch (in)	2.54×10^{-2}	2.54	$2.54 \times 10^{+1}$	1.0	8.333333×10^{-2}
foot (ft)	3.048×10^{-1}	$3.048 \times 10^{+1}$	$3.048 \times 10^{+2}$	$1.2 \times 10^{+1}$	1.0

+ applicable notes are given following [Table 3.19](#)

Table 3.14: Conversion factors for FORCE

FORCE	Newton N	Dyne Dyne	Kilogramf Kgf	Poundf Lbf	Poundal Pdl
Newton N	1.0	1.0×10^5	1.019716×10^{-1}	2.248089×10^{-1}	7.233015
Dyne dyne	1.0×10^5	1.0	1.019716×10^{-6}	2.24089×10^{-6}	7.233015×10^{-5}
Kilogramf kgf	9.80665	9.80665×10^5	1.0	2.204623	$7.093165 \times 10^{+1}$
Poundf lbf	4.448221	4.448221×10^5	4.535924×10^{-1}	1.0	$3.217405 \times 10^{+1}$
Poundal pdl	1.382549×10^{-1}	1.382549×10^4	1.409808×10^{-2}	3.108095×10^{-2}	1.0

Table 3.15: Conversion factors for MASS

MASS	N.sec²/m (kg)	kg_f.sec²/m	dyne.sec²/cm (gm_m)	lb_f.sec²/ft (slug)	pdl.sec²/ft (lb_m)	lb_f.sec²/in
N.sec ² /m (kg)	1.0	1.019716×10^{-1}	1.0×10^3	6.852178×10^{-2}	2.204623	5.710148×10^{-3}
kg _f .sec ² /m	9.80665	1.0	$9.80665 \times 10^{+3}$	6.719691×10^{-1}	2.161997×10^1	5.599743×10^{-2}
dyne.sec ² /cm (gm _m)	1.0×10^{-3}	1.019716×10^{-4}	1.0	6.852178×10^{-5}	2.204623×10^{-3}	5.710148×10^{-6}
lb _f .sec ² /ft (slug)	1.459389×10^1	1.488162	1.459389×10^4	1.0	$3.217403 \times 10^{+1}$	8.333333×10^{-2}
pdl.sec ² /ft (lb _m)	4.535924×10^{-1}	4.625354×10^{-2}	4.535924×10^2	3.108096×10^{-2}	1.0	2.590081×10^{-3}
lb _f .sec ² /in	1.751268×10^2	$1.785796 \times 10^{+1}$	1.751268×10^5	1.2×10^1	3.860883×10^2	1.0

Table 3.16: Conversion factors for MASS DENSITY

DENSITY	N-sec²/m⁴ (kg/m³)	dyne-sec²/cm⁴ (gm_m/cm³)	kg_f-sec²/m⁴	lb_f-sec²/ft⁴ (slug/ft³)	pdl-sec²/ft⁴ (lb_m/ft³)	lb_f-sec²/in⁴
N-sec ² /m ⁴ (kg/m ³)	1.0	1.0 × 10 ⁻³	1.019716 × 10 ⁻¹	1.940321 × 10 ⁻³	6.242795 × 10 ⁻²	9.357253 × 10 ⁻⁸
dyne-sec ² /cm ⁴ (gm _m /cm ³)	1.0 × 10 ³	1.0	1.019716 × 10 ²	1.940321	6.242795 × 10 ¹	9.357253 × 10 ⁻⁵
kg _f -sec ² /m ⁴	9.80665	9.80665 × 10 ⁻³	1.0	1.902804 × 10 ⁻²	6.122091 × 10 ⁻¹	9.176331 × 10 ⁻⁷
lb _f -sec ² /ft ⁴ (slug/ft ³)	5.153789 × 10 ²	5.153786 × 10 ⁻¹	5.255401 × 10 ⁺¹	1.0	3.217405 × 10 ¹	4.822531 × 10 ⁻⁵
pdl-sec ² /ft ⁴ (lb _m /ft ³)	1.601847 × 10 ¹	1.601847 × 10 ⁻²	1.633429	3.108095 × 10 ⁻²	1.0	1.498889 × 10 ⁻⁶
lb _f -sec ² /in	1.068697 × 10 ⁷	1.068690 × 10 ⁴	1.089761 × 10 ⁶	2.073600 × 10 ⁴	6.671611 × 10 ⁵	1.0

Table 3.17: Conversion factors for PRESSURE

Pressure and Stress	N/m² (Pa)	dyne/cm²	kg_f/m²	lb_f/ft²	lb_f/in² (psi)	pbl/ft²
N/m ² (Pa)	1.0	1.0×10^1	1.019716×10^{-1}	2.088543×10^{-2}	1.450377×10^{-4}	6.719691×10^{-1}
dyne/cm ²	1.0×10^{-1}	1.0	1.019716×10^{-2}	2.088543×10^{-3}	1.450377×10^{-5}	6.719691×10^{-2}
kg _f /m ²	9.80665	9.80665×10^1	1.0	2.048161×10^{-1}	1.422334×10^{-3}	6.589766
lb _f /ft ²	4.788027×10^1	4.788027×10^2	4.882428	1.0	6.944444×10^{-3}	$3.217405 \times 10^{+1}$
lb _f /in ² (psi)	6.894759×10^3	6.894759×10^4	$7.030697 \times 10^{+2}$	$1.440000 \times 10^{+2}$	1.0	$4.633065 \times 10^{+3}$
pbl/ft ²	1.488164	1.488165×10^1	1.517505×10^{-1}	3.108095×10^{-2}	2.158398×10^{-4}	1.0

Table 3.18: Conversion factors for ENERGY

ENERGY	N.m (J)⁺	dyne.cm (erg)	ft.lb_f	ft.pdl	(cal)_{IT}⁺⁺	Btu
N.m (J) ⁺	1.0	1.0×10^7	7.375621×10^{-1}	$2.373036 \times 10^{+1}$	2.388459×10^{-1}	9.478172×10^{-4}
dyne.cm (erg)	1.0×10^{-7}	1.0	7.375621×10^{-8}	2.373036×10^{-6}	2.388459×10^{-8}	9.478172×10^{-11}
ft.lb _f	1.355818	1.355818×10^7	1.0	3.217405×10^1	3.238315×10^{-1}	1.285067×10^{-3}
ft.pdl	4.214011×10^{-2}	$4.214011 \times 10^{+5}$	3.108095×10^{-2}	1.0	1.006499×10^{-2}	3.994110×10^{-5}
(cal) _{IT} ⁺⁺	4.1868	4.1868×10^7	3.088025	9.935427×10^1	1.0	3.968321×10^{-3}
Btu	1.055056×10^3	1.055056×10^{10}	$7.781693 \times 10^{+2}$	2.503687×10^4	2.519958×10^2	1.0

+ 1J = 0.999835 int J

++ (cal)_{IT} indicates international steam table calorie, (cal)_{IT} = 1.000669 (cal), where (cal) indicates thermochemical calorie.

Table 3.19: Conversion *equations* for TEMPERATURE⁺⁺

TEMPERATURE	T(°C)	T(K)	T(°F)	T(°R)
T(°C) =	T(°C)	T(K)-273.16	$\frac{5}{9}(T(^{\circ}F) - 32)$	$\frac{5}{9}(T(^{\circ}R) - 491.68)$
T(K) =	T(°C)+273.16	T(K)	$\frac{5}{9}T(^{\circ}F) - 523.67$	$\frac{5}{9}T(^{\circ}R)$
T(°F) =	$\frac{9}{5}T(^{\circ}C) + 32$	$\frac{9}{5}T(K) - 459.67$	T(°F)	(T(°R) - 459.68)
T(°R) =	$\frac{9}{5}T(^{\circ}C) + 491.68$	$\frac{9}{5}T(K)$	(T(°F) + 459.68)	T(°R)

⁺⁺ Note that conversions of the actual temperature values are governed by the equations given in the table, but the relations

between the intervals of the temperature scales are given by:

$$\begin{aligned}
 1 \text{ degree Celsius} &= 1 \text{ degree Kelvin} \\
 &= (9/5) \text{ degree Fahrenheit} \\
 &= (9/5) \text{ degree Rankine}
 \end{aligned}$$

For example, to change a temperature of 100°C to °F, we use the following equation from the table:

$$\begin{aligned}
 T(^{\circ}F) &= (9/5) T(^{\circ}C) + 32 \\
 &= (9/5)(100) + 32 = 212^{\circ}F
 \end{aligned}$$

However, to change a quantity of 1.0×10^{-6} (1/°C) to (1/°F) we use

$$\begin{aligned}
 1.0 \times 10^{-6} \frac{1}{^{\circ}C} &= 1.0 \times 10^{-6} \frac{1}{^{\circ}C} \left(\frac{^{\circ}F}{^{\circ}F} \right) = 1.0 \times 10^{-6} \left(\frac{^{\circ}F}{^{\circ}C} \right) \frac{1}{^{\circ}F} \\
 &= 1.0 \times 10^{-6} \frac{1}{^{\circ}C} \left(\frac{^{\circ}F}{(9/5)^{\circ}F} \right) \left(\frac{1}{^{\circ}F} \right) = \frac{5}{9} \times 10^{-6} (1/^{\circ}F)
 \end{aligned}$$

Notes:

1. Units of temperature may be interchanged, i.e., degree Kelvin ($^{\circ}\text{K}$) with degree Celsius ($^{\circ}\text{C}$) and degree Rankine ($^{\circ}\text{R}$) with degree Fahrenheit ($^{\circ}\text{F}$), provided that other quantities involving temperature are compatible, and in the case of heat transfer analysis, no radiation boundary conditions are specified.
2. For the user's convenience units of angles are in degrees, for input specification. Rotational components of displacements are input and output in radians.
3. For the user's convenience units of frequency may be in cycles/time for input and output purposes. This will be explicitly stated in the input data group or the printout whenever applicable.
4. For convenience, the units of specific heat and other mass related quantities are specified per unit mass. In this case the units of density should be mass per volume (M/L^3). If weight density is used instead, i.e., units of F/L^3 , the units of the specific heat and other mass related quantities should be per unit weight.
5. If radiation boundary conditions are specified, Stefan-Boltzmann constant as well as all quantities involving degree temperature in their units (e.g., heat transfer coefficient, conductivity, specific heat), have to be specified in terms of absolute degrees, i.e., degree Kelvin ($^{\circ}\text{K}$) or degree Rankine ($^{\circ}\text{R}$).

Element Library

4.1 Introduction

4.1.1 Element Types

A comprehensive library of finite elements is available in NISA. This includes structural and heat transfer elements. Structural elements may be used in linear static (STATIC), nonlinear static (NLSTATIC), buckling (BUCKLING), dynamic analyses (EIGENVALUE, TRANSIENT, FREQUENCY, RANDOM and SHOCK), direct transient dynamic analysis (LTRANSIENT) and direct frequency response analysis (DFREQ). Heat transfer elements may be used in steady state and transient heat transfer analyses (SHEAT and THEAT).

Descriptions of structural elements begin in [Section 4.2](#), with heat transfer elements beginning in [Section 4.39](#). For easier recognition, the numbers assigned to sections, figures and tables in this chapter correspond to the element type designation number (NKTP).

Each element in the library is identified by two variables, NKTP and NORDR. The variable NKTP specifies the element type whereas the variable NORDR specifies the element shape and number of nodes. For example, NKTP = 1 and NORDR = 1 designate a plane stress, 4-node quadrilateral element, whereas NKTP = 1 and NORDR = 2 designate a plane stress, 8-node quadrilateral element. [Table 4.1](#) and [Table 4.2](#) list all structural and heat transfer elements available in the library, in ascending order of the element type number (NKTP). For easier identification, the elements are grouped into different categories according to their usage in [Table 4.3](#) and [Table 4.4](#) which also show the element shapes, degrees of freedom per node and element features. Any admissible combination of element types may

be used to make up the finite element model. The two-dimensional elements must lie in the global XY plane, whereas the three-dimensional elements may be oriented anywhere in space. Axisymmetric elements are considered to be two-dimensional and must lie in the right half of the global XY plane (where X is the radial direction and Y is the axis of revolution). The number of degrees of freedom at a typical node in the model is the union of the degrees of freedom of the elements connected to it.

The available NORDR values for each element type are given in the detailed description of each element. NORDR controls both the element shape functions (linear through cubic) and the geometry of the element (quadrilateral or triangular, with straight or curved sides). Considering all available combinations of NKTP and NORDR, more than one hundred different elements are available in NISA element library.

Unless noted to the contrary, the elements are isoparametric, which means that the same shape functions are used to represent both the element geometry and the displacement (temperature) variation within an element. Elements with mid-side nodes can have curved sides. For example, NORDR = 2 elements use three nodes to define each side. Since a parabolic curve can exactly fit through three points, the side of these elements can exactly match a parabolically curved boundary. Similarly, NORDR = 3 elements provide a cubic fit to curved boundaries.

In general, it is recommended to use parabolic elements (NORDR = 2 and 11) for most problems. Generally, these elements give results of high accuracy and the discretization may involve fewer elements. Some linear elements (NORDR = 1 or 12, see detailed element description) use additional shape functions, associated with nodeless degrees of freedom, in the displacement field within an element to enhance the element response. These elements give results of comparable accuracy to the corresponding parabolic elements at a lesser cost. These elements, however, cannot exactly represent curved edges or surfaces. Quadrilateral (hexahedral) elements generally give more accurate results and converge more rapidly than triangles (wedges). The transition elements (NORDR = 4 to 9) should be avoided or restricted in use to small regions of the model transitioning from, say, parabolic elements (NORDR = 2) to linear elements (NORDR = 1 or 12). The latter may be used to reduce the computational expense in parts of the model where high accuracy is not needed. Appendix A includes some general guidelines for modeling.

Table 4.1: Summary of Structural Elements

Element Type No. (NKTP)	Element Description	Analysis [‡] Types				No. of Nodes (NORDR)	DOF per Node
		ST	DY	BU	NL/NT/ DF		
1	2-D Plane Stress	x	x	x	x	3 to 12	2
2	2-D Plane Strain	x	x	x	x	3 to 12	2
3	Axisymmetric Solid	x	x	x	x	3 to 12	2
4	3-D Solid	x	x	x	x	6 to 20	3
5	3-D Thick Shell	x	x	x	-	6 to 16	3
6	3-D Solid Piezoelectric	x	x	-	-	6 to 20	4
7	3-D Composite Solid	x	x	-	-	8	3
8	3-D Membrane Piezoelectric	x	x	-	-	4	4
9	3-D Hybrid Solid	x	x	x	-	8	3
10	3-D Membrane	-	-	-	x/-	10	3
11	3-D Tapered Beam	x	x	x	-	2	6
12	3-D Beam	x	x	x	-	2	6
13	2-D Beam	x	x	x	-	3	3
14	3-D Spar	x	x	x	x	2	3
15	2-D Spar	x	x	x	-	2	2
17	3-D Translational Spring	x	x	x	x	2	3
18	2-D Translational Spring	x	x	x	-	2	3
20	3-D General Shell	x	x	x	x	3 to 12 ⁺	6
21	3-D Torsional Spring	x	x	x	-	2	3
22	2-D Torsional Spring	x	x	x	-	2	2
25	2-D Point Mass	x	x	x	x	1	2
26	3-D Point Mass	x	x	x	x	1	3
27	2-D General Point Mass	x	x	x	x	1	2
28	3-D General Point Mass	x	x	x	x	1	3
29	2-D Point Mass with Rotary Inertia	x	x	x	x	1	3
30	3-D Point Mass with Rotary Inertia	x	x	x	x	1	6

Element Library

Introduction

Element Type No. (NKTP)	Element Description	Analysis [‡] Types				No. of Nodes (NORDR)	DOF per Node
		ST	DY	BU	NL/NT/DF		
32	3-D Laminated Composite General Shell	x	x	x	x	3-12 ⁺	6
33	3-D Sandwich Shell	x	x	x	-	3 to 12	6
34	Axisym. Solid with General Loading	x	-	-	-	3 to 12	3
36	Axisymmetric Shell	x	x	x	-	2 to 4	3
37	Axisym. Shell with General Loading	x	-	-	-	2 to 4	6
38	3-D General Spring	x	x	x	-	2	6
39	3-D General Beam	x	x	x	x	2	6
40	3-D Thin Shell	x	x	x	x**	3 or 4	6
41	3-D Mixed Interpolation General Shell	x	x	x	-	8	6
42	2-D or Axisymmetric Gap	x	-	-	-	2	2
43	3-D Gap	x	-	-	-	2	3
45	3-D Cable element	-	-	-	x	2	3
46	3-D Straight Pipe Element	x	x	-	-	2	6
47	3-D Elbow Element	x	x	-	-	2	6
48	3-D Damper	-	-	-	x*	2	3
49	2-D Gap/Friction Element	-	-	-	x**	2	2

‡ ST = Static; BU = Buckling; NL = Nonlinear static;

DY = Dynamic (eigenvalue, transient, frequency response, random vibration, shock spectrum and transient dynamic analysis).

NT = Nonlinear Transient DF = Direct Frequency Response

x = Available analysis type; - = Analysis type not available

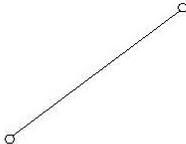
* Available only for direct transient dynamic analysis

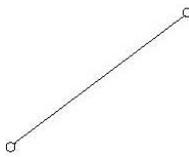
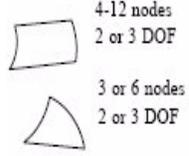
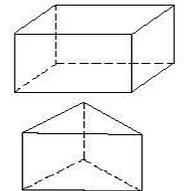
+ Some NORDR may not be available in NL

Table 4.2: Summary of Heat Transfer Elements

Element Type No. (NKTP)	Element Description	No. of Nodes	DOF per Nodes
101	2-D Planar Element for Heat Transfer	3 to 12	1
102	2-D Planar Element of unit thickness for Heat Transfer	3 to 12	1
103	Axisymmetric Solid Element for Heat Transfer	3 to 12	1
104	3-D Solid Element for Heat Transfer	6 to 20	1
105	3-D Thick Shell Element for Heat Transfer	6 to 16	1
112	3-D Bar Element for Heat Transfer	2	1
120	3-D Curved Thin Shell Element	3 to 12	1
149	3-D Convection Link Element	2	1
150	3-D Radiation Link Element	2	1

Table 4.3: Structural element classification

	Shape, DOF / node	NKTP and NORDR	Features
Point mass	1 node up to 6 DOF 	NKTP = 25-30 NORDR = 1	<ul style="list-style-type: none"> - Concentrated mass at a point with or without rotary inertia - Can be used in nonlinear analysis
Spar, spring and cable elements	 2 node 2, 3 or 6 DOF	NKTP = 14, 15, 17, 18, 21, 22, 38,45 NORDR = 1	<ul style="list-style-type: none"> - 2-D or 3-D line element - Up to 6 independent spring rates - Nodes may be coincident (springs only) - Initial tension and no compression capabilities (cables only) - No tension capabilities for NKTP = 45 - Geometric and material nonlinearities for NKTP = 14, 17, 45 - frequency dependent properties for NKTP=17 and NKTP=21

	Shape, DOF / node	NKTP and NORDR	Features
Beam elements	 <p>2 node 3 or 6 DOF</p>	<p>NKTP = 11, 12, 13, 39 NORDR = 1</p>	<ul style="list-style-type: none"> - 2-D or 3-D beam - Stretching, bending and torsional capabilities. - Tapered beam - Shear center eccentricity, offset, end release - Transverse shear effect - Standard and general sections for NKTP = 39 - Geometric and material nonlinearities for NKTP = 39
2-D and axisymmetric solid elements	 <p>4-12 nodes 2 or 3 DOF</p> <p>3 or 6 nodes 2 or 3 DOF</p>	<p>NKTP = 1, 2, 3, 34 NORDR = 1-12</p>	<ul style="list-style-type: none"> - Plane stress, plane strain, axisymmetric solid - Bubble function for low order elements - Axisymmetric solid with non-axisymmetric loading - Geometric and material nonlinearities
3-D solid elements	 <p>8-20 node 3 DOF</p> <p>6 or 15node 3 DOF</p>	<p>NKTP = 4, 5, 7, 9 NORDR = 1-2, 7-12</p>	<ul style="list-style-type: none"> - 3-D solid, thick shell - 3-D layered composite solid. - Hybrid solid (NKTP = 9) can model thin shells - Bubble function for low order elements - Geometric and material nonlinearities (NKTP = 4)

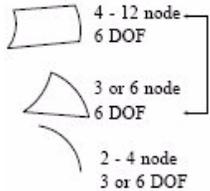
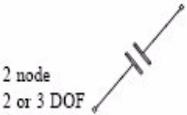
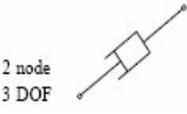
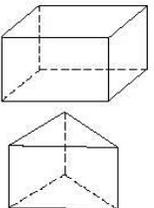
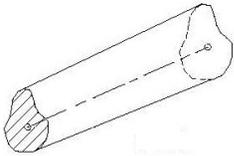
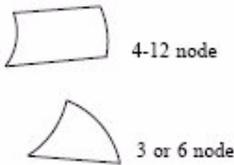
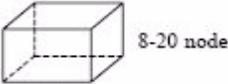
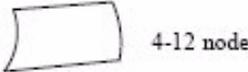
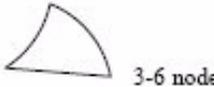
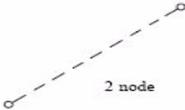
	Shape, DOF / node	NKTP and NORDR	Features
Plate and shell elements	 <p>4 - 12 node 6 DOF</p> <p>3 or 6 node 6 DOF</p> <p>2 - 4 node 3 or 6 DOF</p>	<p>NKTP = 20, 32, 33,40, 41 NORDR = 1 - 11 NKTP = 36, 37 NORDR = 1-3</p>	<ul style="list-style-type: none"> - Moderately thick to thin shells - Laminated composite and sandwich shells - Axisymmetric shell with axisymmetric or non-axisymmetric loading - Membrane, bending and transverse shear effects - Geometric and material nonlinearities for NKTP = 20, 32 - Special treatment for membrane and shear locking in linear and nonlinear analyses
Gap elements	 <p>2 node 2 or 3 DOF</p>	<p>NKTP = 42, 43, 49, 50 NORDR = 1</p>	<ul style="list-style-type: none"> - 2-D, axisymmetric or 3-D gap - Tension-only or compression-only member - Nodes may be coincident (2-D only)
Dash pot elements	 <p>2 node 3 DOF</p>	<p>NKTP = 48 NORDR = 1</p>	<ul style="list-style-type: none"> - 3-D, damping element - One damping coefficient - Nodes may be coincident
Membrane elements	 <p>3 node 3 DOF</p>	<p>NKTP = 10 NORDR = 10</p>	<ul style="list-style-type: none"> - Membrane effect - Geometric nonlinearities
Piezoelectric elements		<p>NKTP = 6, 8 NORDR = 1-2, 7-12</p>	<ul style="list-style-type: none"> - Piezoelectric effect

Table 4.4: Heat transfer element classification

	SHAPE, (one DOF/node, temp')	NKTP and NORDR	Features
3-D bar element	 <p>2 node</p>	NKTP = 112 NORDR = 1	<ul style="list-style-type: none"> - Steady state and transient heat transfer with nonlinear capabilities - Analogous field problems, e.g., diffusion, seepage, magnetostatics, and torsion - Orthotropic material properties
2-D and axisymmetric solid elements	 <p>4-12 node</p> <p>3 or 6 node</p>	NKTP = 101, 102, 103 NORDR = 1-11	<ul style="list-style-type: none"> - Temperature dependent material properties - Nonlinear radiation and convection boundary conditions with temperature dependent emissivity and film coefficients - Time dependent heat flux and heat generation rates - Time dependent film coefficient and emissivity - Time dependent ambient temperature - Phase change

	SHAPE, (one DOF/node, temp')	NKTP and NORDR	Features
3-D solid elements	 	NKTP = 104, 105 NORDR = 1-2, 7-11	
Plate and shell elements	 	NKTP = 120 NORDR = 1-11	
3-D convection and radiation link elements		NKTP = 149, 150 NORDR = 1	

4.1.2 Element Input

Element input definition includes specifying element connectivities, material properties, geometrical properties (e.g., real constants, beam cross section orientation), as well as specifying special element data, (e.g., fiber orientation and lamination sequence for composites). The common data groups that specify most of the above input are listed in [Table 4.5](#), which also shows the requirement of each data group for specific element types (NKTP). Other data groups that specify element loading are mentioned explicitly in the description of each element. These data groups (specifying element loading) are detailed in [Chapter 7](#) and [Chapter 8](#).

Table 4.5: Data groups for element definition

Group ID Name	Section No.	Purpose	Requirement (Required for)
<u>Element Selection and Definition</u>			
*ELTYPE	6.3.1	Select element types used in the model	Always
*ELEMENTS	6.3.2	Define element connectivities and identification numbers (pointers) for element geometric and material properties.	Always
<u>Element Attributes</u>			
*RCTABLE	6.3.4	Define element real constants, e.g. shell thicknesses, beam section properties, spring rates, etc.	All elements except NKTP = 2-5, 9, 34, 102-105
*LAMANGLE	6.3.5	Specify fiber orientation angles for composites	NKTP = 7, 32, 33
*LAMSQ2	6.3.6	Define lamination sequence data for composites	NKTP = 7, 32, 33
*VECTORS	6.7.2	Define cross section orientation vector for 3-D beams and springs	NKTP = 11 optional for NKTP =12, 38
*BMDATA	6.3.7	Specify general data for general beam element	NKTP = 39
*BMSECT	6.3.8	Specify cross-sectional data for general beam element	NKTP = 39
*NLSPRING	6.3.9	Specify force-deflection data for nonlinear springs	NKTP = 17

Group ID Name	Section No.	Purpose	Requirement (Required for)
<u>Element Material Properties</u>			
*MATERIAL	6.5.1	Specify elastic material properties	All elements except NKTP = 17, 18, 21, 22, 25-30, 38, 42, 43, 48, 149, 150
*MATDIR1	6.5.2	Specify orthotropic material axes at nodes	Optional for NKTP = 1-5, 9, 20, 40, 41, 101-106
*MATDIR2	6.5.3	Specify orthotropic material axes at elements	Optional for NKTP = 1-5, 9, 20, 40, 41, 101-106
*MATHEAT	6.5.4	Specify material properties for heat transfer	NKTP = 101-106
*HYPERE	6.5.5	Specify hyperelastic material properties	Optional for NKTP = 1-4
*PLASTIC	6.5.6	Specify plasticity part of elastoplastic material	Optional for NKTP = 1-4, 14, 20, 45
*APLASTIC	6.5.7	Specify anisotropic plastic material properties	Optional for NKTP = 32
*CREEP	6.5.8	Specify creep material law	Optional for NKTP = 1-4, 14, 20, 39

4.1.3 Element Output

The general output from NISA consists of both nodal and element output. Nodal output includes displacements, reactions, temperatures, etc. Element output for structural elements includes:

1. Element internal forces: These are the forces acting at the element vertices, in a free state, to hold the element in equilibrium (i.e., to balance both the element loads and the forces that result from the change in the element configuration). The components of internal forces correspond to the degrees of freedom associated with a typical element. For example, there are three components (FX, FY, FZ) of internal forces for the 3-D solid element, whereas the corresponding internal forces for a 2-D beam element

(NKTP = 13) are (FX, FY, MZ). The internal force output at a typical node is always in global coordinates unless this node has a local displacement coordinate system.

2. Element strain energy: This output is provided for elements which possess structural stiffness. The output includes also the strain energy density (strain energy per unit volume) and the percentage of the total strain energy of all elements.
3. Element stress output: This output varies according to the element type. In general it includes the components of stresses in local and/or global directions at the element centroid, Gauss points and/or nodal points for continuum and shell elements. For line elements, it consists of stress resultants (e.g., normal force, shear forces and bending moments for beam elements). Detailed information about the element stress output is given in the description of each element. The centroidal strains are also available for output for continuum and shell elements. In addition to the components of stresses, the following stress-related quantities are available.

Principal stresses (S_1, S_2, S_3), ($S_1 \geq S_2 \geq S_3$)

Maximum shear stress

$$S_{\max} = \max\left[\frac{1}{2}|S_1 - S_2|, \frac{1}{2}|S_2 - S_3|, \frac{1}{2}|S_3 - S_1|\right]$$

von Mises equivalent stress

$$S_{eq} = \frac{1}{\sqrt{2}}[(S_1 - S_2)^2 + (S_2 - S_3)^2 + (S_3 - S_1)^2]^{1/2}$$

Octahedral shear stress

$$S_{oct} = \frac{1}{3}[(S_1 - S_2)^2 + (S_2 - S_3)^2 + (S_3 - S_1)^2]^{1/2}$$

In addition to the element stress output, NISA computes the stresses at the nodes by averaging the element stresses at common nodes between elements. Principal stresses, maximum shear stress, von Mises equivalent stress and the octahedral shear stress are then computed at the nodes. For models having shell elements, NISA will also compute the averaged nodal stresses for the top and bottom surfaces (in addition to the middle surface). Averaged stress resultants are computed for the laminated composite and laminated sandwich shell elements.

All of the element output options listed above are also written to the NISA postprocessing file which may be saved for further use. This postprocessing file may be used for

restarts or for graphical representation of the results using the postprocessing module of the DISPLAY program.

The element output related to heat transfer elements is given along with the detailed description of each element.

4.2 2-D Plane Stress Element (NKTP = 1)

Description

This element is based on the assumption of a 2-D state of stress and is suited for modeling thin flat structures lying in the global XY plane and subjected to in-plane loading. The element has two degrees of freedom per node (UX, UY). The state of stress is characterized by three components (SXX, SYY, SXY). The theoretical basis of the element is discussed in Section 2.8. An element reference guide is given in Table 4.6 with the available loading given in Table 4.7.

The element can be shaped as a 4 to 12 node quadrilateral, or a 3 or 6 node triangle depending on the selected NORDR value. The element configuration, node locations and face numbering convention are shown in Figure 4.1. A general description of the element input data and output options are given in Section 4.1.2 and Section 4.1.3, respectively. Table 4.6 gives the output pertinent to this element.

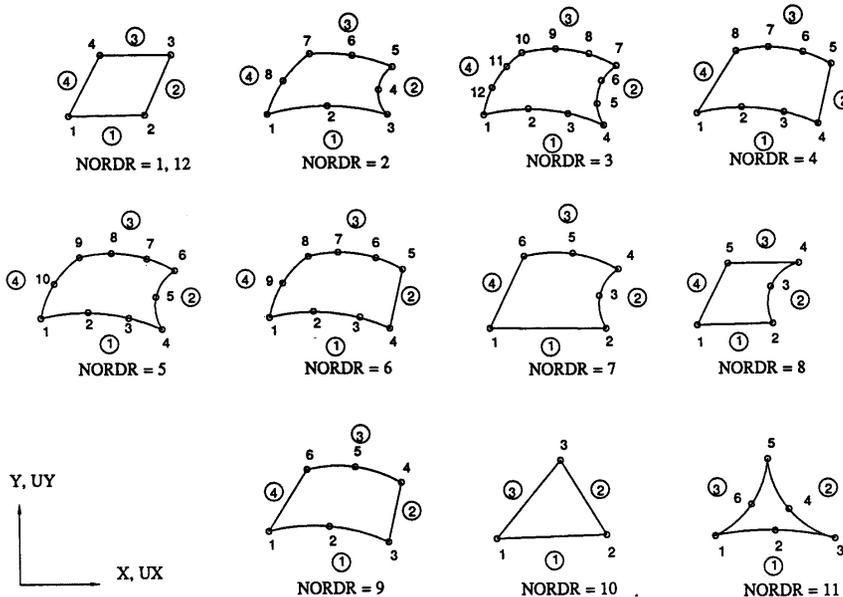


Figure 4.1: Element configuration for available NORDR values, NKTP = 1 (circled numbers indicate face numbers)

Table 4.6: Element reference guide (NKTP = 1)

Element Type	NKTP = 1, 2-D Plane stress element
Analysis Types	Static, Dynamic, Buckling, Nonlinear Static, Nonlinear Transient
Degrees of Freedom	2 per node: UX, UY
NORDR	Quadrilateral: 4 to 12 nodes (NORDR = 1-9, 12), see note 2
(Shape / No. of nodes)	Triangle: 3 or 6 nodes (NORDR = 10, 11)
Real Constants	3 to 12 nodal thicknesses (same as number of nodes)
Material Properties	
Isotropic Elastic	4 properties: EX, NUXY, DENS, ALPX
Orthotropic Elastic	10 properties: EX, EY, NUXY, NUYZ, NUXZ, GXY, DENS, ALPX, ALPY, ALPZ, see note 3
Elastoplastic	For nonlinear static only, see *PLASTIC
Hyperelastic	For nonlinear static only, see *HYPEREL
Creep Law	For nonlinear static only, see *CREEP
Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY), and strain energy - Centroidal strains (EXX, EYY, EZZ, EXY), in principal material directions and/or global directions, see note 4 - Element stresses (SXX, SY, SXY) at centroid, Gauss, and nodal points in principal material directions and/or global directions - Element principal stresses (S1, S2, S3), von Mises equivalent stress, octahedral and maximum shear stresses at centroid, Gauss, and nodal points - Equivalent plastic strain (EPS), and effective yield stress (YLD), for nonlinear static only, at Gauss and nodal points, see note 5 - Second Piola-Kirchhoff or Cauchy stresses, for nonlinear static only, at Gauss and nodal points
Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration, and shock spectrum analyses

Element Library

2-D Plane Stress Element (NKTP = 1)

Nonlinear Capabilities	<ul style="list-style-type: none">- Geometric nonlinearity: Large displacements, rotations, and strains (for plasticity analysis, only small strains are assumed); total and updated Lagrangian formulations; deformation dependent loads- Material nonlinearity: elastoplastic and hyperelastic material models- Bubble function for 4-node element (NORDR = 12)
-------------------------------	---

Table 4.7: Available loading (NKTP = 1)

Nodal Loading	<ul style="list-style-type: none">- Concentrated nodal forces in global X and Y directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE- Specified non-zero nodal displacements UX, UY in global X and Y directions (or in local displacement system, if defined), see *SPDISP- Concentrated nodal follower (deformation dependent) forces perpendicular to a reference face of the element, for nonlinear static only, see *CFOLLOWER- Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
Pressure Loading	<ul style="list-style-type: none">- Uniform or non-uniform pressure (force/area) on any face of the element, see *PRESSURE or *DPRESSURE- Uniform or non-uniform follower (deformation dependent) pressure (force/area) on any face of the element, for nonlinear static only, see *PRESSURE
Thermal Loading	<ul style="list-style-type: none">- Specified nodal temperature, see *NDTEMPER
Body Force	<ul style="list-style-type: none">- Gravity loading or body forces due to linear acceleration in global X and Y directions, see *BODYFORCE- Centrifugal loads due to angular velocity about the global X, Y and Z direction, see *BODYFORCE and note 6- Tangential loads due to angular acceleration about the global Z direction, see *BODYFORCE
Ground Motion	<ul style="list-style-type: none">- Forces due to ground motion (support excitation) in the global X and Y directions, see *GROUND- Forces due to rotational ground motion in the global XY plane about a specified point, see *GROUND

Notes:

1. The element connectivity must be given in the order shown in [Figure 4.1](#), in which the node numbering sequence starts at a corner node and proceeds along the perimeter of the element in a counterclockwise direction.
2. The linear quadrilateral elements (NORDR = 1, 12) have the same configuration and they use the same bilinear shape functions to represent the element geometry. However, in addition to the bilinear shape functions used to represent the displacement variation for NORDR = 1 element, the NORDR = 12 element uses incompatible shape functions (bubble functions) associated with two internal (nodeless) degrees of freedom for both linear and nonlinear static analyses. These extra shape functions enable the NORDR = 12 element to model pure bending exactly. See [Section 2.8](#) for details.
3. For orthotropic elastic material property input, the principal material axes must be confined to the global XY plane. The NUYZ and NUXZ values are needed for the calculation of EZZ for the purpose of printout. In addition, ALPZ is required only if thermal loading exists to calculate the thermal strain in the Z-direction.
4. Centroidal strains output is not available for nonlinear static analysis.
5. For nonlinear static analysis, the nodal equivalent plastic strains, EPS are approximated by extrapolating the Gauss point equivalent plastic strains.
6. For centrifugal loads, the global axes are assumed to be attached to and rotating with the structure.
7. The pressure loading should be specified in units of force/area.

4.3 2-D Plane Strain Element (NKTP = 2)

Description

The element is based on the assumption of a 2-D state of strain and is suited for modeling thick flat structures lying in the global XY plane and subjected to in-plane loading. The element has two degrees of freedom per node (UX, UY). The state of stress is characterized by four components (SXX, SYY, SZZ, SXY). A unit thickness is assumed for this element. The theoretical basis of the element is discussed in Section 2.8. An element reference guide is given in Table 4.8 with the available loading given in Table 4.9.

The element can be shaped as a 4 to 12 node quadrilateral, or a 3 or 6 node triangle depending on the selected NORDR value. The element configuration, node locations and face numbering convention are shown in Figure 4.2. A general description of the element input data and output options are given in Section 4.1.2 and Section 4.1.3, respectively. Table 4.8 gives the output pertinent to this element.

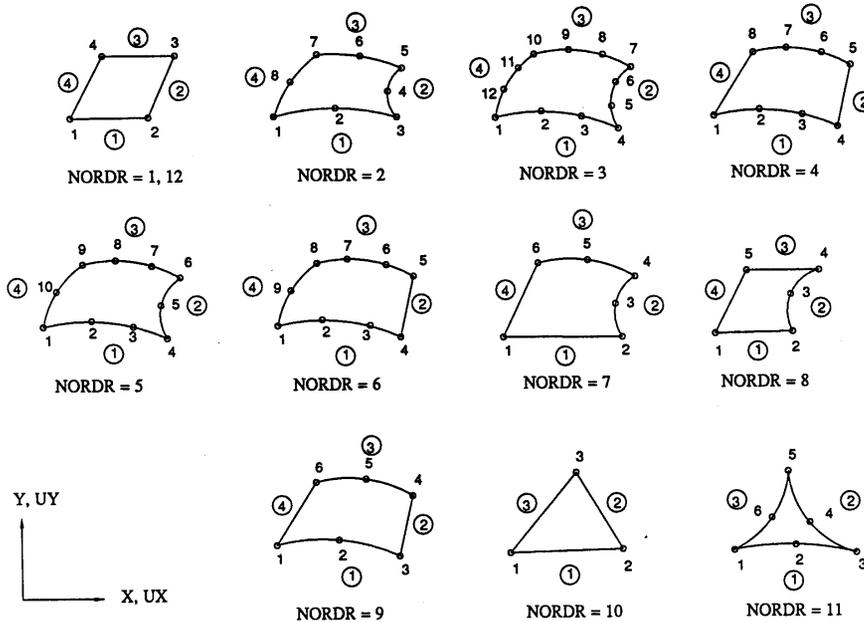


Figure 4.2: Element configuration for available NORDR values, NKTP = 2 (circled numbers indicate face numbers)

Table 4.8: Element reference guide (NKTP = 2)

Element Type	NKTP = 2, 2-D Plane strain element
Analysis Types	Static, Dynamic, Buckling, Nonlinear Static, Nonlinear Transient
Degrees of Freedom	2 per node: UX, UY
NORDR	- Quadrilateral: 4 to 12 nodes (NORDR = 1-9, 12), see note 2
(Shape / No. of nodes)	- Triangle: 3 or 6 nodes (NORDR = 10, 11)
Real Constants	None, unit thickness is assumed
Material Properties	
Isotropic Elastic	4 properties: EX, NUXY, DENS, ALPX
Orthotropic Elastic	11 properties: EX, EY, EZ, NUXY, NUYZ, NUXZ, GXY, DENS, ALPX, ALPY, ALPZ, see note 3
Elastoplastic	For nonlinear static only, see *PLASTIC
Hyperelastic	For nonlinear static only, see *HYPEREL
Creep Law	For nonlinear static only, see *CREEP
Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY), and strain energy - Centroidal strains (EXX, EYY, EXY), in principal material directions and/or global directions, see note 4 - Element stresses (SXX, SYY, SZZ, SXY) at centroid, Gauss, and nodal points in principal material directions and/or global directions - Element principal stresses (S1, S2, S3), von Mises equivalent stress, octahedral and maximum shear stresses at centroid, Gauss, and nodal points - Equivalent plastic strain (EPS), and effective yield stress (YLD), for nonlinear static only, at Gauss and nodal points, see note 5 - Second Piola-Kirchhoff or Cauchy stresses, for nonlinear static only, at Gauss and nodal points
Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration, and shock spectrum analyses

Element Library

2-D Plane Strain Element (NKTP = 2)

Nonlinear Capabilities	<ul style="list-style-type: none">- Geometric nonlinearity: Large displacements, rotations, and strains (for plasticity analysis, only small strains are assumed); total and updated Lagrangian formulations; deformation dependent loads- Material nonlinearity: elastoplastic and hyperelastic material models- Bubble function for 4-node element (NORDR = 12)
-------------------------------	---

Table 4.9: Available loading (NKTP=2)

Nodal Loading	<ul style="list-style-type: none">- Concentrated nodal forces in global X and Y directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE- Specified non-zero nodal displacements UX, UY in global X and Y directions (or in local displacement system, if defined), see *SPDISP- Concentrated nodal follower (deformation dependent) forces perpendicular to a reference face of the element, for nonlinear static only, see *CFOLLOWER- Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
Pressure Loading	<ul style="list-style-type: none">- Uniform or non-uniform pressure (force/area) on any face of the element, see *PRESSURE or *DPRESSURE- Uniform or non-uniform follower (deformation dependent) pressure (force/area) on any face of the element, for nonlinear static only, see *PRESSURE
Thermal Loading	<ul style="list-style-type: none">- Specified nodal temperature, see *NDTEMPER
Body Force	<ul style="list-style-type: none">- Gravity loading or body forces due to linear acceleration in global X and Y directions, see *BODYFORCE- Centrifugal loads due to angular velocity about the global X, Y and Z direction, see *BODYFORCE and note 6- Tangential loads due to angular acceleration about the global Z direction, see *BODYFORCE
Ground Motion	<ul style="list-style-type: none">- Forces due to ground motion (support excitation) in the global X and Y directions, see *GROUND- Forces due to rotational ground motion in the global XY plane about a specified point, see *GROUND

Notes:

1. The element connectivity must be given in the order shown in [Figure 4.2](#), in which the node numbering sequence starts at a corner node and proceeds along the perimeter of the element in a counterclockwise direction.
2. The linear quadrilateral elements (NORDR = 1, 12) have the same configuration and they use the same bilinear shape functions to represent the element geometry. However, in addition to the bilinear shape functions used to represent the displacement variation for NORDR = 1 element, the NORDR = 12 element uses incompatible shape functions (bubble functions) associated with two internal (nodeless) degrees of freedom for both linear and nonlinear static analyses. These extra shape functions enable the NORDR = 12 element to model pure bending exactly. See [Section 2.8](#) for details.
3. For orthotropic elastic material property input, the principal material axes must be confined to the global XY plane. The values of EZ, NUYZ, and NUXZ are needed to calculate the stress in the Z direction, SZZ. In addition, ALPZ is required only if thermal loading exists to calculate the thermal strain in the Z-direction.
4. Centroidal strains output is not available for nonlinear static analysis.
5. For nonlinear static analysis, the nodal equivalent plastic strains, EPS are approximated by extrapolating the Gauss point equivalent plastic strains.
6. For centrifugal loads, the global axes are assumed to be attached to and rotating with the structure.
7. The element is developed assuming a unit thickness. Therefore, concentrated loads should be entered per unit of depth. Reaction forces and strain energy are also per unit of depth. The pressure loading is in units of force/area.

4.4 Axisymmetric Solid Element (NKTP = 3)

Description

This element is based on the assumption that its geometry, material properties and loadings are rotationally symmetric about an axis. The element must lie in the right half of the global XY plane where global X-axis is the radial direction, and global Y-axis is the axis of revolution or axis of symmetry. The element has two degrees of freedom per node (UX, UY). The state of stress is characterized by four components (SXX, SYY, SZZ, SXY). The theoretical basis of the element is discussed in Section 2.8. An element reference guide is given in Table 4.10 with the available loading given in Table 4.11.

The element can be shaped as a 4 to 12 node quadrilateral, or a 3 or 6 node triangle depending on the selected NORDR value. The element configuration, node locations and face numbering convention are shown in Figure 4.3. A general description of the element input data and output options are given in Section 4.1.2 and Section 4.1.3, respectively. Table 4.10 gives the output pertinent to this element.

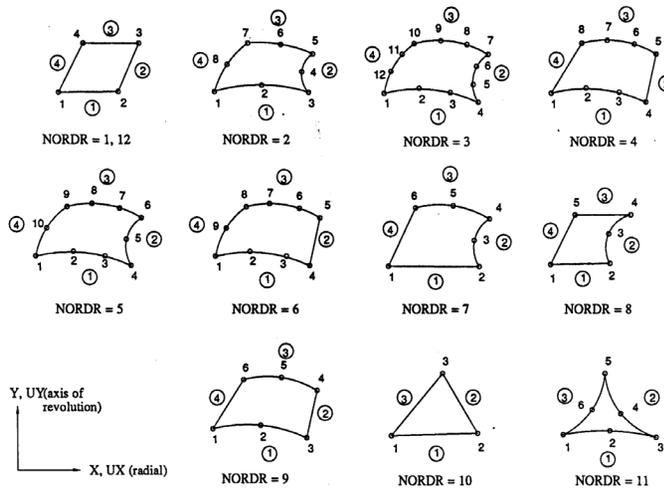


Figure 4.3: Element configuration for available NORDR values, NKTP = 3 (circled numbers indicate face numbers)

Table 4.10: Element reference guide (NKTP = 3)

Element Type	NKTP = 3, Axisymmetric solid element
Analysis Types	Static, Dynamic, Buckling, Nonlinear Static, Nonlinear Transient
Degrees of Freedom	2 per node: UX, UY
NORDR (Shape / No. of nodes)	<ul style="list-style-type: none"> - Quadrilateral: 4 to 12 nodes (NORDR = 1-9, 12), see note 2 - Triangle: 3 or 6 nodes (NORDR = 10, 11)
Real Constants	None
Material Properties	
Isotropic Elastic	4 properties: EX, NUXY, DENS, ALPX
Orthotropic Elastic	11 properties: EX, EY, EZ, NUXY, NUYZ, NUXZ, GXY, DENS, ALPX, ALPY, ALPZ, see note 3
Elastoplastic	For nonlinear static only, see *PLASTIC
Hyperelastic	For nonlinear static only, see *HYPEREL
Creep Law	For nonlinear static only, see *CREEP
Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY), and strain energy - Centroidal strains (EXX, EYY, EZZ, EXY), in principal material directions and/or global directions, see note 4 - Element stresses (SXX, SY, SZZ, SXY) at centroid, Gauss, and nodal points in principal material directions and/or global directions - Element principal stresses (S1, S2, S3), von Mises equivalent stress, octahedral and maximum shear stresses at centroid, Gauss, and nodal points - Equivalent plastic strain (EPS), and effective yield stress (YLD), for nonlinear static only, at Gauss and nodal points, see note 5 - Second Piola-Kirchhoff or Cauchy stresses, for nonlinear static only, at Gauss and nodal points
Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration, and shock spectrum analyses

Element Library

Axisymmetric Solid Element (NKTP = 3)

Nonlinear Capabilities	<ul style="list-style-type: none">- Geometric nonlinearity: Large displacements, rotations, and strains (for plasticity analysis, only small strains are assumed); total and updated Lagrangian formulations; deformation dependent loads- Material nonlinearity: elastoplastic and hyperelastic material models- Bubble function for 4-node element (NORDR = 12)
-------------------------------	---

Table 4.11: Available loading (NKTP = 3)

Nodal Loading	<ul style="list-style-type: none">- Concentrated nodal forces in global X and Y directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE and note 6- Specified non-zero nodal displacements UX, UY in global X and Y directions (or in local displacement system, if defined), see *SPDISP- Concentrated nodal follower (deformation dependent) forces perpendicular to a reference face of the element, for nonlinear static only, see *CFOLLOWER and note 6- Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER and note 6
Pressure Loading	<ul style="list-style-type: none">- Uniform or non-uniform pressure (force/area) on any face of the element, see *PRESSURE or *DPRESSURE and note 7- Uniform or non-uniform follower (deformation dependent) pressure (force/area) on any face of the element, for nonlinear static only, see *PRESSURE and note 7
Thermal Loading	<ul style="list-style-type: none">- Specified nodal temperature, see *NDTEMPER
Body Force	<ul style="list-style-type: none">- Gravity loading or body forces due to linear acceleration in global X and Y directions, see *BODYFORCE and note 8- Centrifugal loads due to angular velocity about the global Y axis, see *BODYFORCE
Ground Motion	<ul style="list-style-type: none">- Forces due to ground motion (support excitation) in the global X and Y directions, see *GROUND and note 8

Notes:

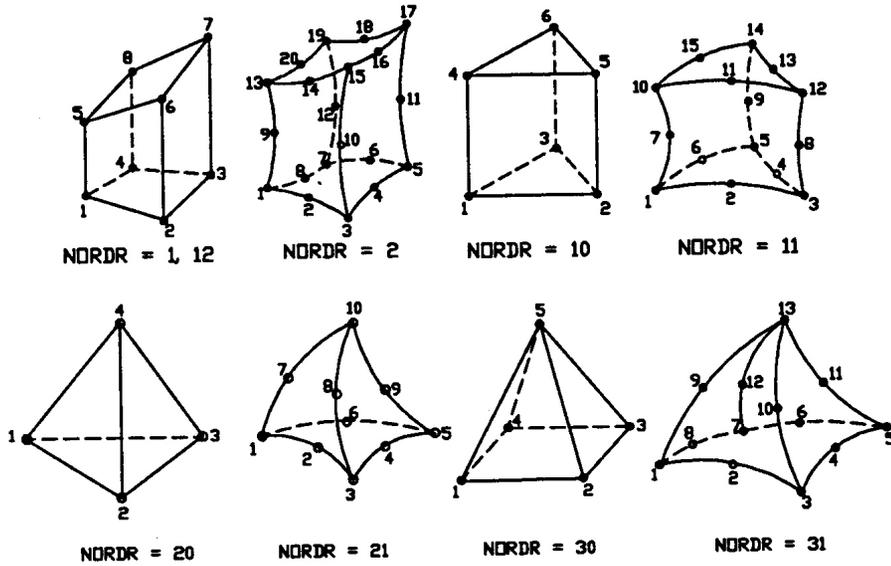
1. The element connectivity must be given in the order shown in [Figure 4.3](#), in which the node numbering sequence starts at a corner node and proceeds along the perimeter of the element in a counterclockwise direction.
2. The linear quadrilateral elements (NORDR = 1, 12) have the same configuration and they use the same bilinear shape functions to represent the element geometry. However, in addition to the bilinear shape functions used to represent the displacement variation for NORDR = 1 element, the NORDR = 12 element uses incompatible shape functions (bubble functions) associated with two internal (nodeless) degrees of freedom for both linear and nonlinear static analyses. These extra shape functions improve the behavior of the NORDR = 12 element in bending. See [Section 2.8](#) for details.
3. For orthotropic elastic material property input, the principal material axes must be confined to the global XY plane.
4. Element internal forces, element strain energy, and centroidal strains output is not available for nonlinear static analysis.
5. For nonlinear static analysis, the nodal equivalent plastic strains, EPS are approximated by extrapolating the Gauss point equivalent plastic strains.
6. Concentrated loads should be specified for the entire 360 degree circumference of the model. The internal force and reaction outputs are also for the entire 360 degree circumference of the model.
7. Pressure loading is input as force/unit area.
8. Caution must be taken that the global X direction is actually the radial direction.

4.5 3-D Solid Element (NKTP = 4)

Description

This element is based on a general 3-D state of stress and is suited for modeling 3-D solid structure under 3-D loading. The element has three degrees of freedom per node (UX, UY, UZ). The state of stress is characterized by six components (SXX, SYY, SZZ, SXY, SYZ, SXZ). The theoretical basis of the element is discussed in Section 2.8. An element reference guide is given in [Table 4.12](#) with the available loading given in [Table 4.13](#).

The element can be shaped as an 8 or 20 node hexahedron (brick) element, a 6 or 15 node wedge, a 4 or 10 node tetrahedron element, or a 5 or 13 node pyramid element depending on the selected NORDR value. The element configuration, node locations and face numbering convention are shown in [Figure 4.4](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.12](#) gives the output pertinent to this element.



Face No.	NORDR							
	1, 12	2	10	11	20	21	30	31
1	1,2,3,4	1,2,3,4,5,6,7,8	1,2,3	1,2,3,4,5,6	1,2,3	1,2,3,4,5,6	1,2,3,4	1,2,3,4,5,6,7,8
2	5,6,7,8	13,14,15,16,17,18,19,20	4,5,6	10,11,12,13,14,15	1,2,4	1,2,3,8,10,7	1,2,5	1,2,3,10,13,9
3	1,2,6,5	1,2,3,10,15,14,13,9	1,2,5,4	1,2,3,8,12,11,10,7	2,3,4	3,4,5,9,10,8	2,3,5	3,4,5,11,13,10
4	2,3,7,6	3,4,5,11,17,16,15,10	2,3,6,5	3,4,5,9,14,13,12,8	3,1,4	5,6,1,7,10,9	3,4,5	5,6,7,12,13,11
5	3,4,8,7	5,6,7,12,19,18,17,11	3,1,4,6	5,6,1,7,10,15,14,9	-	-	4,1,5	7,8,1,9,13,12
6	4,1,5,8	7,8,1,9,13,20,19,12	-	-	-	-	-	-

Figure 4.4: Element configuration and face numbering convention for available NORDR values, NKTP = 4

Table 4.12: Element reference guide (NKTP = 4)

Element Type	NKTP = 4, 3-D Solid element
Analysis Types	Static, Dynamic, Buckling, Nonlinear Static, Nonlinear Transient
Degrees of Freedom	3 per node: UX, UY, UZ
NORDR (Shape / No. of nodes)	Hexahedron (brick): 8 or 20 nodes (NORDR = 1, 2, 12), see notes 2, 3 Wedge: 6 or 15 nodes (NORDR = 10, 11) Tetrahedron: 4 or 10 nodes (NORDR = 20, 21) Pyramid: 5 or 13 nodes (NORDR = 30, 31)
Real Constants	None
Material Properties	
Isotropic Elastic	4 properties: EX, NUXY, DENS, ALPX
Orthotropic Elastic	13 properties: EX, EY, EZ, NUXY, NUYZ, NUXZ, GXY, GYZ, GXZ, DENS, ALPX, ALPY, ALPZ
Elastoplastic	For nonlinear static only, see *PLASTIC
Hyperelastic	For nonlinear static only, see *HYPEREL
Creep Law	For nonlinear static only, see *CREEP
Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ), and strain energy - Centroidal strains (EXX, EYY, EZZ, EXY, EYZ, EXZ), in principal material directions and/or global directions, see note 4 - Element stresses (SXX, SY, SZZ, SXY, SYZ, SXZ) at centroid, Gauss, and nodal points in principal material directions and/or global directions - Element principal stresses (S1, S2, S3), von Mises equivalent stress, octahedral and maximum shear stresses at centroid, Gauss, and nodal points - Equivalent plastic strain (EPS), and effective yield stress (YLD), for nonlinear static only, at Gauss and nodal points, see note 5 - Second Piola-Kirchhoff or Cauchy stresses, for nonlinear static only, at Gauss and nodal points

Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration, and shock spectrum analyses
Nonlinear Capabilities	<ul style="list-style-type: none"> - Geometric nonlinearity: Large displacements, rotations, and strains (for plasticity analysis, only small strains are assumed); total and updated Lagrangian formulations; deformation dependent loads - Material nonlinearity: elastoplastic and hyperelastic material models - Bubble function for 8-node element (NORDR = 1)

Table 4.13: Available loading (NKTP = 4)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X Y and Z directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal displacements UX, UY, UZ in global X,Y and Z directions (or in local displacement system, if defined), see *SPDISP - Concentrated nodal follower (deformation dependent) forces perpendicular to a reference face of the element, for nonlinear static only, see *CFOLLOWER - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
Pressure Loading	<ul style="list-style-type: none"> - Uniform or non-uniform pressure (force/area) on any face of the element, see *PRESSURE or *DPRESSURE - Uniform or non-uniform follower (deformation dependent) pressure (force/area) on any face of the element, for nonlinear static only, see *PRESSURE
Thermal Loading	<ul style="list-style-type: none"> - Specified nodal temperature, see *NDTEMPER
Body Force	<ul style="list-style-type: none"> - Gravity loading or body forces due to linear acceleration in global X,Y and Z directions, see *BODYFORCE - Centrifugal loads due to angular velocity about the global X, Y, Z directions, see *BODYFORCE - Tangential loads due to angular acceleration about the global X, Y, and Z directions, see *BODYFORCE

Element Library

3-D Solid Element (NKTP = 4)

Ground Motion	<ul style="list-style-type: none">- Forces due to ground motion (support excitation) in the global X, Y, and Z directions, see *GROUND- Forces due to rotational ground motion in the global XY, YZ and ZX plane about a specified point, see *GROUND
----------------------	--

Notes:

1. The element connectivity must be given in the order shown in [Figure 4.4](#), in which the node numbering sequence starts at a corner node and proceeds along the perimeter of a face on the element. This face will be considered face-1 of the element. Details about face numbering for all NKTP = 4 elements are listed in the table in [Figure 4.4](#).
2. The linear hexahedron element (NORDR = 1) uses trilinear shape functions to represent the element geometry. However, for both linear and nonlinear analyses, three additional incompatible shape functions (bubble functions) associated with internal nodeless degrees of freedom are added, along with the trilinear shape functions, to represent the displacement variation. These extra shape functions enable the NORDR = 1 element to model pure bending exactly. See [Section 2.8](#) for details.
3. The linear hexahedron element (NORDR = 12) uses trilinear shape functions to represent both element geometry and displacement variations. This element is available *only* for nonlinear static analysis. For linear static analysis, a corresponding element, that uses trilinear shape functions for both geometry and displacement variations, is the NKTP = 5, NORDR = 1 element.
4. Centroidal strains output is not available for nonlinear static analysis.
5. For nonlinear static analysis, the nodal equivalent plastic strains (EPS) are approximated by extrapolating the Gauss point equivalent plastic strains.

4.6 3-D Thick Shell Element (NKTP = 5)

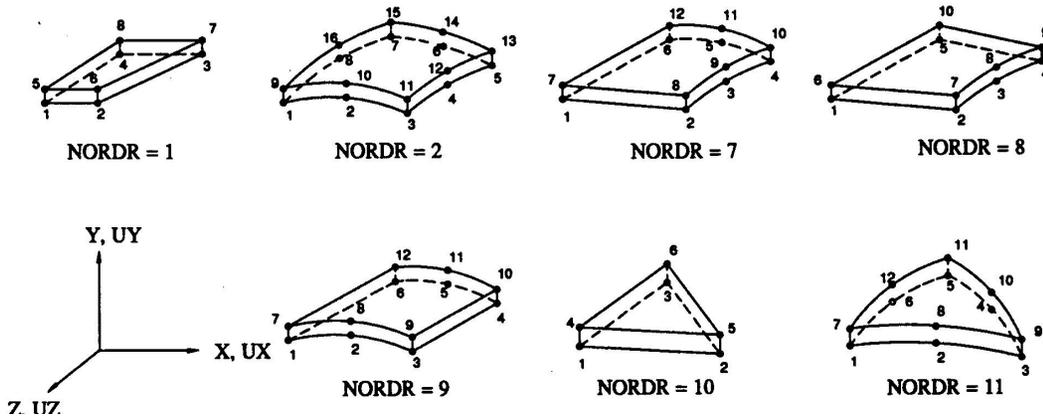
Description

This element is derived by specializing the 3-D solid element (NKTP = 4) to have a linear variation of displacement through the thickness. It is suited for modeling 3-D thick shell structures where the normal stress and strain through the thickness are not negligible. For a thin shell problem, users are advised to use element NKTP = 20, 40 or 41. The element has three degrees of freedom (UX, UY, UZ). The state of stress is characterized by six components (SXX, SYY, SZZ, SXY, SYZ, SXZ). The theoretical basis of the element is discussed in [Section 2.8](#). An element reference guide is given in [Table 4.14](#) with the available loading given in [Table 4.15](#).

The element can have 6 to 16 nodes depending on the selected NORDR values. The element configuration, node locations and face numbering convention are shown in [Figure 4.5](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.14](#) gives the output pertinent to this element.

Element Library

3-D Thick Shell Element (NKTP = 5)



Face No.	NORDR						
	1, 12	2	7	8	9	10	11
1	1,2,3,4	1,2,3,4,5,6,7,8	1,2,3,4,5,6	1,2,3,4,5, 1,2,3,4,5,6	1,2,3	1,2,3,4,5,6	-
2	5,6,7,8	9,10,11,12, 13,14,15,16	7,8,9,10,11,12	6,7,8,9,10	7,8,9,10,11,12	4,5,6	7,8,9,10,11,12
3	1,2,6,5	1,2,3,11,10,9	1,2,8,7	1,2,7,6	1,2,3,9,8,7	1,2,5,4	1,2,3,9,8,7
4	2,3,7,6	3,4,5,13,12,11	2,3,4,10,9,8	2,3,4,9,8,7	3,4,10,9	2,3,6,5	3,4,5,11,10,9
5	3,4,8,7	5,6,7,15,14,13	4,5,6,12,11,10	4,5,10,9	4,5,6,12,11,10	3,1,4,6	5,6,1,7,12,11
6	4,1,5,8	7,8,1,9,16,15	6,1,7,12	5,1,6,10	6,1,7,12	-	-

Figure 4.5: Element configuration and face numbering convention for available NORDR values, NKTP = 5

Table 4.14: Element reference guide (NKTP = 5)

Element Type	NKTP = 5, 3-D Thick shell element
Analysis Types	Static, Dynamic, Buckling
Degrees of Freedom	3 per node: UX, UY, UZ
NORDR	Hexahedron (brick): 8 to 16 nodes (NORDR = 1, 2, 7-9)
(Shape / No. of nodes)	Wedge: 6 or 12 nodes (NORDR = 10, 11)
Real Constants	None
Material Properties	
Isotropic Elastic	4 properties: EX, NUXY, DENS, ALPX
Orthotropic Elastic	13 properties: EX, EY, EZ, NUXY, NUYZ, NUXZ, GXY, GYZ, GXZ, DENS, ALPX, ALPY, ALPZ
Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ), and strain energy - Centroidal strains (EXX, EYY, EZZ, EXY, EYZ, EXZ), in principal material directions and/or global directions - Element stresses (SXX, SYX, SZZ, SXY, SYZ, SXZ) at centroid, Gauss, and nodal points in principal material directions and/or global directions - Element principal stresses (S1, S2, S3), von Mises equivalent stress, octahedral and maximum shear stresses at centroid, Gauss, and nodal points
Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration, and shock spectrum analyses

Table 4.15: Available loading (NKTP = 5)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X, Y and Z directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal displacements UX, UY, UZ in global X, Y and Z directions (or in local displacement system, if defined), see *SPDISP - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
Pressure Loading	<ul style="list-style-type: none"> - Uniform or non-uniform pressure (force/area) on any face of the element, see *PRESSURE or *DPRESSURE
Thermal Loading	<ul style="list-style-type: none"> - Specified nodal temperature, see *NDTEMPER
Body Force	<ul style="list-style-type: none"> - Gravity loading or body forces due to linear acceleration in global X, Y and Z directions, see *BODYFORCE - Centrifugal loads due to angular velocity about the global X, Y and Z directions, see *BODYFORCE - Tangential loads due to angular acceleration about the global X, Y, and Z directions, see *BODYFORCE
Ground Motion	<ul style="list-style-type: none"> - Forces due to ground motion (support excitation) in the global X, Y and Z directions, see *GROUND - Forces due to rotational ground motion in the global XY, YZ, and ZX planes about a specified point, see *GROUND

Notes:

1. Nodes must be numbered according to the sequence given in [Figure 4.5](#), in which the node numbering sequence starts at a corner node and proceeds along the perimeter of a face on the element. This will be considered face-1 of the element. Following the right hand rule, the nodes of face-2 of the element are then defined with the first node being above the first node of face-1 to face-2 has to be the thickness direction, through which displacement variation is restricted to be linear.

4.7 3-D Solid Piezoelectric Element (NKTP = 6)

Description

This element is based on a general 3-D electromechanical constitutive formulation and is suited for modeling 3-D solid body made of piezoelectric materials under mechanical and electrical loading. The element has three displacements and one electric potential degree of freedom per node (UX, UY, UZ, PHI). The state of the formulation is characterized by six stress components (SXX, SYX, SZZ, SXY, SYZ, SXZ) and three electric field components (EFLDX, EFLDY, EFLDZ). An element reference guide is given in [Table 4.16](#). The available loading is given in [Table 4.17](#)

The element can be shaped as an 8 or 20 node hexahedron (brick) element, a 6 or 15 node wedge, a 4 or 10 node tetrahedron element, or a 5 or 13 node pyramid element depending on the selected NORDR value. The element configuration, node locations and face numbering convention are shown in [Figure 4.6](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.16](#) gives the output pertinent to this element.

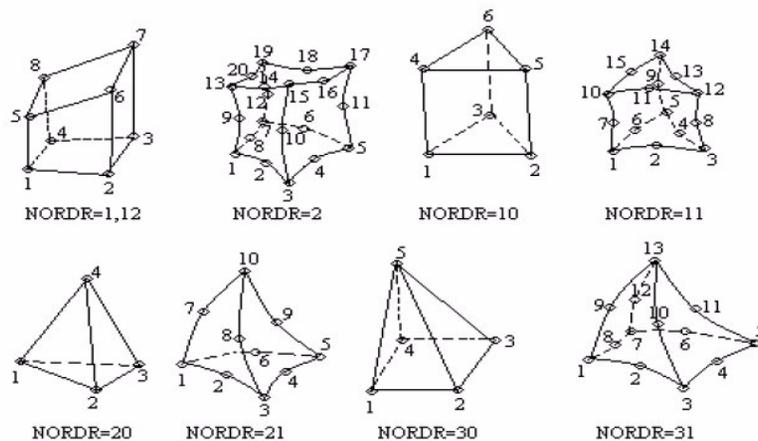


Figure 4.6: Element configuration and face numbering convention for available NORDR values, NKTP = 6

Table 4.16: Element reference guide (NKTP = 6)

Element Type	NKTP = 6, 3-D solid piezoelectric element
Analysis Type	Static, Eigenvalue, linear transient dynamics
Degrees of Freedom	4 per node: UX, UY, UZ, PHI
NORDR (Shape/No. of nodes)	<ul style="list-style-type: none"> - Hexahedron (brick): 8 or 20 nodes (NORDR = 1, 2, 12), see notes - Wedge: 6 or 15 nodes (NORDR = 10,11) - Tetrahedron: 4 or 10 nodes (NORDR = 20,21) - Pyramid: 5 or 13 nodes (NORDR = 30,31)
Real Constants	None
Material Properties	
Isotropic Elastic	4 properties: EX, NUXY, DENS, ALPX
Orthotropic Elastic	13 properties: EX,EY, EZ, NUXY, NUYZ, NUXZ, GXY, GYZ, GXZ, DENS, ALPZ, APLY, ALPZ
Piezoelectric Properties	
Piezoelectric matrix	<ul style="list-style-type: none"> - 18 properties: PZ11, PZ12, PZ13, PZ21, PZ22, PZ23, PZ31, PZ32, PZ33, PZ41, PZ42, PZ43, PZ51, PZ52, PZ53, PZ61, PZ62, PZ63
Dielectric matrix	<ul style="list-style-type: none"> - 6 properties: EXX, EXY, EXZ, EYY, EYZ, EZZ
Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ), and strain energy - Centroidal strains (EXX, EYY, EZZ, EXY, EYZ, EXZ), in principal material directions and/or global directions, see note - Element stresses (SXX, SYY, SZZ, SXY, SYZ, SXZ) at centroid, Gauss, and nodal points in principal material directions and/or global directions - Element principal stresses (S1, S2, S3), von Mises equivalent stress, octahedral and maximum shear stresses at centroid, Gauss, and nodal points - Electric fields (EFLDX, EFLDY, EFLDZ) at centroid, Gauss, and nodal points in global directions - Electric fluxes (FLXX, FLXY, FLXZ) at centroid, Gauss, and nodal points in global directions
Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue and linear transient dynamic analyses

Table 4.17: Available loading (NKTP = 6)

Nodal Loading	Concentrated nodal forces (FX, FY, FZ) and electric charge (ECHR) in global X, Y and Z directions (or in local displacement system, if defined), see *CFORCE Specified non-zero displacements UX, UY, UZ and electric potential PHI in global X, Y, Z directions (or in local displacement system, if defined), see *SPDISP
Pressure Loading	Uniform or non-uniform pressure (force/area) on any face of the element. See *PRESSURE
Thermal Loading	Specified nodal temperature, see *NDTEMPER
Body Force	Gravity loading or body forces due to linear acceleration in global X, Y, and Z directions, see *BODYFORCE Centrifugal loads due to angular velocity about the global X, Y, and Z directions, see *BODYFORCE Tangential loads due to angular acceleration about global X, Y, and Z directions, see *BODYFORCE
Electric charge density	Uniform or non-uniform electric charge density on any face of the element. See *ECDENSITY

Notes:

1. The element connectivity must be given in the order shown in [Figure 4.6](#) in which the node numbering sequence starts at a corner node and proceeds along the perimeter of a face on the element. This face will be considered face-1 of the element. Detail about face numbering for all NKTP = 6 elements are listed in the table in [Figure 4.6](#).
2. The linear hexahedron element (NORDR = 1) uses trilinear shape functions to represent the element geometry. However, three additional incompatible shape functions (bubble functions) associated with internal node less degrees of freedom are added, along with the trilinear shape functions, to represent the displacement variation. These extra shape functions enable the NORDR = 1 element to model pure bending exactly.
3. The linear hexahedron element (NORDR = 12) uses trilinear shape functions to represent both element geometry and displacement variations. No bubble functions are used in this element.

4.8 3-D Laminated Composite Solid Element (NKTP = 7)

Description

This element is based on a general 3-D state of stress and is suited for modeling thick laminated composite structures. The element consists of a number of layers of perfectly bonded orthotropic materials. The element has three translational degrees of freedom per node (UX, UY, UZ). The state of stress is characterized by six stress components (SXX, SY Y, SZZ, SXY, SYZ, SXZ). The theoretical basis of the element is discussed in [Section 2.8](#). An element reference guide is given in [Table 4.18](#) with the available loading given in [Table 4.19](#).

The element can be shaped as an eight or twenty node solid hexahedron element depending on the selection of NORDR value. The element configuration, layer setup, node locations and face numbering convention are shown in [Figure 4.7](#). The lamination sequence is between the top and bottom faces of the element with the layer setup starting from the top face. A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#), respectively. [Table 4.18](#) gives the output pertinent to this element.

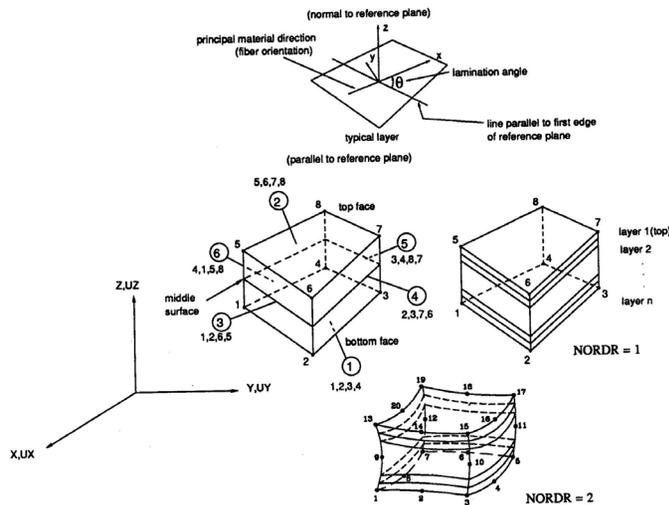


Figure 4.7: Element configuration, NKTP = 7, NORDR = 1
(circled numbers indicate face numbers)

Table 4.18: Element reference guide (NKTP = 7)

Element Type	NKTP = 7, 3-D Laminated composite solid element
Analysis Types	Static, Dynamic
Degrees of Freedom	3 per node: UX, UY, UZ
NORDR (Shape / No. of nodes)	Hexahedron: 8 node (NORDR = 1), 20 node (NORDR = 2) 4 (NORDR = 1) / 8 (NORDR = 2)
Real Constants	4 layer thicknesses for each layer, entered in *RCTABLE, see notes 2-5 1 layer rotation angle for each layer, entered in *LAMANGLE, see notes 2-5
Material Properties Orthotropic Elastic Strength Properties (used in failure criteria)	13 properties for each <i>different</i> material in principal material directions, see note 6: EX, EY, EZ, NUXY, NUYZ, NUXZ, GXY, GYZ, GXZ, DENS, ALPX, ALPY, ALPZ 12 properties for each <i>different</i> material (in principal material directions) FXT, FXC, FYT, FYC, FZT, FZC, FSXY, FSYZ, FSXZ, F12, F23, F13 (Default strength properties assumed, if not given, see note 7)
Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ), and strain energy - Centroidal strains (EXX, EYY, EZZ, EXY, EYZ, EXZ) at the midsurface of each layer in principal material directions and/or global directions - Element stresses (SXX, SYX, SZZ, SXY, SYZ, SXZ) at the midsurface of each layer at layer centroid, Gauss and/or nodal points in principal material directions and/or global directions, see notes 9, 10 - Element stress failure criteria: maximum stress, Tsai-Wu, Tsai-Hill (or user-defined failure criterion), see notes 8, 9
Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration, and shock spectrum analyses

Table 4.19: Available loading (NKTP = 7)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X, Y, and Z directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal displacements UX, UY, UZ in global X, Y and Z directions (or in local displacement system, if defined), see *SPDISP - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
Pressure Loading	<ul style="list-style-type: none"> - Uniform or non-uniform pressure (force/area) on any face of the element, see *PRESSURE or *DPRESSURE
Thermal Loading	<ul style="list-style-type: none"> - Specified nodal temperature, see *NDTEMPER
Body Force	<ul style="list-style-type: none"> - Gravity loading or body forces due to linear acceleration in global X, Y and Z directions, see *BODYFORCE - Centrifugal loads due to angular velocity about global X, Y, and Z axes, see *BODYFORCE - Tangential loads due to angular acceleration about the global X, Y, and Z axes, see *BODYFORCE
Ground Motion	<ul style="list-style-type: none"> - Force due to ground motion (support excitation) in the global X, Y and Z directions, see *GROUND - Force due to rotational ground motion in the global XY, YZ, XZ planes about a specified point, see *GROUND

Notes:

1. The element connectivity should be given as shown in [Figure 4.7](#), in which the node numbering starts at a corner node and proceeds along the perimeter of the bottom face, then the top face is defined such that node 5 is above node 1 (in the right-hand rule sense).
2. For most NISA elements, the variable MATID in the *ELEMENTS data group refers to a material property table defined in *MATERIAL data group. However, this is inadequate for composite elements since each layer may be of a different material. Therefore, for this element, MATID is a pointer to an entry in the *LAMSQ2 (lamination sequence) data group. The *LAMSQ2 data group in turn contains pointers to the layer thicknesses, rotation angles and material properties for all layers (*RCTABLE, *LAMANGLE and *MATERIAL data groups, respectively). Note that the number of real constant tables, lamination angle tables, or material property tables

may not necessarily be equal to the number of layers, since more than one layer may be of the same material type, thickness type, or lamination angle (orientation) type.

3. There is no need to enter a non-zero value for the variable IDRC (read constant ID number) in the *ELEMENTS data group for this element. Any value entered for IDRC will be reset to zero.
4. The rotation angle is constant for each layer. Therefore, only the first entry in the referenced rotation angle table is used. The rotation angle is measured from the first edge of the reference plane and about its normal using the right-hand-rule. The reference plane is specified as either the bottom face, top face or the middle surface depending upon the value of IDRA given in the *LAMSQ2 data group. IDRA = 0, 7, 8 corresponds to bottom, top, middle surface of the element, respectively (use IDRA = 8 for NORDR = 2). If the reference surface is warped, a mean flat plane is generated (The mean plane is the plane passing through the middle points of the four edges.). The first edge of the reference plane is the line joining nodes 1 and 2 for bottom face, nodes 5 and 6 for top face or the average of these two lines for middle reference plane (see [Figure 4.7](#)). If the reference surface is warped, the projection of its first edge on its mean plane is used. Thus, for the purpose of defining the rotation angle, all layers are assumed to be flat and parallel to the reference plane. The user should be aware of the inaccuracy introduced by this assumption for warped elements.
5. Each layer needs four/eight thickness entries in the real constant table, *RCTABLE group. The total thickness of the laminate is inferred from the element connectivity and the nodal coordinates data, and if it does not match the sum of the layer thicknesses, the input layer thicknesses are scaled in order to be consistent with the element thickness. Thus, the layer thicknesses may be given as percentages of the total thickness or they may be normalized by the total thickness while inputting in *RCTABLE data.

The actual thickness of a typical layer is internally computed as:

$$t_i = T \cdot r_i / \sum_{j=1}^n r_j$$

where, t_i is the calculated i -th layer thickness, r_i is thickness entered in the real constant table, n is the number of layers and T is the total thickness of the laminate (computed from the nodal coordinates data).

6. The element consists of a number of layers of perfectly bonded orthotropic materials. For each different material, all of the mechanical properties (EX, EY, EZ, NUXY, NUYZ, NUXZ, GXY, GYZ, GXZ, ALPX, ALPY, ALPZ, and DENS) should be defined. A default value of zero is assumed if a typical property is not specified. These properties are defined in the material principal directions (x, y, z), such that x is the first

material axis (fiber direction), y is the second material axis and is normal to x, and z is normal to the laminate as shown in [Figure 4.7](#). The layer is in the xy plane of the material axes system, which is assumed to be parallel to the reference plane.

7. The strength properties are used to compute the maximum stress and Tsai-Wu failure criteria for each layer. The properties are defined as:

FXT, FXC	– x-direction tensile and compressive failure stress, respectively
FYT, FYC	– y-direction tensile and compressive failure stress, respectively
FZT, FZC	– z-direction tensile and compressive failure stress, respectively
FSXY	– inplane (xy) shear failure stress
FSYZ	– out-of-plane (yz) shear failure stress
FSXZ	– out-of-plane (xz) shear failure stress
F12	– coupling coefficient between x and y directions for Tsai-Wu theory
F23	– coupling coefficient between y and z directions for Tsai-Wu theory
F13	– coupling coefficient between x and z directions for Tsai-Wu theory

(Note: compressive failure stresses are also input as positive values)

If a typical strength property is not specified, a default strength property in proportion to the graphic/epoxy strength properties [4.1] is assumed as follows:

$$\begin{aligned}
 FXC &= \frac{10 \times 10^4}{30 \times 10^6} EX, & FXT &= \frac{15 \times 10^4}{30 \times 10^6} EX \\
 FYC = FZC &= \frac{17 \times 10^3}{7.5 \times 10^5} EY, & FYT = FZT &= \frac{6 \times 10^3}{7.5 \times 10^5} EY \\
 FSXY = FSXZ = FSYZ &= \frac{10^4}{3.75 \times 10^5} GXY
 \end{aligned}$$

The default values for the coupling coefficients are:

$$\begin{aligned}
 F12 &= \frac{1}{FXAVG * FYAVG}, \\
 F23 &= \frac{1}{FYAVG * FZAVG}, & F13 &= \frac{1}{FXAVG * FZAVG}
 \end{aligned}$$

in which

$$\begin{aligned}
 FXAVG &= \frac{1}{2}(FXC + FXT), \quad FYAVG = \frac{1}{2}(FYC + FYT), \quad FZAVG \\
 &= \frac{1}{2}(FZC + FZT)
 \end{aligned}$$

8. The following failure theories are implemented for this element:

- (a) **The maximum stress failure theory:** each of the stress components is compared independently to a unidirectional allowable (failure) stress.

The allowable stress may be different in tension and compression. The ratio of actual to allowable is computed at every point where stresses are computed. Layer failure is presumed to have occurred when the ratio exceeds 1.0. This theory neglects all interaction among stress components.

- (b) **The Tsai-Wu failure theory** [4.2]: the interaction of the stress components is accounted for by defining a failure surface with the functional form:

$$\begin{aligned}
 F = & F1*SXX + F2*SYX + F3*SZZ + F11*SXX**2 + F22*SYX**2 + \\
 & F33*SZZ**2 + F12*SXX*SYX + F13*SXX*SZZ + \\
 & F23*SYX*SZZ + F44*SYZ**2 + F55*SXZ**2 + F66*SYX**2
 \end{aligned}$$

where,

$$\begin{aligned}
 F1 = 1/FXT - 1/FXC, \quad F2 = 1/FYT - 1/FYC, \quad F3 = 1/FZT - 1/FZC, \\
 F11 = 1/(FXT * FXC), \quad F22 = 1/(FYT * FYC), \quad F33 = 1/(FZT * FZC), \\
 F44 = 1/FSYZ**2, \quad F55 = 1/FSXZ**2, \quad F66 = 1/FSXY**2
 \end{aligned}$$

when the value of this function reaches unity, failure of the layer under combined loading is predicted. The above coefficients are computed from the user-input engineering failure stresses. The calculation of F12, F23, F13 are left to the user since they depend upon the experimental method chosen to perform the failure tests, see [4.3].

- (c) **The Tsai-Hill failure theory** or user-defined failure theory (see Appendix D.4): If a user-defined failure theory is used, the Tsai-Hill theory will be replaced by the user-defined failure theory. The Tsai-Hill theory is defined as follows:

$$\begin{aligned}
 F = & (SXX/FX)**2 + (SYX/FY)**2 + (SZZ/FZ)**2 \\
 & - [1/(FX * FX) + 1/(FY * FY) - 1/(FZ * FZ)] *SXX * SYX \\
 & - [1/(FX * FX) - 1/(FY * FY) + 1/(FZ * FZ)] *SXX * SZZ \\
 & - [-1/(FX * FX) + 1/(FY * FY) + 1/(FZ * FZ)] *SYX * SZZ \\
 & + (SXY/FSXY)**2 + (SYZ/FSYZ)**2 \\
 & + (SXZ/FSXZ)**2
 \end{aligned}$$

FX, FY, FZ, FSXY, FSYZ and FSXZ are the failure stresses under unidirectional loading in the axial and shear directions. This theory does not account for the difference between tensile and compressive failure stress. NISA applies the average value of the user-input tensile and compressive failure stresses in the formula.

9. The element stresses (SXX, SYY, SZZ, SXY, SYZ, SXZ) are calculated at the midsurface of each layer at the layer centroid, Gauss and/or nodal points. The stresses may be obtained in principal material directions and/or global directions. The first four nodes of the element (bottom face) are used as reference nodes for the layers in the printout. The stresses may be filtered before printout. The ratio of actual stress to allowable stress will be compared to the threshold value entered in *SFDCOMP data group. The stress printout for any layer will be suppressed if all the stress ratios and the Tsai-Wu failure index are less than the threshold value.
10. To obtain the element stresses at the top or bottom of a layer in addition to the midsurface which is the default location, define a very thin layer of thickness $t \leq 10^{-5}T$ on top or bottom face of the actual layer, where, T is the total thickness of the laminate. The stiffness contributions for all such very thin layers are ignored. The rotation angle and material properties of the fictitious layer should be the same as those of the actual layer. Using this procedure, the stresses can also be obtained at locations very close to the actual nodes of the element.
11. The nodal stress averaging option is not available for this element. However, the following contours are available on an element by element basis using DISPLAY-POST program.
 - (a) Unaveraged nodal layers stresses, failure criteria and layer strains.
 - (b) Maximum nodal value over all the layers of an individual stress component (e.g., SXX). The layer number and component number are indicated on the plot.
 - (c) Maximum nodal value over all the layers and all the stress components considered together. The layer number and component number are indicated on the plot.

4.9 3-D Membrane Piezoelectric Element (NKTP = 8)

Description

This element is based on 3-D membrane deformation and electromechanical constitutive formulation. It is intended for modeling 3-D thin structures, made of piezoelectric materials under mechanical and electrical loading. Membrane element does not include bending and transverse shear stiffness in the formulation. There will be no stiffness in the direction normal to the element surface. The element has three displacements and one electric potential degrees of freedom per node (UX, UY, UZ, PHI). The state of the formulation is characterized by three local stress components (SXX, SYY, SXY) and three electric field components (EFLDX, EFLDY, EFLDZ). An element reference guide is given in [Table 4.20](#). The available loading is given in [Table 4.21](#).

The element can be shaped as a 4-node element. The element configuration, node locations and face numbering convention are shown in [Figure 4.8](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.20](#) gives the output pertinent to this element.

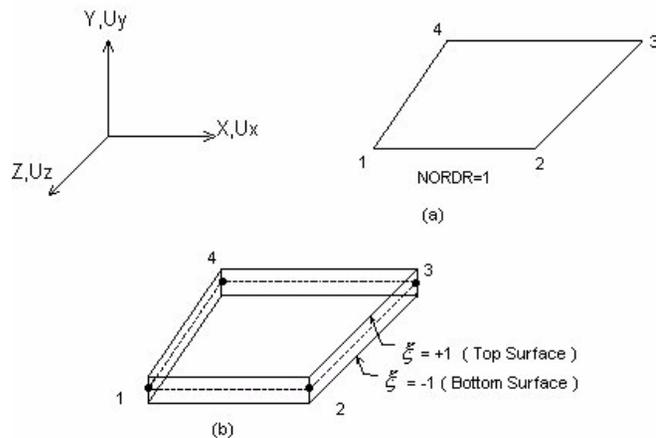


Figure 4.8: 3D Membrane Piezoelectric Element (NKTP = 8) (a) Element configuration
(b) Face numbering convention for bottom and top surface

Table 4.20: Element reference guide (NKTP = 8)

Element Type	NKTP = 8, 3-D membrane piezoelectric element
Analysis Type	Static, Eigenvalue, linear transient dynamics
Degrees of Freedom	4 per node: UX, UY, UZ, PHI
NORDR (Shape/No. of nodes)	Quad: 4 nodes (NORDR = 1), see notes
Real Constants	Nodal Thickness
Material Properties	
Isotropic Elastic	4 properties: EX, NUXY, DENS, ALPX
Orthotropic Elastic	13 properties: EX, EY, EZ, NUXY, NUYZ, NUXZ, GXY, GYZ, GXZ, DENS, ALPX, ALPY, ALPZ
Piezoelectric Properties	
Piezoelectric matrix	18 properties: PZ11, PZ12, PZ13, PZ21, PZ22, PZ23, PZ31, PZ32, PZ33, PZ41, PZ42, PZ43, PZ51, PZ52, PZ53, PZ61, PZ62, PZ63
Dielectric matrix	6 properties: EXX, EXY, EXZ, EYY, EYZ, EZZ
Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ), and strain energy - Centroidal strains (EXX, EYY, EZZ, EXY, EYZ, EXZ), in principal material directions and/or global directions, see note - Element stresses (SXX, SYX, SZZ, SXY, SYZ, SXZ) at centroid, Gauss, and nodal points in principal material directions and/or global directions - Element principal stresses (S1, S2, S3), von Mises equivalent stress, octahedral and maximum shear stresses at centroid, Gauss, and nodal points - Electric fields (EFLDX, EFLDY, EFLDZ) at centroid, Gauss, and nodal points in global directions - Electric fluxes (FLXX, FLXY, FLXZ) at centroid, Gauss, and nodal points in global directions
Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, and linear transient dynamic analyses

Table 4.21: Available loading (NKTP = 8)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces (FX, FY, FZ) and electric charge (ECHR) in global X, Y and Z directions (or in local displacement system, if defined), see *CFORCE - Specified non-zero displacements UX, UY, UZ and electric potential PHI in global X, Y, Z directions (or in local displacement system, if defined), see *SPDISP
Pressure Loading	Uniform or non-uniform pressure (force/area) on any face of the element. See *PRESSURE
Thermal Loading	Specified nodal temperature, see *NDTEMPER
Body Force	<ul style="list-style-type: none"> - Gravity loading or body forces due to linear acceleration in global X, Y, and Z directions, see *BODYFORCE - Centrifugal loads due to angular velocity about the global X, Y, and Z directions, see *BODYFORCE - Tangential loads due to angular acceleration about global X, Y, and Z directions, see *BODYFORCE
Electric charge density	Uniform or non-uniform electric charge density on any face of the element. See *ECDENSITY

Notes:

1. The element connectivity must be given in the order shown in [Figure 4.8](#) in which the node numbering sequence starts at a corner node and proceeds along the perimeter of the surface of the element. The face will be considered face-1 (top face) of the element if the nodal connectivity is counterclockwise from the viewpoint above the face. The face on the other side is face-2 (bottom face).
2. For orthotropic materials, the material principal axes should be defined in such a way that the material Z-axis is normal to the middle surface of the element. If the X and Y material axes are not tangent to the middle surface, a revised material axes are chosen such that the material Z-axis is normal to the element and the new system is closest to the user defined system.
3. Since the element does not have stiffness in normal direction, it may create singularity problem when membrane elements are coplanar. Build small curvature in the model or use other type of element as support may be needed to avoid singularity problem.

4.10 3-D Hybrid Solid Element (NKTP = 9)

Description

This element is an 8-node hybrid solid hexahedron (brick) and is based on a three-dimensional state of stress. It is suited for modeling thick structures. However, moderately thin plate or shell structures (up to a thickness to side length ratio of 10^{-3}) can also be efficiently analyzed with this element. The element has three translational degrees of freedom per node (UX, UY, UZ). The state of stress is characterized by six stress components (SXX, SYY, SZZ, SXY, SYZ, SXZ). The theoretical basis of the element is discussed in [Section 2.8](#). An element reference guide is given in [Table 4.22](#) with the available loading given in [Table 4.23](#).

The element configuration, node locations and face numbering convention are shown in [Figure 4.9](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.22](#) gives the output pertinent to this element.

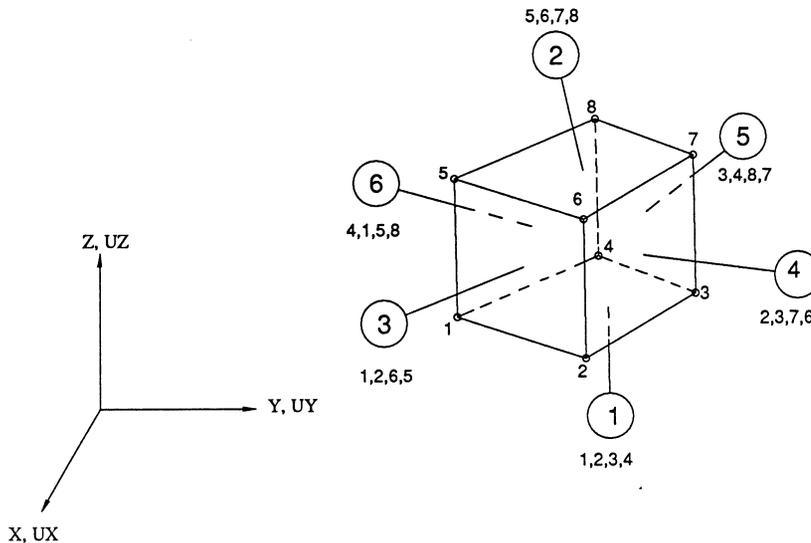


Figure 4.9: Element configuration, NKTP = 9, NORDR = 1
(circled numbers indicate face numbers)

Table 4.22: Element reference guide (NKTP = 9)

Element Type	NKTP = 9, 3-D Hybrid solid element
Analysis Types	Static, Dynamic, Buckling
Degrees of Freedom	3 per node: UX, UY, UZ
NORDR (Shape / No. of nodes)	Hexahedron (brick): 8 node (NORDR = 1)
Real Constants	None
Material Properties	
Isotropic Elastic	4 properties: EX, NUXY, DENS, ALPX
Orthotropic Elastic	13 properties: EX, EY, EZ, NUXY, NUYZ, NUXZ, GXY, GYZ, GXZ, DENS, ALPX, ALPY, ALPZ
Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ) and strain energy - Centroidal strains (EXX, EYY, EZZ, EXY, EYZ, EXZ) in principal material directions and/or global directions - Element stresses (SXX, SYX, SZZ, SXY, SYZ, SXZ) at centroid, Gauss and nodal points in principal material directions and/or global directions - Element principal stresses (S1, S2, S3) von Mises equivalent stress, octahedral and maximum shear stresses at centroid, Gauss and nodal points
Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration, and shock spectrum analyses

Table 4.23: Available loading (NKTP = 9)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X, Y, and Z directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal displacements UX, UY, UZ in global X, Y and Z directions (or in local displacement system, if defined), see *SPDISP - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
----------------------	---

Element Library

3-D Hybrid Solid Element (NKTP = 9)

Pressure Loading	- Uniform or non-uniform pressure (force/area) on any face of the element, see *PRESSURE or *DPRESSURE
Thermal Loading	- Specified nodal temperature, see *NDTEMPER
Body Force	- Gravity loading or body forces due to linear acceleration in global X, Y and Z directions, see *BODYFORCE - Centrifugal loads due to angular velocity about global X, Y, and Z directions, see *BODYFORCE - Tangential loads due to angular acceleration about the global X, Y, and Z directions, see *BODYFORCE
Ground Motion	- Force due to ground motion (support excitation) in the global X, Y and Z directions, see *GROUND - Force due to rotational ground motion in the global XY, YZ, XZ planes about a specified point, see *GROUND

Notes:

1. The element connectivity must be given in the order shown in [Figure 4.9](#) in which the node numbering sequence starts at a corner node and proceeds along the perimeter of a face on the element. This will be considered face-1 of the element. Following the right hand rule, the nodes of face-2 of the element are then defined with the first node being above the first node of face-1.
2. Unlike the 3-D solid element, NKTP = 4, this element may be used to model thin plates or shells (up to a thickness to side length ratio of 10^{-3}). However, this element is more expensive to use.

4.11 3-D Membrane Element (NKTP = 10)

Description

This 3-D membrane element includes membrane deformation only and is suited for modeling thin structure where bending and transverse shear deformations can be ignored. This element has no stiffness in the direction normal to element surface. The element has three translational degrees of freedom (UX, UY, and UZ) per node and has no rotational degrees of freedom. An element reference guide is given in [Table 4.24](#) and with the available loading listed in [Table 4.25](#).

The element configuration, node locations and face numbering convention for top and bottom surface (note 2) are shown in [Figure 4.10](#). [Table 4.24](#) also gives the output pertinent to this element.

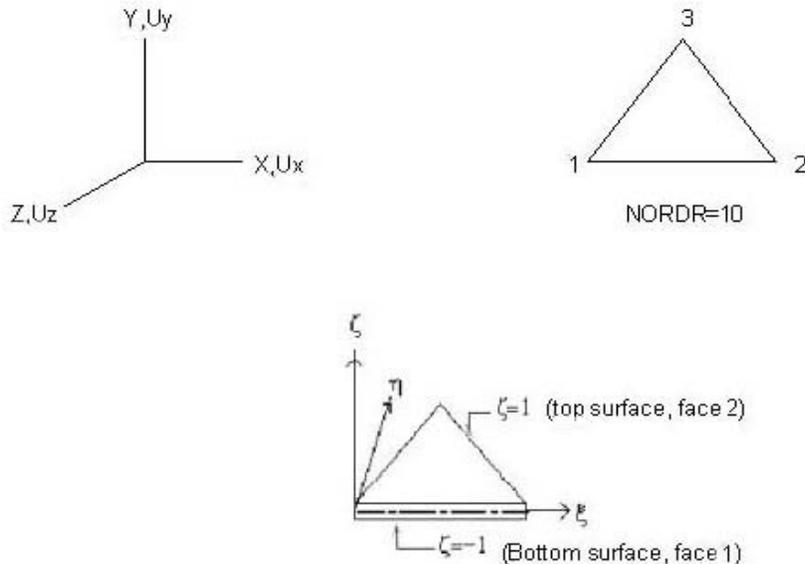


Figure 4.10: 3-D Configuration and face numbering convention, NKTP = 10, NORDR = 10

Table 4.24: Element reference guide (NKTP = 10)

Element Type	NKTP = 10, 3D membrane element
Analysis Types	Nonlinear Static
Degrees of Freedom	3 per node UX, UY, UZ
NORDR	Triangle: 3 nodes (NORDR = 10)
Real Constants	3 nodal thickness (same as number of nodes)
Material Properties	
Isotropic Elastic	4 properties: EX, NUXY, DENS
Orthotropic Elastic	7 properties: EX, EY, NYXY, GXY, DENS, ALPX, ALPY
Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ, MX, MY, MZ) - Element strains at Gauss points and nodal points (EX, EY, EXY) in local and global directions - Element stresses at Gauss points and nodal points in local and global directions (SXX, SYX, SXY) - Element principal stresses (S1, S2, S3), von Mises equivalent stress, octahedral and maximum shear stress at Gauss and nodal points - Stress resultants (NXX, NYX, NXY) at nodal points
Nonlinear Capabilities	<ul style="list-style-type: none"> - Geometric nonlinearity, large displacement, updated Lagrangian formulations, deformation dependent loads, see note 4

Table 4.25: Available loading (NKTP = 10)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X, Y, and Z directions (or in local displacement system, if defined), see *CFORCE - Specified non-zero nodal displacements UX, UY, and UZ in global directions (or in local displacement system, if defined), see *SPDISP - Concentrated nodal follower (deformation dependent) forces perpendicular to the middle surface of the element, see *CFOLLOWER
Pressure Loading	<ul style="list-style-type: none"> - Uniform pressure (force / area) and distributed pressure on top or bottom faces of the element, see *PRESSURE and note 2. - Uniform or non-uniform follower (deformation dependent) pressure (force/area) on top or bottom faces of the element, see *PRESSURE and note 2
Thermal Loading	<ul style="list-style-type: none"> - Specified nodal temperature. See *NDTEMPER
Body Force	<ul style="list-style-type: none"> - Gravity Loading or body forces due to linear acceleration in global X, Y, and Z directions, see *BODYFORCE - Centrifugal loads due to angular velocity about the global X, Y, and Z directions, see *BODYFORCE
	<ul style="list-style-type: none"> - Tangential loads due to angular acceleration about the global X, Y, and Z directions, see *BODYFORCE

Notes:

1. The element may be oriented in space in any arbitrary direction. The element connectivity must be in the order shown in [Figure 4.10](#), in which the node numbering sequence starts at a corner node and proceeds along the perimeter of the element.
2. The bottom and top faces of the element are numbered as faces 1 and 2 respectively. The top and bottom faces are determined as follows:
 - Choose a viewpoint so that the element is defined counterclockwise.
 - The top face is the face closer to you.
3. For orthotropic materials, the material principal axes should be defined in such a way that the material Z-axis is normal to the middle surface of the element. If the X and Y material axes are not tangent to the middle surface, a revised material axes are chosen such that the material Z-axis is normal to the element and the new system is closest to the user defined system.

Element Library

3-D Membrane Element (NKTP = 10)

4. Since this element does not have stiffness in normal direction, any load normal to the element surface can create singular solution situation. Membrane stresses and curvature in the model are required to build-up resistance to loads in normal direction. It is advised to use very small loads or pre-stress in the first step to allow membrane force and curvature build-up in the model to avoid possible singular solution.

4.12 3-D Tapered Beam Element (NKTP = 11)

Description

This element is a 2-node isoparametric 3-D beam element. The cross-section properties (e.g., area and moment of inertia) may be varied along the beam axis. They are specified at the end points of the beam element. Four types of variations of section properties between the specified locations are available (see note 3). The element formulation includes stretching, bending, transverse shear and torsional effects. The offset of beam vertices from the mesh nodal points and the offset of beam centroid from the shear center are allowed in this beam element. The element also has end release capability (see note 10). The deformation is characterized by three translation components of displacement (UX, UY, UZ) and three components of rotation (ROTX, ROTY, ROTZ). The local x-axis of the beam is along the centroidal axis. The local y and z axes are user defined and are not necessary principal axes of the cross section (see note 9). The element configuration, node locations, and face numbering convention are shown in [Figure 4.11](#). An element reference guide is briefed in [Table 4.26](#) with the available loading listed in [Table 4.27](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#), respectively. [Table 4.26](#) gives the output pertinent to this element. The required real constants for this element are given in [Table 4.28](#), [Table 4.29](#) and [Table 4.29](#). For a given problem, not all the entries in the real constant table may need to be specified. Default values are assumed for entries not specified, (see notes 2, 4, and 7). Whenever all entries in any one card are not needed, this card should be skipped in the real constant table.

Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration, and spectrum analyses
Special Feature	<ul style="list-style-type: none"> - End release (see note 9) - -Partial fixity (see note 10)

Table 4.27: Available loading (NKTP = 11)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X, Y and Z directions and concentrated nodal moments about global X, Y and Z axes (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal displacements UX, UY, UZ and nodal rotations ROTX, ROTY, ROTZ in global directions (or in local displacement system, if defined), see *SPDISP - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
Pressure and Partially Distributed Loading	Uniform or non-uniform pressure (force/length) on top or bottom face of the element; see *PRESSURE, *BEAMLOAD, *DPRESSURE and note 13, 14
Thermal Loading	Specified nodal temperature and nodal temperature difference in element local y and z directions, see *NDTEMPER, *NDTEMPDIF, and note 12
Body Force	<ul style="list-style-type: none"> - Gravity loading or body forces due to linear acceleration in global X, Y and Z directions, see *BODYFORCE - Centrifugal loads due to angular velocity about the global X, Y and Z axes, see *BODYFORCE - Tangential loads due to angular acceleration about the global X, Y, and Z axes, see *BODYFORCE
Ground Motion	<ul style="list-style-type: none"> - Forces due to ground motion (support excitation) in the global X, Y and Z directions, see *GROUND - Forces due to rotational ground motion in the global XY, YZ and ZX planes about a specified point, see *GROUND

Table 4.28: List of real constants for 3-D beam (NKTP = 11)

Entry	Variable	Description
1	KEY	Key for variation of section properties between consecutive locations (stations) along the beam, see note 3
2	A	Cross-sectional area at first vertex (station 1)
3	IYY	Moment of inertia about y-axis ($\int z^2 dA$) at station 1
4	IZZ	Moment of inertia about z-axis ($\int y^2 dA$) at station 1
5	IYZ	Product moment of inertia in yz plane ($\int yz dA$) at station 1
6	J	Torsional constant at station 1, see note 2
7	NPT	Number of points on cross-section for stress calculation at station 1, $0 \leq NPT \leq 4$. Skip entries 9 - 16 (one card) if NPT = 0 not used, enter zero
8	—	not used, enter zero
9	Y1	y-coordinate of the first point on cross-section at station 1 for stress recovery (required if $NPT \geq 1$), otherwise entries 9 - 16 (one card) should be skipped
10	Z1	z-coordinate of the first stress point
11	Y2	y-coordinate of the second stress point (required if $NPT \geq 2$, otherwise enter zero)
12	Z2	z-coordinate of the second stress point
13	Y3	y-coordinate of the third stress point (required if $NPT \geq 3$, otherwise enter zero)
14	Z3	z-coordinate of the third stress point
15	Y4	y-coordinate of the fourth stress point (required if $NPT \geq 4$, otherwise enter zero)
16	Z4	z-coordinate of the fourth stress point
17	--	Not used
18	A	Cross-sectional area at node 2

Entry	Variable	Description
19	IYY	Moment of inertia about y-axis ($\int y^2 dA$) at node 2
20	IZZ	Moment of inertia about z-axis ($\int z^2 dA$) at node 2
21	IYZ	Product moment of inertia in yz plane ($\int yz dA$) at node 2
22	J	Torsional constant at node 2
23	NPT	Number of points on cross-section at node 2 for stress calculation, $0 \leq NPT \leq 4$. Skip entries 25 - 32 (one card) if NPT = 0
24	—	Not used, enter zero
25	Y1	y-coordinate of the first point on cross-section at node 2 for stress recovery, required if $NPT \geq 1$, otherwise entries 25-32 (one card) should be skipped
26	Z1	z-coordinate of the first stress point
27	Y2	y-coordinate of the second stress point
28	Z2	z-coordinate of the second stress point
29	Y3	y-coordinate of the third stress point
31	Z3	z-coordinate of the third stress point
30	Y4	y-coordinate of the fourth stress point
32	Z4	z-coordinate of the fourth stress point
33	--	Not used
34	--	Not used
35	--	Not used
36	--	Not used
37	--	Not used
38	--	Not used

Element Library**3-D Tapered Beam Element (NKTP = 11)**

Entry	Variable	Description
39	EY1	Eccentricity in the y-direction at the first vertex, see note 6
40	EZ1	Eccentricity in the z-direction at the first vertex
41	EY2	Eccentricity in the y-direction at the second vertex. see note 6
42	EZ2	Eccentricity in the z-direction at the second vertex
43	DY1	Depth in the y-direction at the first vertex (used for temperature gradient loading), see note 12
44	DZ1	Depth in the z-direction at the first vertex
45	DY2	Depth in the y-direction at the second vertex
46	DZ2	Depth in the z-direction at the second vertex

Table 4.29: Real constant table for beam vertex offset from mesh nodal points
(NKTP = 11)

Entry	Variables	Description
1	ISYS	Key for coordinate system used to define offset between the beam vertices and the corresponding mesh nodal points such that (see note 8): ISYS = 0 – offset vectors are defined in the global Cartesian system ISYS = 1 – offset vectors are defined in the local displacement system at each node ISYS = 2 – offset vectors are defined in the beam local system (see note 15)
2	V_{1x}	X-component of offset at first vertex
3	V_{1y}	Y-component of offset at first vertex
4	V_{1z}	Z-component of offset at first vertex
5	V_{2x}	X-component of offset at second vertex
6	V_{2y}	Y-component of offset at second vertex
7	V_{2z}	Z-component of offset at second vertex

Notes:

1. All cross-section properties are defined in the local coordinate system of the beam. The origin of the local coordinate system is at the centroid of the cross-section at the beam's first vertex. See note 9 for the definition of the local coordinate system of the element.
2. The torsional constant J will default to the polar moment of inertia ($I_{YY} + I_{ZZ}$) at both ends if the corresponding value is entered as zero.
3. The variation of section properties (A , I_{YY} , I_{ZZ} , I_{YZ} , J) between two end nodes along the beam is determined by the variable KEY (first entry in [Table 4.28](#)). Four different types of variation are available as follows:
 - (a) $KEY = 0$ assumes a linear variation in A , I_{YY} , I_{ZZ} , I_{YZ} , and J .
 - (b) $KEY = 1$ assumes a linear variation in A and I_{YY} , a quadratic variation in I_{YZ} , and a cubic variation in I_{ZZ} and J .
 - (c) $KEY = 2$ assumes a linear variation in A and I_{ZZ} , a quadratic in I_{YZ} , and a cubic variation in I_{YY} and J .
 - (d) $KEY = 3$ assumes a quadratic variation in A , a fourth order variation in I_{YY} , I_{ZZ} , I_{YZ} , and J .
4. The section properties (A , I_{YY} , I_{ZZ} , I_{YZ}) at the beam's first node ($X/L = 0$) must be specified. The default value for any unspecified property (excluding I_{YZ}) at the end section (last station, $X/L = 1.0$) is taken from the corresponding value at the beam's first section.
5. The real constant ID number in the *ELEMENTS data group refers to the real constant table given in [Table 4.28](#). This table defines the section properties of the two end sections of the beam, the first of which must be at the beam's first node.
6. The eccentricities EY_i and EZ_i ; $i = 1, 2$ (entries 39 - 42 in [Table 4.28](#)), at either end allow the shear center of the beam to be offset from the centroid. EY_i and EZ_i are the coordinates of the shear center in the beam local coordinate system.
7. The offset vectors (v_1 and v_2 , see [Figure 4.11\(a\)](#)) allow the beam centroid to be offset from the mesh nodal points at either end. They are defined in a separate real constant table ([Table 4.29](#)) which is referenced in the *ELEMENT data group. The offset vector is directed from the mesh node point to the beam centroid at the corresponding beam vertex. The components of each offset vector can be specified in the global Cartesian system if $ISYS = 0$, in the local displacement coordinate system at the corresponding

mesh point if ISYS = 1, or in the beam local coordinate system if ISYS = 2. If the local displacement coordinate system is not specified and ISYS = 1, the components of the offset vector at the corresponding node are assumed in the global directions.

8. The element local coordinate system is defined such that the local x-axis is along the line joining the beam vertices. The local xy plane passes through the local x axis and the local vector which can be either the vector from node 1 to the third point or the vector originating at node 1 and parallel to the orientation vector given in the *ELEMENT data group, see [Figure 4.11](#). The local z axis is parallel to the cross product of the local x axis and the local vector. The local y axis is defined by the right hand rule of Cartesian coordinate system. If beam offset is defined in beam local system (ISYS = 2), the definition of beam orientation will be different. See note 15 for details.
9. End release can be specified in the *ELEMENTS data group to release up to five of six degrees of freedom (DOF), defined in the beam local coordinate system, at either end of the element. A released degree of freedom means that the connection between this DOF and the corresponding nodal point is removed (e.g., to simulate a hinge connection). Singularity problems may arise if the assembled stiffness associated with a released DOF is not properly compensated for from neighboring elements. Therefore, the end release option should be used with caution. For example, if the same DOF is released at the common node of two colinear elements, the released DOF should be constrained to zero. Internally, automatic constraints are applied if all beam elements connected to a common node have all translations and/or all rotations released (at the common node).
10. Partial fixity is a feature to allow partial beam force transfer to the joint (structure node). To activate this feature, the DOF should be an end released DOF and the coefficient of partial fixity should be defined. Two RCTABLEs can be given in the *ELEMENTS data group to define the coefficients of partial fixity at each node. Six coefficients from 0 to .999 can be specified for each DOF of a node in one RCTABLE. If the DOF is not released or the coefficient is greater than .999, there will be no partial fixity effect for the DOF. In case the RCTABLE is not provided, the connection of the joint is determined by the end release code of the node.
11. The stress resultants in the beam local coordinate system (normal force, shear forces, bending moments and twisting moment) are computed at the beam vertices and at all the intermediate stations (if specified). Stresses may be computed for up to four points on the cross-section at any specified station. The transverse shear stresses are computed as average values.
12. Thermal loads can cause axial and/or bending deformation of the beam element. Nodal temperatures (specified in *NDTEMPER data group) are used to compute axial expansion or contraction. Nodal temperature differences (*NDTEMPDIF data group)

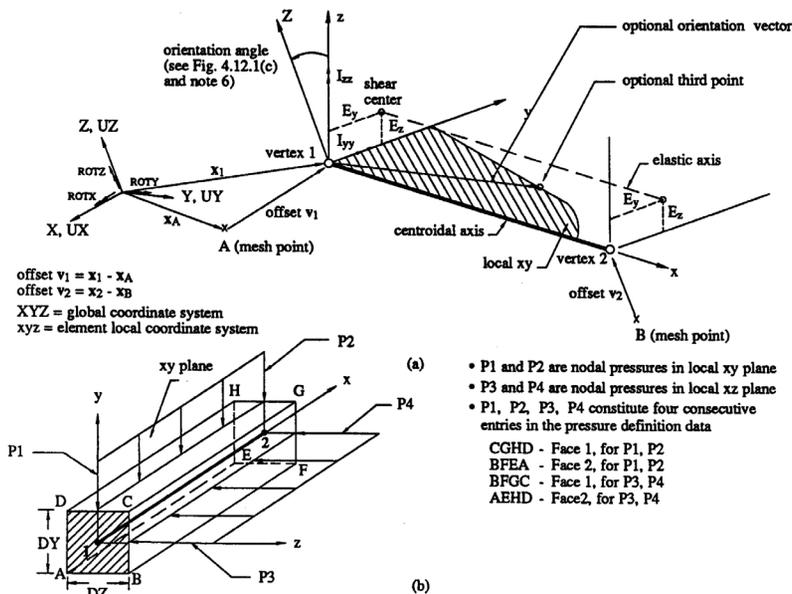
will be used in conjunction with the beam depth in both the y-direction and z-direction (entries 43-46 in [Table 4.28](#)) to compute beam bending in xy and xz planes, respectively. If the temperature gradient loading is to be applied in only one direction, enter zero value for the depth in the other direction.

13. Pressure loading can be applied to any or all of the four faces of the beam element. The face numbering is shown in [Figure 4.11](#). Positive pressure loading is applied towards the face and parallel to the local axes of the beam cross-section. The unit of this loading must be force/length. With reference to [Figure 4.11\(b\)](#), it should be noted that P1, P2, P3, and P4 constitute four consecutive entries in the pressure definition data. Therefore, if the pressure is applied in only one direction, appropriate entries must be set to zero.
14. A partially distributed load, concentrated force or concentrated moment (all defined in the beam local coordinate system) at arbitrary points along the length of the beam element can be specified in the *BEAMLOAD data group.
15. When beam offsets are defined in local system, the orientation of the beam local axes may be different from the way as described in note 9. A local coordinate system has to be defined for beam offsets before the beam orientation can be defined. The third node or a vector is not used to define the beam orientation directly but is used to define the local system. First, a local xy plane is formed by using a third point (or a vector) and the axis from node 1 to node 2. Then the local system is formed by the local xy plane and the local z axis normal to the local xy plane. The value of offsets are specified in local y and z. The beam x axis is along the line joining the beam vertices. The beam xy plane is formed by a vector parallel to local y and the beam x axis. If the beam offsets are uniform, the beam orientation is parallel to local axis.

4.13 3-D Beam Element (NKTP = 12)

Description

This element is a 2-node prismatic 3-D beam element. The formulation includes stretching, bending and torsion effects. The transverse shear deformation effect is included as an option. The beam vertices maybe offset from the corresponding nodal points and the centroid may be offset from the shear center. The element has end release capability, see note 7 below. The deformation is characterized by three translations (UX, UY, UZ) and three rotations (ROTX, ROTY, ROTZ). The local x-axis of the beam is along the centroidal axis. The local y and z axes are user defined and are not necessary principal axes of the cross section. The theoretical basis of the element is discussed in Section 2.8. The element configuration, node locations and face numbering convention are shown in Figure 4.12. An element reference guide is briefed in Table 4.30 with the available loading listed in Table 4.31. A general description of the element input data and output options are given in Section 4.1.2 and Section 4.1.3 respectively. Table 4.30 gives the output pertinent to this element. The required real constants for this element are given in Table 4.32. For a given problem, not all of the entries in the real constant table may need to be specified. Entries not specified will default to zero (except the torsional constant entry 4).



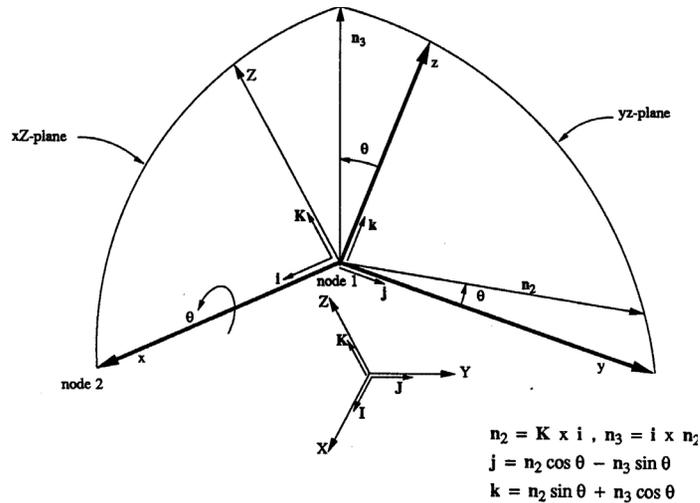


Figure 4.12: 3-D beam element, NKTP = 12, NORDR = 1, (a) Element configuration and orientation, (b) Face numbering convention for pressure loading, (c) Definition of orientation angle (valid when x-axis is not parallel to Z-axis)

Table 4.30: Element reference guide (NKTP = 12)

Element Type	NKTP = 12, 3-D Beam element
Analysis Types	Static, Dynamic, Buckling
Degrees Of Freedom	6 per node: UX, UY, UZ, ROTX, ROTY, ROTZ
NORDR (Shape / No. of Nodes)	Line: 2 nodes (NORDR = 1)
Real Constants	Up to 43 constants, see Table 4.32
Material Properties	
Isotropic Elastic	4 properties: EX, NUXY, DENS, ALPX
Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ, MX, MY, MZ) at beam ends in global coordinate system (or in local displacement system, if defined), and strain energy - Resultant forces and moments at beam ends in beam local coordinate system (FX, FY, FZ, MX, MY, MZ) - Element local stresses (SXX, SXY, SXZ), principal stresses (S1, S2) and maximum shear stress (TAU) at specified points, see note 8

Element Library

3-D Beam Element (NKTP = 12)

Dynamic Capabilities	<ul style="list-style-type: none">- Consistent or lumped mass- Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration, and shock spectrum analyses
Special Feature	<ul style="list-style-type: none">- End release (see note 7)- -Partial fixity (see note 13)

Table 4.31: Available loading (NKTP = 12)

Nodal Loading	<ul style="list-style-type: none">- Concentrated nodal forces in global X, Y and Z directions and concentrated nodal moments about global X, Y and Z axes (or in local displacement system, if defined), see *CFORCE or *DCFORCE- Specified non-zero nodal displacements UX, UY, UZ and nodal rotations ROTX, ROTY, ROTZ in global directions (or in local displacement system, if defined), see *SPDISP- Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
Pressure and Partially Distributed Loading	Uniform or non-uniform pressure (force/length) on top or bottom face of the element; see *PRESSURE, *BEAMLOAD, *DPRESSURE and note 12, 14
Thermal Loading	Specified nodal temperature and nodal temperature difference in element local y and z directions, see *NDTEMPER, *NDTEMPDIF, and note 11
Body Force	<ul style="list-style-type: none">- Gravity loading or body forces due to linear acceleration in global X, Y and Z directions, see *BODYFORCE- Centrifugal loads due to angular velocity about the global X, Y and Z axes, see *BODYFORCE- Tangential loads due to angular acceleration about the global X, Y, and Z axes, see *BODYFORCE
Ground Motion	<ul style="list-style-type: none">- Forces due to ground motion (support excitation) in the global X, Y and Z directions, see *GROUND- Forces due to rotational ground motion in the global XY, YZ and ZX planes about a specified point, see *GROUND

Table 4.32: List of real constants for 3-D beam (NKTP = 12)

Entry	Variable	Description
1	A	Cross-sectional area
2	IYY	Moment of inertia about y-axis ($\int z^2 dA$)
3	IZZ	Moment of inertia about z-axis ($\int y^2 dA$)
4	J	Torsional constant, see note 2
5	EY	Eccentricity in the y-direction, see note 3
6	EZ	Eccentricity in the z-direction
7	DY	Depth in the y-direction, (used for temperature gradient Loading), see note 11
8	DZ	Depth in the z-direction
9	ALPHAY	Shear coefficient in y-direction, see note 4
10	ALPHAZ	Shear coefficient in z-direction
11	V1x	X-component of offset at first node, see note 5
12	V1y	Y-component of offset at first node
13	V1z	Z-component of offset at first node
14	V2x	X-component of offset at second node, see note 5
15	V2y	Y-component of offset at second node
16	V2z	Z-component of offset at second node
17	OKEY	Section orientation key, see note 6
18	THETA or X3	Orientation angle (THETA) or X-coordinate of the third point
19	Y3	0.0 or Y-coordinate of the third point
20	Z3	0.0 or Z-coordinate of the third point
21	SKEY	Number of points on cross-section for stress calculation, $0 \leq NPT \leq 4$, see note 8

Element Library**3-D Beam Element (NKTP = 12)**

Entry	Variable	Description
22	TKEY	Section type key, see note 9, 10
23	YS	y-coordinate of the first stress point
24	ZS	z-coordinate of the first stress point
25	TSF	Torsional shear factor, see note 10
26	SBFY	Shear due to bending factor for y direction
27	SBFZ	Shear due to bending factor for z direction
28	YS	y-coordinate of the second stress point
29	ZS	z-coordinate of the second stress point
30	TSF	Torsional shear factor
31	SBFY	Shear due to bending factor for y direction
32	YS	Shear due to bending factor for z direction
33	ZS	y-coordinate of the third stress point
34	TSF	z-coordinate of the third stress point
35	SBFY	Torsional shear factor
36	SB	Shear due to bending factor for y direction
37	FZ	Shear due to bending factor for z direction
38	YS	y-coordinate of the forth stress point
39	ZS	z-coordinate of the forth stress point
40	TSF	Torsional shear factor
41	SBFY	Shear due to bending factor for y direction
42	SBFZ	Shear due to bending factor for z direction
43	IYZ	Product moment of inertia in yz plane $(\int yz dA)k$
44	LOFST	Key to define offset in the beam local axis = 0 beam offset defined in global coordinates (default) = 1 beam offset defined in beam local coordinates (see note 14)

Notes:

1. Except for the beam offsets and the coordinates of the third point, all other properties are defined in the local coordinate system of the beam element. The origin of the local coordinate system is located at the centroid of the cross-section at the beam's first node. See note 6 for the definition of the element local coordinate system.
2. The torsional constant J will default to the polar moment of inertia ($I_{YY} + I_{ZZ}$) if it is entered as zero.
3. The eccentricities EY and EZ allow the shear center of the beam to be offset from the centroid. EY and EZ are the coordinates of the shear center in the local coordinate system (Figure 4.12). All forces are input at the centroid. Therefore, transverse forces will produce twist about the shear center if the centroid and the shear center are not coincident.
4. The transverse shear coefficients $ALPHAY$ and $ALPHAZ$ account for the non-uniformity of the shear strain over the beam cross section. The shear coefficient defines an effective area for shear as the actual cross-sectional area divided by the shear coefficient. Typical values are:

solid circular cross section	1.111
solid rectangular cross section	1.200
circular tube cross section	1.887
square tube cross section	2.000

For an I-beam, the shear coefficient in the strong direction can be approximated by the ratio of total cross-sectional area to the cross-sectional area of the web.

The shear deformations in y and/or z direction will be suppressed if $ALPHAY$ and/or $ALPHAZ$ are set to zero. For non-zero value of I_{YZ} , the transverse shear deformations are ignored.

5. The offset vectors ($v1$ and $v2$, see Figure 4.12) allow the beam centroid to be offset from the mesh node points at either end. It is defined by the global X , Y and Z components of a vector pointing from the mesh node point to the beam centroid at the corresponding beam vertex.
6. The element local x -axis is along the line joining the beam vertices. The section orientation (the beam local y and z axes) may be defined in the `*ELEMENTS` data group or in the `*RCTABLE` data group. The definition made in the `*ELEMENTS` data group *supersedes*.

A. Define the section orientation in the *ELEMENTS data group:

The local xy plane passes through the beam centroidal axis (local x axis) and the local vector which can be either the vector from node 1 to the third point or the vector originating at node 1 and parallel to the orientation vector given in the *ELEMENT data group, see [Figure 4.12](#). The local z axis is parallel to the cross product of the beam centroidal axis and the local vector. The local y axis is defined by the right hand rule of the Cartesian coordinate system.

B. Define the section orientation in the *RCTABLE data group:

- (a) OKEY = 0 (entry 17 in [Table 4.32](#)), use the orientation angle THETA in degrees, where THETA is defined as follows, see [Figure 4.12](#):
 - (i) Construct a plane containing the element x-axis (which is along the beam centroidal axis and pointing from node 1 to node 2) and a line parallel to the global Z-axis. This will be the xZ-plane.
 - (ii) Find the angle between the element z-axis and the xZ-plane. This will be the positive rotation angle about the element x-axis (right hand rule) necessary to bring the element z-axis onto the xZ-plane.
 - (iii) If the above approach fails because the element x-axis is aligned with the global Z-axis, then steps (i) and (ii) above should be repeated using the global Y-axis to form a xY-plane and θ will be the angle between the element y-axis and the xY-plane.
 - (b) OKEY = 1, the local xy plane is defined by the plane passing through the beam centroidal axis (local x axis) and the third point. The coordinates of the third point (global X3, Y3 and Z3) are defined in entries 18 to 20 of the real constants table.
7. End release can be specified in the *ELEMENTS data group to release up to five of the six degrees of freedom (DOF), defined in the beam local coordinate system, at either end of the element. A released degree of freedom means that the connection between this DOF and the corresponding point is removed (e.g., to simulate a hinge connection). Singularity problems may arise if the assembled stiffness associated with a released DOF is not properly compensated for from neighboring elements. Therefore, the end release option should be used with caution. For example, if the same DOF is released at the common node of two colinear elements, the released DOF should be constrained to zero. Internally, automatic constraints are applied if all beam elements connected to a common node have all translations and/or all rotations released (at the common node).

8. The stress resultants in the beam local coordinate system (normal force, shear forces, bending moments and twisting moment) are computed at the beam ends. In addition, up to 4 points on the cross-section may be specified for stress calculations at the two ends of the beam. No subsequent entries in the real constant table are needed if SKEY is set to zero, unless the product moment of inertia IYZ (entry 43) is non-zero.

9. The variable TKEY (entry 22 in [Table 4.32](#)) selects the type of the cross-section:

TKEY = 0 - solid or thick walled section

TKEY = 1 - open thin walled section

TKEY = 2 - closed thin walled section

Different stress components are printed for different types of sections. Normal stress, shear due to bending, and shear due to torsion are printed for thin walled sections. Whereas normal stress, local shear stresses (SXY, SXZ) are printed for solid or thick walled sections.

10. The torsional constant and the stress factors for different section types may be calculated using the following formulae:

- (a) Solid or thick walled sections:

- (i) For solid sections J can be approximated as:

$$J = \frac{A^4}{40 I_p}$$

where, A is the cross-sectional area and I_p is the polar moment of inertia ($I_p = I_{yy} + I_{zz}$).

- (ii) The torsional shear factor TSF is not used in solid section. The shear stress due to torsion is calculated by

$$\tau = \frac{M_t}{J} r$$

where, r is the radius from the elastic axis to the point (y,z), J is the torsional constant and M_t is the torsional moment.

(iii) The shear due to bending factors SBFY and SBFZ are defined as,

$$SBFY = \left(\frac{I_{yy}Q_y - I_{yz}Q_z}{I_{yy}I_{zz} - I_{yz}^2} \right) \frac{1}{b_y}$$

$$SBFZ = \left(\frac{I_{zz}Q_z - I_{yz}Q_y}{I_{yy}I_{zz} - I_{yz}^2} \right) \frac{1}{b_z}$$

$$Q_y = A_y \bar{y} \quad Q_z = A_z \bar{z}$$

where, Q_y and Q_z are the moment of the area A_y about the z-axis and the moment of the area A_z about the y-axis, respectively. b_z and b_y are the widths of the section at the point (y,z) normal to the y and z axes, respectively, see [Figure 4.13](#).

(b) Thin walled open sections:

(i) The torsional constant J can be calculated as,

$$J = 1/3 \sum_{i=1}^n b_i t_i^3$$

where, b_i is the length of the ith segment, t_i is the thickness and n is the number of segments in the section, see [Figure 4.14](#).

(ii) The torsional shear factor is defined as,

$$\tau = \frac{M_t}{J} TSF$$

and

$$TSF = t$$

where, M_t is the torsional moment, J is the torsional constant and t is the thickness at the point (y,z). For points located at the fillets with the radius r, $TSF = 1.74(t/r)^4$ can be used to take into account stress concentration at the fillets, see [4.4].

- (iii) The shear due to bending factors SBFY and SBFZ are defined as,

$$SBFY = \frac{1}{t} \left(\frac{I_{yy}Q_y - I_{yz}Q_z}{I_{yy}I_{zz} - I_{yz}^2} \right)$$

$$SBFZ = \frac{1}{t} \left(\frac{I_{zz}Q_z - I_{yz}Q_y}{I_{yy}I_{zz} - I_{yz}^2} \right)$$

where, Q_y and Q_z are the moments of the area A about the z and y axes respectively and t is the thickness of the segment at the point (y,z), see [Figure 4.14](#).

- (c) Thin walled closed sections (single-cell tube):

- (i) The torsional constant J can be calculated as,

$$J = 4A_s^2 / \int dU/t$$

where, A_s is the area enclosed by the centerline of the tube wall, U is the length of the centerline of the tube and t is the thickness at any point in the section, see [Figure 4.15](#).

- (ii) The torsional shear factor TSF for closed section is defined as,

$$\tau = M_t \cdot TSF$$

and

$$TSF = \frac{1}{2A_s t}$$

where, M_t is torsional moment, A_s is the area enclosed by the centerline of the tube wall and t is the thickness of the tube wall at the point (y,z).

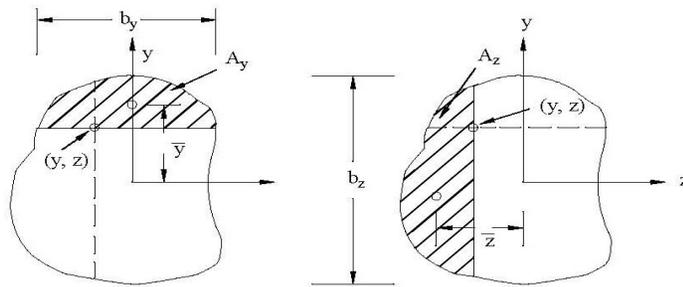


Figure 4.13: Solid or thick walled sections (3-D beam, NKTP = 12)

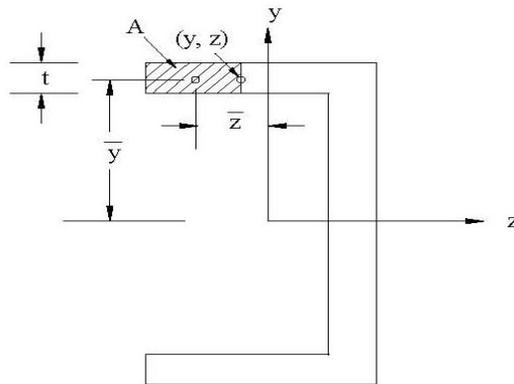


Figure 4.14: Thin walled open section (3-D beam, NKTP = 12)

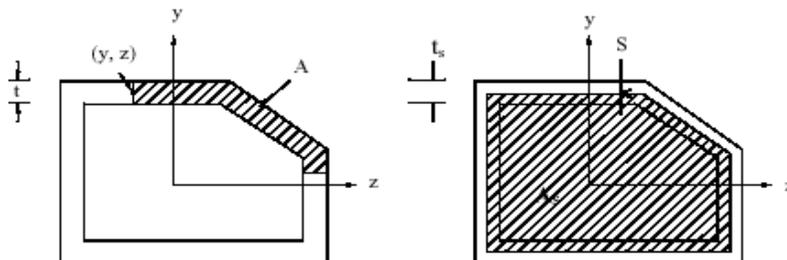


Figure 4.15: Thin walled closed section (3-D beam, NKTP = 12)

(iii) The shear due to bending factors SBFY and SBFZ are defined as,

$$SBFY = \frac{1}{t} \left(\frac{I_{yy}(Q_y - K_y) - I_{yz}(Q_z - K_z)}{I_{zz} - I_{yz}^2} \right)$$

$$SBFZ = \frac{1}{t} \left(\frac{I_{zz}(Q_z - K_z) - I_{yz}(Q_y - K_y)}{I_{yy}I_{zz} - I_{yz}^2} \right)$$

where, Q_y and Q_z are the moments of the area A about the z and y axes, respectively; t is the thickness of the section at the point (y,z), see [Figure 4.15](#). K_z and K_y are defined as,

$$K_z = \frac{\int_s Q_z(s) \frac{ds}{t_s}}{\int_s \frac{ds}{t_s}}$$

$$K_y = \frac{\int_s Q_y(s) \frac{ds}{t_s}}{\int_s \frac{ds}{t_s}}$$

where, \int_s denotes the integral around the closed curve of the section center line, and

$Q_y(s)$ and $Q_z(s)$ are the moments of the area A_s about the z and y axes, respectively. t_s is the thickness at the distance s measured in the counterclockwise direction, see [Figure 4.15](#)

11. Thermal loads can cause axial and/or bending deformation of the beam element. Nodal temperatures (specified in *NDTEMPER data group) are used to compute axial expansion or contraction. Nodal temperature differences (*NDTEMPDIF data group) will be used in conjunction with the beam depth in both the y-direction and z-direction (DY and DZ, the 7-th and 8-th entries in the real constant table) to compute beam bending in the local xy and xz planes, respectively. If the temperature gradient loading is to be applied in only one direction, enter zero value for the depth in the other direction.

12. Pressure loading can be applied to any or all of the four faces of the beam element. The face numbering is shown in [Figure 4.12](#). Positive pressure loading is applied toward to the face and parallel to the local axes of the beam cross-section. The unit of this loading must be force/length. With reference to [Figure 4.12\(b\)](#), it should be noted that P1, P2, P3, and P4 constitute four consecutive entries in the pressure definition data. Therefore, if the pressure is applied in only one direction, appropriate entries must be set to zero.
13. Partial fixity is a feature to transfer beam end forces partially, instead of full forces to the joints (structure node) to simulate a plastic hinge. To activate this feature, the degree of freedom of the beam has to be released using end release code. A RCTABLE has to be specified for the six coefficients of partial fixity for each DOF of each beam node. If no RCTABLE is given for that node, the release DOFs are treated as an end release. For those DOFs without end release, the specified coefficients will have no effect. The coefficient of partial fixity is the ratio of the transferred beam force to the full force. It can vary from 0 to 1. 0 means end release and 1 means normal connection without partial fixity. To avoid numerical problem, any coefficient is greater than 0.999 will be treated as normal connection without partial fixity.
14. A partially distributed load, concentrated force or concentrated moment (all defined in the beam local coordinate system) at arbitrary points along the length of the beam element can be specified in the *BEAMLOAD data group.
15. When beam offsets are defined in local system, the orientation of the beam local axes may be different from the way as described in note 6. A local coordinate system has to be defined for beam offsets before the beam orientation can be defined. The third node or a vector is not used to define the beam orientation directly but is used to define the local system. First, a local xy plane is formed by using a third point (or a vector) and the axis from node 1 to node 2. Then the local system is formed by the local xy plane and the local z axis normal to the local xy plane. The value of offsets are specified in local y and z. The beam x axis is along the line joining the beam vertices. The beam xy plane is formed by a vector parallel to local y and the beam x axis. If the beam offsets are uniform, the beam orientation is parallel to local axis.

4.14 2-D Beam Element (NKTP = 13)

Description

The formulation of this 2-D prismatic beam element is similar to 3-D beam (NKTP = 12), except that this element is restricted to use in 2-D problems. The element should lie in the global XY plane. The formulation includes stretching and bending effects. The transverse shear deformation is included as an option. The deformation is characterized by two translational components of displacement (UX, UY) and one component of rotation (ROTZ). The theoretical basis of the element is discussed in [Section 2.8](#).

The element configuration, node locations and face numbering convention are shown in [Figure 4.16](#). An element reference guide is briefed in [Table 4.33](#) with the available loading listed in [Table 4.34](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.1](#) gives the output pertinent to this element.

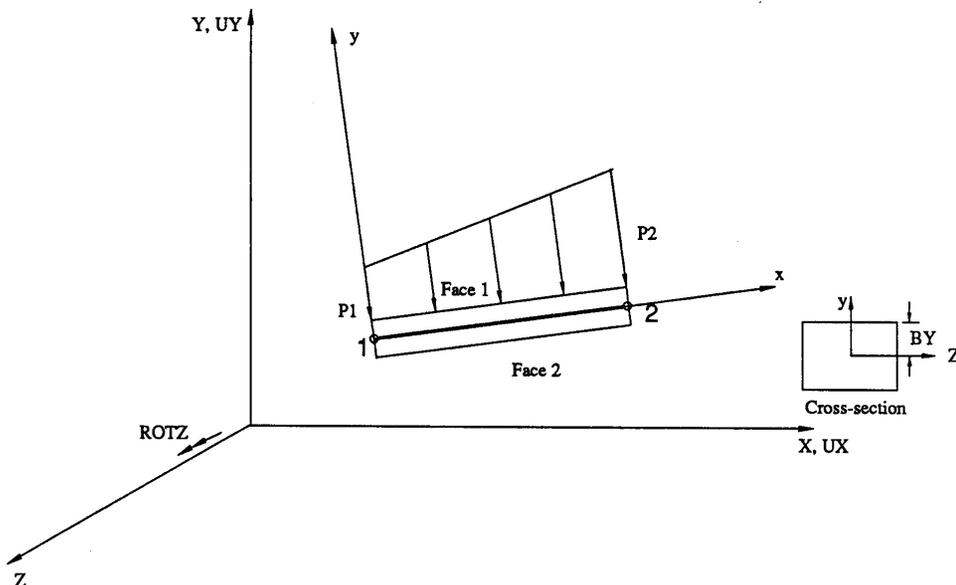


Figure 4.16: Element configuration and face numbering convention for 2-D beam (NKTP = 13, NORDR = 1)

Table 4.33: Element reference guide (NKTP = 13)

Element Type	NKTP = 13, 2-D Beam element
Analysis Types	Static, Dynamic, Buckling
Degrees of Freedom	3 per node: UX, UY, ROTZ
NORDR (Shape / No. of nodes)	- Line: 2 nodes (NORDR = 1)
Real Constants	4 constants : A, IZZ, ALPHAY, BY where, A : cross-sectional area IZZ : moment of inertia about Z-axis ($\int y^2 dA$) ALPHAY : transverse shear coefficient, see note 1 BY : local y-coordinate for stress recovery, default = 1.0, see Figure 4.16
Material Properties	
Isotropic elastic	4 properties: EX, NUXY, DENS, ALPX
Element Output	- Internal forces (FX, FY, MZ) at beam ends in global coordinate system (or in local displacement system, if defined), and strain energy - Resultant forces (FX, FY, MZ), and stresses due to stretching and bending in beam local coordinate system
Dynamic Capabilities	- Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration, and shock spectrum analyses

Table 4.34: Available loading (NKTP = 13)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X, and Y directions and concentrated nodal moment about global Z axis (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal displacements UX, UY and nodal rotation ROTZ in global directions (or in local displacement system, if defined), see *SPDISP - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
Pressure and Partially Distributed Loading	<ul style="list-style-type: none"> - Uniform or non-uniform pressure (force/length) on top or bottom face of the element, see *PRESSURE, *BEAMLOAD, *DPRESSURE and note 2, 3
Thermal Loading	<ul style="list-style-type: none"> - Specified nodal temperature see *NDTEMPER
Body Force	<ul style="list-style-type: none"> - Gravity loading or body force due to linear acceleration in global X, and Y directions, see *BODYFORCE - Centrifugal loads due to angular velocity about the global X and Y axes, or the global Z axis, see *BODYFORCE and note 5 - Tangential loads due to angular acceleration about the global Z-axis, see *BODYFORCE
Ground Motion	<ul style="list-style-type: none"> - Forces due to ground motion (support excitation) in the global X and Y directions, see *GROUND - Force due to rotational ground motion in the global XY plane about a specified point, see *GROUND

Notes:

1. The transverse shear coefficient ALPHAY accounts for the non uniformity of the shear strain over the depth of the beam section. Setting ALPHAY = 0 eliminates the effect of transverse shear deformation. Refer to note 4 in [Section 4.13](#) for suggested values of ALPHAY.
2. Non uniform or uniform pressure can be applied to the top or bottom face. Pressure should be entered in units of force/length. Positive pressure is assumed acting toward to the face, see [Section 4.16](#) for face numbering convention.

Element Library

2-D Beam Element (NKTP = 13)

3. Partially distributed load, concentrated force or concentrated moment (all defined in the beam local coordinate system) at arbitrary points along the length of the beam can be specified in *BEAMLOAD data group.
4. Thermal loads may be applied to this element. However, the effect of a temperature gradient through the thickness of the beam is not accounted for.
5. Body forces due to angular velocity (i.e., centrifugal loads) should be applied such that the spin vector either lies in the global XY plane or is aligned with the global Z-axis.

4.15 3-D Spar Element (NKTP = 14)

Description

This element is a 2-node uniaxial tension-compression 3-D line element and is used to model space trusses. The element may be oriented anywhere in space. The element has three translational degrees of freedom per node (UX, UY, UZ). The theoretical basis of the element is discussed in [Section 2.8](#).

The element configuration is shown in [Figure 4.17](#). An element reference guide is given in [Table 4.35](#) with the available loading listed in [Table 4.36](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.35](#) gives the output pertinent to this element.

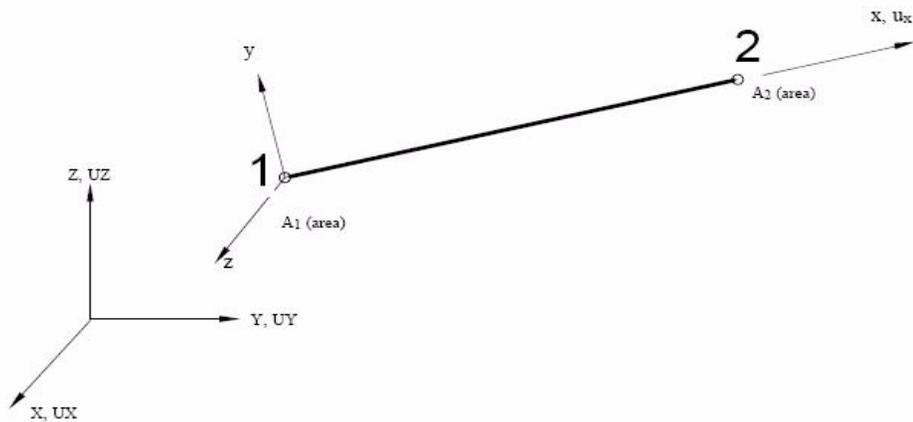


Figure 4.17: Element configuration (NKTP = 14, NORDR = 1)

Table 4.35: Element reference guide (NKTP = 14)

Element Type	NKTP = 14, 3-D Spar element
Analysis Types	Static, Dynamic, Buckling, Nonlinear Static, Nonlinear Transient
Degrees of Freedom	3 per node: UX, UY, UZ
NORDR (Shape / No. of nodes)	- Line: 2 nodes (NORDR = 1)
Real Constants	<p>5 constants : $A_1, A_2, TENSIN, TCKEY, C$</p> <p>where,</p> <p>$A_1, A_2$: Cross sectional areas at node 1 and 2</p> <p>TENSIN : Initial tension in the element (for nonlinear analysis only)</p> <p>TCKEY : Key to define the type of the member (for nonlinear analysis only)</p> <p>= 0 (default) tension-compression member</p> <p>= 1 tension only member</p> <p>= -1 compression only member</p> <p>C : Artificial stiffness for tension only member under compression or compression only member under tension</p>
Material Properties	
Isotropic Elastic	3 properties: EX, DENS, ALPX
Elastoplastic	For nonlinear static only, see *PLASTIC
Creep Law	For nonlinear static only, see *CREEP

Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ) at the two ends in the global coordinate system (or in local displacement system if defined), and strain energy - Axial force (FX) and axial stress in element local coordinate system - Element stresses at nodal points in element local coordinate system - Equivalent plastic strain (EPS), and effective yield stress, for nonlinear static only, at nodal points - Second Piola-Kirchhoff or Cauchy stresses, for nonlinear static only, at nodal points
Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration, and shock spectrum analyses
Nonlinear Capabilities	<ul style="list-style-type: none"> - Geometric nonlinearity: Large displacements, rotations and strains (for plasticity analysis, only small strains are assumed); total and updated Lagrangian formulation; deformation dependent loads - Material nonlinearity: elastoplastic material model

Table 4.36: Available loading (NKTP = 14)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X, Y, Z directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal displacements UX, UY, UZ in global X, Y and Z directions (or in local displacement system, if defined), see *SPDISP - Concentrated nodal follower (deformation dependent) forces perpendicular to the axis of the element, for nonlinear static only. See *FOLLOWER and note 2. - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
Thermal Loading	<ul style="list-style-type: none"> - Specified nodal temperature, see *NDTEMPER
Body Force	<ul style="list-style-type: none"> - Gravity loading or body forces due to linear acceleration in global X, Y, Z directions, see *BODYFORCE - Centrifugal loads due to angular velocity about the global X, Y and Z directions, see *BODYFORCE - Tangential loads due to angular acceleration about the global X, Y and Z axes, see *BODYFORCE

Element Library

3-D Spar Element (NKTP = 14)

Ground Motion	<ul style="list-style-type: none">- Forces due to ground motion (support excitation) in the global X, Y and Z directions, see *GROUND- Forces due to rotational ground motion in the global XY, YZ, and ZX planes about a specified point, see *GROUND
----------------------	---

Notes:

1. This element has three real constants entered in the *RCTABLE data group. The first two constants are the cross-sectional areas at the first and second nodes of the element, respectively. The third real constant is the initial tension in the element and is only applicable for nonlinear static analysis.
2. To calculate the direction of the concentrated follower force, the following approximation is assumed:
 - The third point required in *CFOLLOWER will define the local y- and z-axes in the initial undeformed configuration such that nodes 1, 2 and the third point will be in the xy plane.
 - In any subsequent deformed configuration, the local z-axis will be defined as the cross product of two unit vectors aligned with the current and the previous x-axis, and the local y-axis is the cross product of two unit vectors aligned with the local z- and x-axes.

4.16 2-D Spar Element (NKTP = 15)

Description

This element is a 2-node uniaxial tension-compression 2-D line element and is used to model two-dimensional trusses. The element may be oriented anywhere in the global XY plane. The element has two translational degrees of freedom per node (UX, UY). The theoretical basis of the element is discussed in [Section 2.8](#).

The element configuration is shown in [Figure 4.18](#). An element reference guide is given in [Table 4.37](#) with the available loading listed in [Table 4.38](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.37](#) gives the output pertinent to this element.

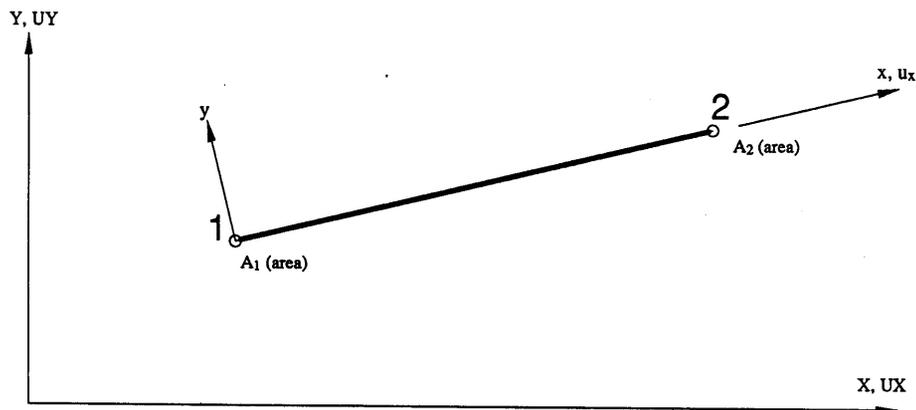


Figure 4.18: Element configuration (NKTP = 15, NORDR = 1)

Table 4.37: Element reference guide (NKTP = 15)

Element Type	NKTP = 15, 2-D Spar element
Analysis Types	Static, Dynamic, Buckling
Degrees of Freedom	2 per node: UX, UY
NORDR (Shape / No. of nodes)	Line: 2 nodes (NORDR = 1)
Real Constants	2; Cross sectional areas at node 1 and 2, respectively
Material Properties	
Isotropic Elastic	3 properties: EX, DENS, ALPX
Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY) at the two ends in global coordinate system (or in local displacement system, if defined), and strain energy - Axial force (FX) and axial stress in element local coordinate system
Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration, and shock spectrum analyses

Table 4.38: Available loading (NKTP = 15)

Body Force	<ul style="list-style-type: none"> - Gravity loading or body forces due to linear acceleration in global X, Y directions, see *BODYFORCE - Centrifugal loads due to angular velocity about the global X and Y axes, or about the global Z axis, see *BODYFORCE and note 2 - Tangential loads due to angular acceleration about the global Z-axis, see *BODYFORCE
Ground Motion	<ul style="list-style-type: none"> - Forces due to ground motion (support excitation) in the global X and Y directions, see *GROUND - Forces due to rotational ground motion in the global XY plane about a specified point, see *GROUND

Notes:

1. This element has *two* real constants which must be entered in the *RCTABLE data group. They are the cross-sectional areas at the first and second nodes of the element, respectively.
2. Body forces due to angular velocity (i.e., centrifugal loads) should be applied such that the spin vector either lies in the global XY plane or is aligned with the global Z-axis.

4.17 3-D Translational Spring Element (NKTP = 17)

Description

This element is a 2-node uniaxial tension-compression massless spring in three dimensions. The element may be oriented anywhere in space. The element has three translational degrees of freedom per node (UX, UY, UZ). The two nodes defining the spring may be coincident, in which case a direction vector is specified indicating the direction of the spring axis.

The element configuration is shown in [Figure 4.19](#). An element reference guide is given in [Table 4.39](#) with the available loading given in [Table 4.40](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.39](#) gives the output pertinent to this element.

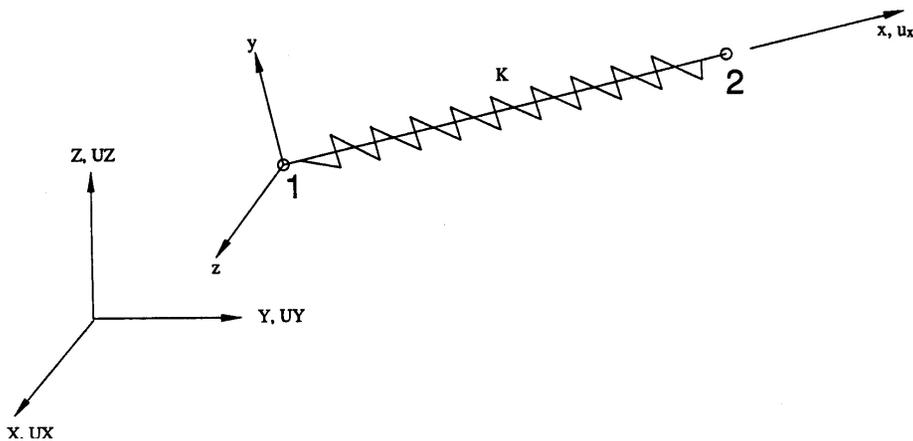


Figure 4.19: Element configuration (NKTP = 17, NORDR = 1)

Table 4.39: Element reference guide (NKTP = 17)

Element Type	NKTP = 17, 3-D Translational spring element
Analysis Types	Static, Dynamic, Buckling, Nonlinear Static, Nonlinear Transient, Direct frequency analysis, see note 1
Degrees of Freedom	2 per node: UX, UY
NORDR (Shape / No. of nodes)	- Line: 2 nodes (NORDR = 1)
Real Constants (RCTABLE)	<p>Up to 7 constants: K, V₁, V₂, V₃, ISYS, IDSP, DEFINI Where,</p> <p>K Axial spring constant (force/length) for linear analysis = 0 for nonlinear and direct frequency analysis except if IDSP = 0, see note 3.</p> <p>V_i V_i = 1, 2, 3; required only if the two nodes are coincident; components of a vector pointing in the spring axis direction</p> <p>ISYS Coordinate system in which the components V_i, i = 1, 2, 3, are given = 0 global cartesian system = 1 local displacement coordinate system at first node, see note 2 = 2 local displacement coordinate system at second node, see note 2</p> <p>IDSP Non linear spring identification number Frequency function ID of the frequency dependent spring constant for direct frequency analysis</p> <p>DEFINI Initial Deflection</p>
Material Properties	Material ID (optional); see note 4
Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ) at the two ends in global coordinate system (or in local displacement system, if defined), and strain energy - Axial force (FX)

Dynamic Capabilities	<ul style="list-style-type: none"> - Null mass matrix is assumed, see note 1 - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration and shock spectrum analyses
Nonlinear Capabilities	<ul style="list-style-type: none"> - Follows nonlinear force-deflection curve (piecewise linear, spline or polynomial curve fit) - Conservative or nonconservative loading (see *NLSRING data group and Section 2.8.2)

Table 4.40: Available loading (NKTP = 17)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X, Y, and Z directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal displacements UX, UY, UZ in global X, Y, and Z directions (or in local displacement system, if defined), see *SPDISP - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
----------------------	--

Notes:

1. This spring element is a massless element. It may be used in dynamic analyses provided that it is connected to other elements with non-zero masses. Similarly, the element may be used in buckling analysis (a null geometric stiffness matrix is assumed), provided it is connected to other elements with non-zero geometric stiffness matrices.
2. If the two nodes of the element are not coincident, only one real constant, i.e., the torsional spring constant, is required, otherwise ISYS is set to zero. If the two nodes are coincident, a direction vector (v_i , $i = 1, 2, 3$) is required. The direction vector is described in the coordinate system indicated by ISYS. If ISYS = 1 or 2, it is assumed that the local displacement coordinate system is defined at the referenced node (node 1 or node 2).
3. To use a linear spring in nonlinear analysis, set IDSP in the Real Constant table to zero. In this case, the program uses the spring constant K (which should not be zero) in the Real Constant Table as a constant stiffness in the entire nonlinear analysis.
4. Material damping in the modal dynamic analysis can be applied to this spring element. For the elements with material damping, a material ID should be specified in eigenvalue analysis to be used later in the modal dynamic analysis. Even though material properties are not required for the spring element, a dummy material property

table should be given for the material ID used for the spring element. For any other analysis without material damping, the material ID for the spring element is not required.

4.18 2-D Translational Spring Element (NKTP = 18)

Description

This element is a 2-node uniaxial tension-compression massless spring in two dimensions. The element must lie in the global XY plane. The element has two translational degrees of freedom per node (UX, UY). The two nodes defining the spring may be coincident, in which case a direction vector is specified indicating the direction of the spring axis.

The element configuration is shown in [Figure 4.20](#). An element reference guide is given in [Table 4.41](#) with the available loading listed in [Table 4.42](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.41](#) gives the output pertinent to this element.

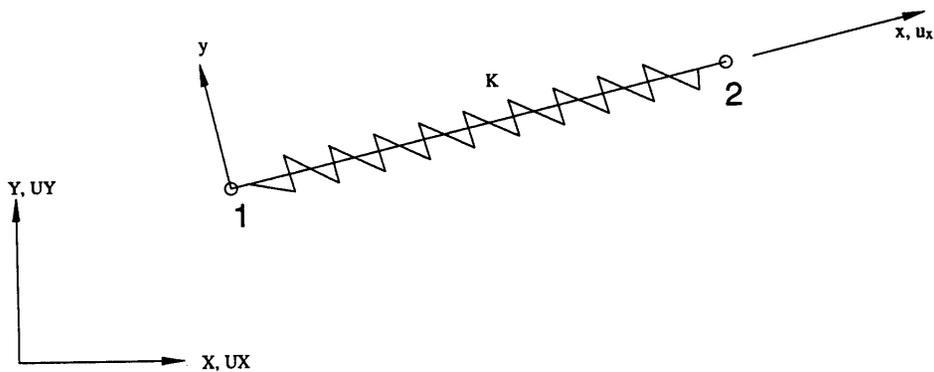


Figure 4.20: Element configuration (NKTP = 18, NORDR = 1)

Table 4.41: Element reference guide (NKTP = 18)

Element Type	NKTP = 18, 2-D Translational spring element
Analysis Types	Static, Dynamic, Buckling, see note 1
Degrees of Freedom	2 per node: UX, UY
NORDR (Shape / No. of nodes)	- Line: 2 nodes (NORDR = 1)
Real Constants (RCTABLE)	Up to 5 constants: K, V ₁ , V ₂ , V ₃ , ISYS Where, K Axial spring constant (force/length) V _i V _i = 1, 2, 3; required only if the two nodes are coincident; components of a vector pointing in the spring axis direction (V ₃ must be zero) ISYS Coordinate system in which the components V _i , i = 1, 2, 3, are given = 0 global cartesian system = 1 local displacement coordinate system at first node, see note 2 = 2 local displacement coordinate system at second node, see note 2
Material Properties	Material ID (optional); see note 3
Element Output	- Internal forces (FX, FY) at the two ends in global coordinate system (or in local displacement system, if defined), and strain energy - Axial force (FX)
Dynamic Capabilities	- Null mass matrix is assumed, see note 1 - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration and shock spectrum analyses

Table 4.42: Available loading (NKTP = 18)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X and Y directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal displacements UX, UY in global X and Y directions (or in local displacement system, if defined), see *SPDISP - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
----------------------	--

Notes:

1. This spring element is a massless element. It may be used in dynamic analyses provided that it is connected to other elements with non-zero masses. Similarly, a null geometric stiffness matrix is assumed in buckling analysis, and the element must be connected to other elements having non-zero geometric stiffness matrices.
2. If the two nodes of the element are not coincident, only one real constant, the axial spring constant, is required. If the two nodes are coincident, a direction vector (V_i , $i = 1, 2, 3$) is required ($V_3 = 0$). The direction vector is described in the coordinate system indicated by ISYS. If ISYS = 1 or 2, it is assumed that the local displacement coordinate system is defined at the referenced node (node 1 or 2), otherwise ISYS is reset to zero.
3. Material damping in the modal dynamic analysis can be applied to this spring element. For the elements with material damping, a material ID should be specified in eigenvalue analysis to be used later in the modal dynamic analysis. Even though material properties are not required for the spring element, a dummy material property table should be given for the material ID used for the spring element. For any other analysis without material damping, the material ID for the spring element is not required.

4.19 3-D General Shell Element (NKTP = 20)

Description

The 3-D general shell element includes membrane, bending and transverse shear deformation effects (three node triangular element is an exception, see note 3), and is suited for modeling moderately thick to thin shell structures. The element has six degrees of freedom per node (UX, UY, UZ, ROTX, ROTY, ROTZ), but it possesses no rotational stiffness about the normal to the shell surface, see note 4. The offset of shell vertices from the mesh nodal points are allowed in this shell element for all types of analysis in the linear domain, such as, the linear static, dynamic, buckling, and direct frequency analyses. The theoretical basis of the element is discussed in [Section 2.8](#). An element reference guide is briefed in [Table 4.43](#) with the available loading given in [Table 4.44](#).

The element can be shaped as a 4 to 12 node quadrilateral, or a 3 or 6 node triangle depending on the selected NORDR value. The element configuration, node locations and face numbering convention for top and bottom surfaces are shown in [Table 4.43](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.43](#) lists the output pertinent to this element.

Element Library

3-D General Shell Element (NKTP = 20)

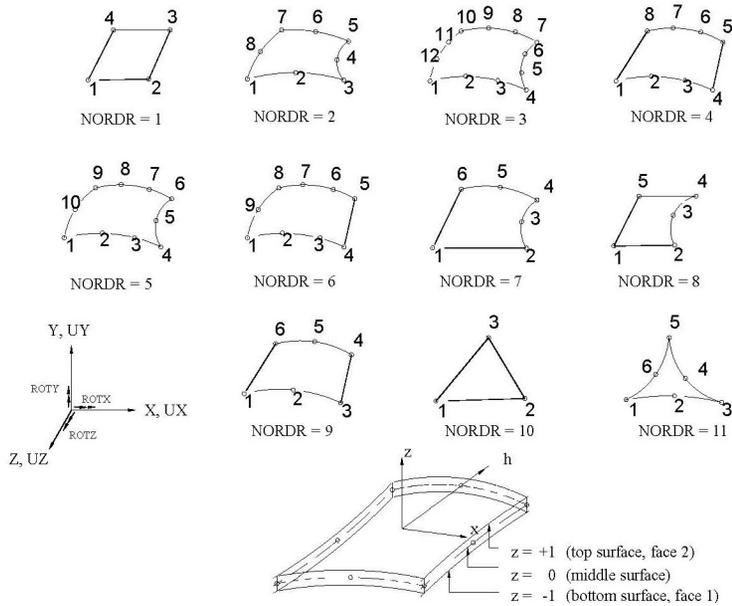


Figure 4.21: Element configuration and face numbering convention for top and bottom surfaces

Table 4.43: Element reference guide (NKTP = 20)

Element Type	NKTP = 20, 3-D General shell element
Analysis Types	Static, Eigen, Modal Dynamic, Buckling, Direct Frequency
Degrees of Freedom	6 per node: UX, UY, UZ, ROTX, ROTY, ROTZ
NORDR (Shape / No. of nodes)	<ul style="list-style-type: none"> - Quadrilateral: 4 to 12 nodes (NORDR = 1 to 9) - Triangle: 3 or 6 nodes (NORDR = 10, 11) - (For nonlinear analysis, only NORDR = 1,2 and 10 are currently available and NORDR = 10 is NORDR = 1 element by collapsing node3 and node4)
Real Constants	<p>First 12 entries: 3 to 12 nodal thickness (same as number of nodes) 13th entry: 0 - No offset 1 - Bottom surface is offset to the mid-plane 2 - Top surface is offset to the mid-plane 3 - User defined offset</p> <p>If the 13th entry is '3', then the next (up to) 12 entries correspond to the nodal offset values. It should be noted that the offset option should not be used for nonlinear problems.</p>
Material Properties	
Isotropic Elastic	4 properties: EX, NUXY, DENS, ALPX
Orthotropic Elastic	9 properties: EX, EY, NUXY, GXY, GYZ, GXZ, DENS, ALPX, ALPY (see note 5)
Elastoplastic	For nonlinear static only, see *PLASTIC
Creep Law	For nonlinear static only, see *CREEP

<p>Element Output</p>	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ, MX, MY, MZ) and strain energy - Centroidal strains at the top, middle and bottom surface in element local directions (EXX, EYY, EXY, EYZ, EXZ) and/or in global directions (EXX, EYY, EZZ, EXY, EYZ, EXZ) (not available for nonlinear static analysis) - Element stresses at centroid, Gauss and nodal points for top, middle and bottom surfaces in element local directions (SXX, SYY, SXY, SYZ, SXZ) and in global directions (SXX, SYY, SZZ, SXY, SYZ, SXZ) - Element principal stresses (S1, S2, S3), von Mises equivalent stress, octahedral and maximum shear stresses at centroid, Gauss and nodal points for top, middle and bottom surfaces - Equivalent plastic strain (EPS), and effective yield stress (YLD), for nonlinear static only, at Gauss and nodal points, see note 7 - Second Piola-Kirchhoff or Cauchy stresses, for nonlinear static only, at Gauss and nodal points - Stress resultants (NXX, NYY, NXY, MXX, MYY, MXY, QYZ, QXZ) at nodal points. (see note 8)
<p>Dynamic Capabilities</p>	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration and shock spectrum analyses
<p>Nonlinear Capabilities</p>	<ul style="list-style-type: none"> - Geometric nonlinearity, large displacement and rotation; total Lagrangian formulations; deformation dependent loads - Material nonlinearity; elasto plastic material model

Table 4.44: Available loading (NKTP = 20)

<p>Nodal Loading</p>	<ul style="list-style-type: none"> - Concentrated nodal forces in global X, Y and Z directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal displacements UX, UY, UZ and nodal rotations ROTX, ROTY, ROTZ in global directions (or in local displacement system, if defined), see *SPDISP - Concentrated nodal follower (deformation dependent) forces perpendicular to the middle surface of the element, for nonlinear static only, see *CFOLLOWER - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
-----------------------------	--

Pressure Loading	<ul style="list-style-type: none"> - Uniform or non-uniform pressure (force/area) on top or bottom faces of the element, see *PRESSURE or *DPRESSURE and note 2 - Uniform or non-uniform follower (deformation dependent) pressure (force/area) on top or bottom faces of the element, for nonlinear static only, see *PRESSURE
Thermal Loading	<ul style="list-style-type: none"> - Specified nodal temperature and nodal temperature difference in element thickness direction, see *NDTEMPER and *NDTEMPDIF
Body Force	<ul style="list-style-type: none"> - Gravity loading or body forces due to linear acceleration in global X, Y and Z directions, see *BODYFORCE - Centrifugal loads due to angular velocity about the global X, Y and Z directions, see *BODYFORCE - Tangential loads due to angular acceleration about the global X, Y and Z directions, see *BODYFORCE
Ground Motion	<ul style="list-style-type: none"> - Forces due to ground motion (support excitation) in the global X, Y and Z directions, see *GROUND - Forces due to rotational ground motion in the global XY, YZ and ZX planes about a specified point, see *GROUND

Notes:

1. The element may be oriented anywhere in space. The element connectivity must be given in the order shown in [Figure 4.21](#), in which the node numbering sequence starts at a corner node and proceeds along the perimeter of the element.
2. The bottom and top faces of the element are numbered as face 1 and 2 respectively ([Figure 4.21](#)). Top and bottom faces are determined as follows:
 - (a) Choose a viewpoint so that the element is defined counterclockwise.
 - (b) The top face is now closest to you.

For pressure loading, the pressure (force/area) is integrated over the area of the loaded face. For curved shells, this area will be somewhat different than the area of the shell midsurface.

3. Transverse shear effects are included in the element formulation. An exception is the linear triangular element (NORDR = 10) which is based on classical (Kirchhoff) shell theory. The linear triangular element is not isoparametric. Linear polynomials are used

to represent the inplane deformations, while constrained cubic polynomials are used for the bending deformations.

4. This element possesses no rotational stiffness about the normal to the shell surface. To avoid any possible difficulties, set AUTO = ON in the executive commands. A small rotational stiffness is then added at the nodes where the shell surface has continuous curvature.
5. For orthotropic materials, the material principal axes should be defined in such a way that the material Z-axis is normal to the middle surface of the element. If the X and Y material principal axes are not tangent to the middle surface, the local material axes are chosen such that the material Z-axis is normal to the shell and the revised system is closest to the user defined system. For geometrically complex shell structures, it may be advantageous to use laminated composite shell elements (NKTP = 32) even for the case of single layer shells, because of the ease with which material directions can be specified.
6. Stresses can be calculated in both local and global systems, at the centroid, node and/or Gauss points for the top, middle and bottom surfaces. The direction cosines of local directions, which are tangent to the shell midsurface, are also printed along with the local stress components. The inplane components of stress (SXX, SYY, SXY) vary linearly through the thickness and reach their maximum values at either the top or bottom surface. The transverse shear stresses (SXZ, SYZ) are zero on the top and bottom surfaces and maximum at the middle surface.
7. For nonlinear static analysis, the nodal equivalent plastic strains (EPS) are approximated by extrapolating the Gauss point equivalent plastic strains.
8. Eight components of stress resultants are defined as,

$$\begin{aligned} NXX &= \int \sigma_{xx} dz, NYY = \int \sigma_{yy} dz, NXY = \int \sigma_{xy} dz, MXX = \int \sigma_{yy} z dz \\ MYY &= \int \sigma_{xx} z dz, MXY = \int \sigma_{xy} z dz, QYZ = \int \sigma_{xz} dz, QXZ = \int \sigma_{yz} dz. \end{aligned}$$

These resultants are computed in the element local coordinate system specified in NRCS of *ELEMENT data group. The positive resultant moments MXX and MYY are chosen in the direction of positive rotation about X and Y axes. The averaged nodal stress resultants are the average of all local stress resultants at the node.

Table 4.45: Element reference guide (NKTP = 21)

Element Type	NKTP = 21, 3-D Torsional spring element
Analysis Types	Static, Dynamic, Buckling, Nonlinear Static and dynamics, Direct frequency analysis
Degrees of Freedom	3 per node: ROTX, ROTY, ROTZ
NORDR (Shape / No. of nodes)	- Line: 2 nodes (NORDR = 1)
Real Constants (RCTABLE)	<p>Up to 7 constants: K, V₁, V₂, V₃, ISYS, IDSP, DEFINI Where,</p> <p>K Torsional spring constant (force/length) for linear analysis. = 0 for nonlinear and direct frequency analysis except if IDSP = 0, see note 4.</p> <p>V_i V_i = 1, 2, 3; required only if the two nodes are coincident; components of a vector pointing in the spring axis direction</p> <p>ISYS Coordinate system in which the components V_i, i = 1, 2, 3, are given = 0 global Cartesian system = 1 local displacement coordinate system at first node, see note 2 = 2 local displacement coordinate system at second node, see note 2</p> <p>IDSP Non linear spring identification number Frequency function ID of the frequency dependent spring constant for direct frequency analysis</p> <p>DEFINI Initial Deflection</p>
Material Properties	Material ID (optional); see note 3
Element Output	<ul style="list-style-type: none"> - Internal moments (MX, MY, MZ) at the two ends in global coordinate system (or in local displacement system, if defined), and strain energy - Torsional moment (MX) about the spring axis

Dynamic Capabilities	<ul style="list-style-type: none"> - Null mass matrix is assumed, see note 1 - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration and shock spectrum analyses
Nonlinear Capabilities	<ul style="list-style-type: none"> - Follows nonlinear force-deflection curve (piecewise linear, spline or polynomial curve fit) - Conservative or nonconservative loading (see *NLSRING data group and Section 2.8.2)

Table 4.46: Available loading (NKTP = 21)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal moments in global X, Y, and Z directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal rotations ROTX, ROTY, and ROTZ in global X, Y, and Z directions (or in local displacement system, if defined), see *SPDISP - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
----------------------	---

Notes:

1. This spring element is a massless element. It may be used in dynamic analyses provided that it is connected to other elements with non-zero masses. Similarly, a null geometric stiffness matrix is assumed in buckling analysis, and the element must be connected to other elements having non-zero geometric stiffness matrices.
2. If the two nodes of the element are not coincident, only one real constant, i.e., the torsional spring constant, is required, otherwise ISYS is set to zero. If the two nodes are coincident, a direction vector (V_i , $i = 1, 2, 3$) is required. The direction vector is described in the coordinate system indicated by ISYS. If ISYS = 1 or 2, it is assumed that the local displacement coordinate system is defined at the referenced node (node 1 or node 2).
3. Material damping in the modal dynamic analysis can be applied to this spring element. For the elements with material damping, a material ID should be specified in eigenvalue analysis to be used later in the modal dynamic analysis. Even though material properties are not required for the spring element, a dummy material property table should be given for the material ID used for the spring element. For any other analysis without material damping, the material ID for the spring element is not required.

Element Library

3-D Torsional Spring Element (NKTP = 21)

4. To use a linear spring in nonlinear analysis, set IDSP in the Real Constant table to zero. In this case, the program uses the spring constant K (which should not be zero) in the Real Constant Table as a constant stiffness in the entire nonlinear analysis.

4.21 2-D Torsional Spring Element (NKTP = 22)

Description

This element is a massless torsional spring in two dimensions. The element must lie in the global XY plane. The element has two rotational degrees of freedom per node (ROTX, ROTY). The two nodes defining the spring may be coincident, in which case a direction vector is specified indicating the direction of the spring axis.

The element configuration is shown in [Figure 4.23](#). An element reference guide is given in [Table 4.47](#) with the available loading given in [Table 4.48](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.47](#) gives the output pertinent to this element.

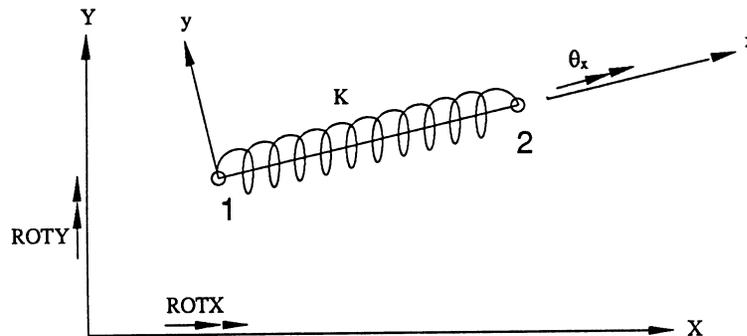


Figure 4.23: Element configuration (NKTP = 22, NORDR = 1)

Table 4.47: Element reference guide (NKTP = 22)

Element Type	NKTP = 22, 2-D Torsional spring element
Analysis Types	Static, Dynamic, Buckling, see note 1
Degrees of Freedom	2 per node: ROTX, ROTY
NORDR (Shape / No. of nodes)	- Line: 2 nodes (NORDR = 1)
Real Constants (RCTABLE)	Up to 5 constants: K, V1, V2, V3, ISYS Where, K Torsional spring constant (moment/radian) V _i V _i = 1, 2, 3; required only if the two nodes are coincident; components of a vector pointing in the spring axis direction (V3 must be zero) ISYS Coordinate system in which the components V _i , i = 1, 2, 3, are given = 0 global cartesian system = 1 local displacement coordinate system at first node, see note 2 = 2 local displacement coordinate system at second node, see note 2
Material Properties	Material ID (optional); see note 3
Element Output	- Internal moments (MX, MY) at the two ends in global coordinate system (or in local displacement system, if defined), and strain energy - Torsional moment (MX) about the spring axis
Dynamic Capabilities	- Null mass matrix is assumed, see note 1 - Eigenvalue, transient dynamic, frequency response, random vibration and shock spectrum analyses

Table 4.48: Available loading (NKTP = 22)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal moments in global X and Y directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal rotations ROTX and ROTY in global X and Y directions (or in local displacement system, if defined), see *SPDISP - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
----------------------	--

Notes:

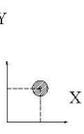
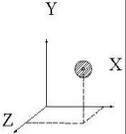
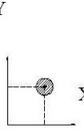
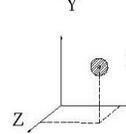
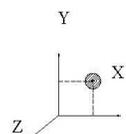
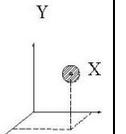
1. This spring element is a massless element. It may be used in dynamic analyses provided that it is connected to other elements with non-zero masses. Similarly, a null geometric stiffness matrix is assumed in buckling analysis, and the element must be connected to other elements having non-zero geometric stiffness matrices.
2. If the two nodes of the element are not coincident, only one real constant, the torsional spring constant, is required. If the two nodes are coincident, a direction vector (V_i , $i = 1, 2, 3$) is required ($V_3 = 0$). The direction vector is described in the coordinate system indicated by ISYS. If ISYS = 1 or 2, it is assumed that the local displacement coordinate system is defined at the referenced node (node 1 or node 2), otherwise ISYS is reset to zero.
3. Material damping in the modal dynamic analysis can be applied to this spring element. For the elements with material damping, a material ID should be specified in eigenvalue analysis to be used later in the modal dynamic analysis. Even though material properties are not required for the spring element, a dummy material property table should be given for the material ID used for the spring element. For any other analysis without material damping, the material ID for the spring element is not required.
4. To use a linear spring in nonlinear analysis, set IDISP in the Real Constant table to zero. In this case, the program uses the spring constant K (which should not be zero) in the Real Constant Table as a constant stiffness in the entire nonlinear analysis.

4.22 Concentrated Mass Elements (NKTP = 25-30)

Description

The concentrated mass elements provide a means to represent the inertia properties of the model and to apply external loads due to body forces (e.g., gravity). The inertia properties may also be represented by the mass density of the structural elements. 2-D and 3-D concentrated mass elements (with or without rotary inertia effects) are available. A reference guide for all of the concentrated mass elements is given in [Table 4.49](#).

Table 4.49: Element reference guide (NKTP = 25 - 30)

Element Type	NKTP = 25 2-D Mass	NKTP = 26 3-D Mass	NKTP = 27 2-D General Mass	NKTP = 28 3-D General Mass	NKTP = 29 2-D General Mass with Rotary Inertia	NKTP = 30 3-D General Mass with Rotary Inertia
Analysis Types	Static, Dynamic, Buckling, Nonlinear Static, Nonlinear Transient (see note 2)					
Configuration (NORDR and No. of nodes = 1 for all Elements)						
Degrees of Freedom	2 UX, UY	3 UX, UY, UZ	2 UX, UY	3 UX, UY, UZ	3 UX, UY, ROTZ	6 UX, UY, UZ ROTX, ROTY, ROTZ
Real Constants (see note 3)	1 (MX)	1 (MX)	2 (MX, MY)	3 (MX, MY, MZ)	3 (MX, MY, IZ)	6 (MX, MY, MZ, IX, IY, IZ)
Dynamic Capabilities	- Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration, and shock spectrum analyses					
Nodal Loading	- Concentrated nodal forces consistent with the number of degrees of freedom of each element, see *CFORCE or *DCFORCE - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER					

Body Force	<ul style="list-style-type: none"> - Gravity loading or body forces due to linear acceleration, see *BODYFORCE - Centrifugal loads due to angular velocity, see *BODYFORCE - Tangential loads due to angular acceleration, see *BODYFORCE
Ground Motion	<ul style="list-style-type: none"> - Forces due to ground motion (support excitation), see *GROUND - Forces due to rotational ground motion about a specified point, see *GROUND

Notes:

1. The 2-D mass elements (NKTP = 25, 27, 29) must lie in the global XY plane.
2. These elements may be connected only to nodes of the structure that possess stiffness (since a null stiffness matrix is assumed). If used in buckling analysis, the mass element should be connected to other elements having non-zero geometric stiffness matrices.
3. These elements have one or more real constants, which must be entered in *RCTABLE data group. They do not require any material properties. The real constants are always in the global coordinate system. The units of mass should be (force. time²/length), and the units of mass moment of inertia should be (force. time². length). The real constants listed in [Table 4.49](#) are defined as follows:

MX : Mass in the X-direction. For NKTP = 25, 26, MX is also the mass in the other direction(s)

MY : Mass in the Y-direction

MZ : Mass in the Z-direction

$\left. \begin{array}{l} IX \\ IY \\ IZ \end{array} \right\}$	Mass moment of inertia about axes located at the mass center of gravity and parallel to the global X, Y, and Z axes, respectively
---	---

The capability to specify different effective mass values in different directions allows the resistance to acceleration to be orientation dependent.

4.23 3-D Laminated Composite General Shell Element (NKTP = 32)

Description

This 3-D shell element includes deformation due to membrane, bending, membrane-bending coupling and transverse shear effects (three node triangular element is an exception, see note 3) and is suited for modeling moderately thick to thin laminated composite shells. The element consists of a number of layers of perfectly bonded orthotropic materials. The element has six degrees of freedom per node (UX, UY, UZ, ROTX, ROTY, ROTZ), but it possesses no rotational stiffness about the normal to the shell surface (note 4). The theoretical basis of the element is discussed in Section 2.8. An element reference guide is briefed in Table 4.50 with the available loading listed in Table 4.51.

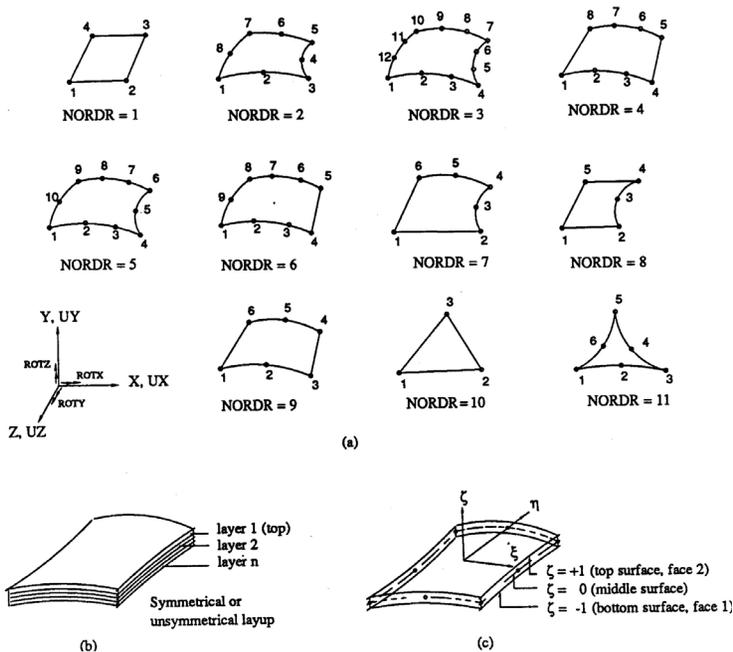


Figure 4.24: 3-D laminated composite shell (NKTP = 32), (a) element configuration, (b) layer setup, (c) face numbering convention for top and bottom surfaces

The element can be shaped as a 4 to 12 node quadrilateral, or a 3 or 6 node triangle depending on the selected NORDR value. The element configuration, layer setup, node

locations and face numbering convention are shown in [Figure 4.24](#). The lamination sequence is between the top and bottom faces of the element with the layer setup starting from the top face. A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.50](#) gives the output pertinent to this element.

Table 4.50: Element reference guide (NKTP = 32)

Element Type	NKTP = 32, 3-D Laminated composite general shell element
Analysis Types	Static, Dynamic, Buckling, Nonlinear Static, Nonlinear Transient
Degrees of Freedom	6 per node: UX, UY, UZ, ROTX, ROTY, ROTZ
NORDR (Shape / No. of nodes)	<ul style="list-style-type: none"> - Quadrilateral: 4 to 12 nodes (NORDR = 1 to 9) - Triangle: 3 or 6 nodes (NORDR = 10, 11) <p>(For nonlinear analysis, only NORDR = 1,2 10 are currently available and NORDR = 10 is NORDR = 1 element by collapsing node3 and node4)</p>
Real Constants (RCTABLE)	<ul style="list-style-type: none"> - 3 to 12 nodal layer thicknesses, (same as number of nodes), entered in *RCTABLE, see notes 5, 6 - 3 to 12 nodal layer rotation angles (same as number of nodes), entered in *LAMANGLE, see notes 5-7
Material Properties	
Orthotropic Elastic	<p>9 properties for each <i>different</i> material (in principal material directions, see note 7):</p> <p>EX, EY, NUXY, GXY, GXZ, GYZ, DENS, ALPX, ALPY</p>
Strength Properties (used in failure criteria)	<p>6 properties for each <i>different</i> material (in principal material directions, see note 8, 14 and 15):</p> <p>FXT, FXC, FYT, FYC, FS, F12</p>

<p>Orthotropic Plastic</p>	<p>Note that strength properties are not required or used in nonlinear analysis. For nonlinear static only, see notes 14, 15 and *APLASTIC data group. Depending on the yield criteria and the stress-strain curve specification, properties in the following material principal directions are required (note that 11, 22, 45 indicate the first, second material principal direction, and a direction inclined 45° to the first material principal direction, respectively. Also, negative sign indicates compressive properties and directions 12, 23 and 13 indicate the in-plane and two transverse shear directions, respectively)</p> <p>Hill's yield criteria: 11, 22, 45, 12, 23, 13</p> <p>Modified Hill's yield criteria: 11, -11, 22, -22, 45, -45, 12, 23, 13</p> <p>The number of required properties in each direction is given in Table 6.6.</p>
<p>Element Output</p>	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ, MX, MY, MZ) and strain energy - Centroidal strains at the midsurface of each layer in material principal directions (EXX, EYY, EXY) - Stresses and failure criteria at the midsurface of each layer, at centroid, Gauss and nodal points in the material principal directions (SXX, SXX/ALLOW, SYX, SYX/ALLOW, SXY, SXY/ALLOW, TSAI-WU, HILL-MISES, TSAI-HILL (or user-defined criterion)), see note 12 - For linear analysis, inter-laminar shear stresses (SXZ and SYZ) with respect to material principal axes of each layer and/or with respect to the material principal axes of the top-layer (see *SFDCOMP data group), see note 17. - Stresses resultant at centroid, Gauss and nodal points in the material principal axes associated with the top layer (NXX, NYY, NXY, MXX, MYY, MXY, QX, QY), see note 10 - For nonlinear analysis, stresses at midsurface of each layer, at centroid, Gauss and nodal points, in the material principal directions (SXX, SYX, SXY, SYZ, SXZ). - Equivalent plastic strain (EPS), effective stress (EFS), and yield radius (YLD), for nonlinear static only, at Gauss and nodal points, see note 13. - Second Piola-Kirchhoff or Cauchy stresses, for nonlinear static only at Gauss and nodal points.

Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration and shock spectrum analyses
Nonlinear Capabilities	<ul style="list-style-type: none"> - Geometric nonlinearity: Large displacements and rotations; total Lagrangian formulation; and deformation dependent loads. - Material nonlinearity: Elastoplastic material models with various yield criteria.

Table 4.51: Available loading (NKTP = 32)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X, Y and Z directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal displacements UX, UY, UZ and nodal rotations ROTX, ROTY, ROTZ in global directions (or in local displacement system, if defined), see *SPDISP - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER - Concentrated nodal follower (deformation dependent) forces perpendicular to the middle surface of the element, for nonlinear static only, see *FOLLOWER
Pressure Loading	<ul style="list-style-type: none"> - Uniform or non-uniform pressure (force/area) on top or bottom faces of the element, see *PRESSURE or *DPRESSURE and note 2 - Uniform or non-uniform follower (deformation dependent) pressure (force/area) on top and/or bottom faces of the element, for nonlinear static only, see *PRESSURE
Thermal Loading	<ul style="list-style-type: none"> - Specified nodal temperature and nodal temperature difference in element thickness direction, see *NDTEMPER and *NDTEMPDIF
Body Force	<ul style="list-style-type: none"> - Gravity loading or body forces due to linear acceleration in global X, Y and Z directions, see *BODYFORCE - Centrifugal loads due to angular velocity about the global X, Y and Z direction, see *BODYFORCE - Tangential loads due to angular acceleration about the global X, Y and Z directions, see *BODYFORCE

Element Library

3-D Laminated Composite General Shell Element (NKTP = 32)

Ground Motion	<ul style="list-style-type: none">- Forces due to ground motion (support excitation) in the global X, Y and Z directions, see *GROUND- Forces due to rotational ground motion in the global XY, YZ and ZX planes about a specified point, see *GROUND
----------------------	--

Notes:

1. The element may be oriented anywhere in space. The element connectivity must be given in the order shown in [Figure 4.24](#), in which the node numbering sequence starts at a corner node and proceeds along the perimeter of the element.
2. The bottom and top faces of the element are numbered as faces 1 and 2, respectively ([Figure 4.24 \(c\)](#)). Top and bottom faces are determined as follows:
 - (a) Choose a viewpoint so that the element is defined counterclockwise
 - (b) The top face is now closest to you

For pressure loading, the pressure (force/area) is integrated over the area of the loaded face. For curved shells, this area will be somewhat different than the area of the shell midsurface.

3. Transverse shear effects are included in the element formulation.
4. This element possesses no rotational stiffness about the normal to the shell surface. To avoid any possible difficulties, set AUTO = ON in the executive commands. A small rotational stiffness is then added at the nodes where the shell surface has continuous curvature.
5. For most NISA elements, the variable MATID in the *ELEMENTS data group refers to a material property table defined in *MATERIAL data group. However, this is inadequate for composite shells, since each layer may be of a different material. Therefore, for this element, MATID is a pointer to an entry in the *LAMSQ2 (lamination sequence) data group. The *LAMSQ2 data group in turn contains pointers to the layer thicknesses, rotation angles, and materials for all layers (*RCTABLE, *LAMANGLE and *MATERIAL data groups, respectively). In this way, we retain the ability to model composite shells which have tapered thickness or variable rotation angles. There is no limitation on the number of layers or the leap sequence. Note that the number of real constant tables, lamination angle tables, or material property tables may not necessarily be equal to the number of layers, since more than one layer may be of the same material type, thickness type, or lamination angle (orientation) type.

6. There is no need to enter a non-zero value for the variable IDRC (real constant ID number) in the *ELEMENTS data group for this element. Any value entered for IDRC will be reset to zero.
7. The element consists of a number of layers of perfectly bonded orthotropic materials. For each different material, all of the mechanical properties (EX, EY, NUXY, GXY, GYZ, GXZ, ALPX, ALPY and DENS) should be defined, otherwise a default value of zero is assumed. These properties are defined in the material principal directions, such that x is the first material axis (fiber direction), y is the second material axis and is normal to x, and z is normal to the shell midsurface. The xy plane of the material axes system is tangent to the shell midsurface. Several options are available for the definition of the material principal axes, see *LAMSQ2 for details.
8. For linear analysis, the strength properties (entered in *MATERIAL data group) are used to evaluate layer failure based on maximum stress, Hill-Mises, Tsai-Wu, and Tsai-Hill (Tsai-Hill criterion can be replaced by a user-defined criterion) criteria. The properties are defined as:

FXT, FXC : x-direction tensile and compressive failure stress, respectively

FYT, FYC : y-direction tensile and compressive failure stress, respectively

FS : inplane (xy) shear failure stress

F12 : coupling coefficient between x and y directions for Tsai-Wu theory

Compressive failure stresses are also input as positive values.

If a typical strength property is not specified, a default value in proportion to the graphite/epoxy strength properties [4.1] is assumed as follows,

$$FXC = \frac{10 \times 10^4}{30 \times 10^6} EX, \quad FXT = \frac{15 \times 10^4}{30 \times 10^6} EX$$

$$FYC = \frac{17 \times 10^3}{7.5 \times 10^5} EY, \quad FYT = \frac{6 \times 10^3}{7.5 \times 10^5} EY$$

$$FS = \frac{10^4}{3.75 \times 10^5} GXY$$

The default value for F12 is,

$$F12 = \frac{1}{FXAVG * FYAVG}$$

where,

$$FXAVG = \frac{1}{2}(FXC + FXT), \quad FYAVG = \frac{1}{2}(FYC + FYT)$$

9. For linear analysis, stresses are computed in the material principal axes at the midsurface of each layer. These stresses may be filtered before printout. The ratio of actual stress to allowable stress will be compared to the threshold value entered on the *SFDCOMP data group. The printout will be suppressed if all the stress ratio, Tsai-Wu failure index and Hill-Mises failure index are less than the threshold value.
10. Stress resultants will be calculated in the coordinate system associated with layer number 1 (the top layer). The three force resultants (NXX, NYY, NXY) are the integrals through the shell thickness of the inplane stresses (SXX, SYX, SXY, respectively). Similarly, the moment resultants (MXX, MYY, MXY) are the integrals of the inplane stresses, weighted by their distance from the middle surface. Finally, the transverse shear stress resultants (QX, QY) are the integrals through the thickness of the transverse (interlaminar) shear stresses (SXZ, SYZ, respectively). To obtain a stress resultant printout in a preferred coordinate system, define an extremely small thickness for layer number 1. The stress printout will be suppressed for any layer having all nodal thicknesses (*RCTABLE data group) less than 1.E-9. Thus, the rotation angles (*LAMANGLE data group) associated with such a layer can be input to control the orientation for stress resultant calculations.

Another use of this feature is to obtain a stress printout at the top or bottom of a layer, instead of the midsurface which is the default location. To do this, define thin layers (but greater than 1.E-9) on the top or bottom of the actual layer. Stresses at the midsurface of these thin layers will be essentially at the top and bottom of the actual layer. (Note that the formulation of this element is such that there is no significant computational penalty for increasing the number of layers.)

11. The following special plots are available for this element. These plots may be obtained using the postprocessing module of the DISPLAY program:
 - (a) Stress survey plots: The most critical stress ratio, see note 9, found in each element will be printed, as a percentage, on a plot of the undeformed geometry. This plot is available only for linear analysis.
 - (b) Layer stress contour plots: The three inplane stresses (SXX, SYX, SXY) in material coordinates can be contoured on any layer of the shell. For nonlinear analysis SXZ and SYZ plots as well as the effective stress and the effective plastic strain plots are also available.

- (c) Stress resultant contour plots, see note 10 for the definition of NXX, NYY, NXY, MXX, MYY, MXY, QX and QY.

The following contours are available on an element-by-element basis using the DISPLAY-POST program:

- (a) Unaveraged nodal layer stresses, failure criteria and layer strains.
- (b) Maximum nodal values over all the layers of an individual stress component (e.g., SXX). The layer number and component number are indicated on the plot.
- (c) Maximum nodal value over all the layers and all the stress components considered together. The layer number and component number are indicated on the plot.
- (d) Stress resultants.
- (e) Interlaminar shear stresses (see note 17).

12. In linear analysis, the following failure theories are implemented for this element.

- (a) In the *maximum stress failure theory*, each of the stress components is compared independently to a unidirectional allowable (failure) stress. The allowable stresses for SXX and SYY may be different in tension and compression, and all allowable stresses may be temperature dependent. The ratio of actual to allowable is computed at every point where stresses are computed. *If the ratio exceeds 1.0, layer failure is presumed to have occurred.* This theory neglects all interactions among the stress components.
- (b) *The modified Hill-Mises failure theory* [4.5] accounts for interaction of the inplane stress components by defining a failure surface with the functional form,

$$F(SXX, SYY, SXY) = [(SXX/FX)**2 - (SXX/FX)*(SYY/FX) + (SYY/FY)**2 + (SXY/FS)**2]**(1/2)$$

When the value of this function reaches unity, failure of the layer under combined loading is predicted. FX, FY and FS are the failure stresses under unidirectional loading in the axial directions and shear, respectively. Note that this theory does not account for the difference between tensile and compressive failure stresses. Before applying this theory in NISA, the user-input tensile and compressive failure stresses are averaged.

(c) *The Tsai-Wu failure theory* [4.2] defines the following failure surface,

$$F = F1*SXX + F2*SYY + F11*SXX**2 + F22*SYY**2 + F33*SXY**2 + F12*SXX*SYY$$

Except for the linear terms, which allow the difference between tensile and compressive failure stress to be modeled, this looks much the same as the Hill-Mises equation. However, the definitions of the coefficients F1, F2,... F12 are more general. We have,

$$F1 = 1/FXT - 1/FXC, \quad F2 = 1/FYT - 1/FYC$$

$$F11 = 1/(FXT*FXC), \quad F22 = 1/(FYT*FYC)$$

$$F33 = 1/FS**2$$

The above coefficients are computed from the user-input engineering failure stresses. The calculation of F12 is left to the user since it depends upon the experimental method chosen to perform a biaxial failure test (see [4.3]). Hence, the value of F12 is input directly as a material property in the *MATERIAL data group. In the above expression F can be formulated using failure strength ratio, R, in the following manner,

$$(F11*SXX**2 + F22*SYY**2 + F33*SXY**2 + F12*SXX*SYY)R^2 + (F1*SXX + F2*SYY)R - 1 = 0$$

The Tsai-Hill theory or user defined failure theory (See Appendix D.4): If a user-defined failure theory is used, the Tsai-Hill theory will be replaced by the user-defined failure theory. The Tsai-Hill theory is defined as follows,

$$F = (SXX/FX)**2 - (SXX/FX)*(SYY/FX) + (SYY/FY)**2 + (SXY/FS)**2$$

FX, FY and FS are the failure stresses under unidirectional loading in the axial and shear directions. This theory does not account for the difference between tensile and compressive failure stresses. NISA applies the average value of the user-input tensile and compressive failure stresses in the formula.

13. For nonlinear static analysis, the nodal equivalent plastic strains (EPS) are approximated by extrapolating the Gauss point equivalent plastic strains.
14. For material nonlinearity, the user may choose from two different yield criteria. These are the Hill's anisotropic yield function with identical yield strengths in tension and compression [4.5] or a modified function that takes into account differential yield strengths in tension and compression [4.9]. The strength properties specified here for

the above yield criteria correspond to the strength properties specified for failure criteria in linear analysis (see *APLASTIC). The coupling term F12 is not required in nonlinear analysis, since it will be determined internally in the program by satisfying the plastic incompressibility constraint. However, the input for the yield surface requires an extra strength property i.e. strength along 45° axis to the fibre direction. In the absence of this input the program internally computes an approximate value for H according to Ref. [4.10].

15. It should be noted that the modified Hill-Mises failure theory is based on Hill's anisotropic yield function [4.5] with the assumptions of plane stress and equal transverse material strengths in two directions, e.g., $F_Y = F_Z$. The yield surface specified in *PLASTIC does not imply, however, the equal transverse strengths when specialized to plane stress. Therefore, the input for the yield surface requires an extra strength property, e.g. strength at 45° from one of the principal material directions, to determine all the *four* constants needed for the yield surface specification.
16. Only four-node linear element (NORDR = 1) and eight-node quadratic element (NORDR = 2) are available for nonlinear static analysis.
17. Interlaminar shear stress computation is available for NORDR = 2, 3, 5, 6, 7, and 11 only (see *SFDCOMP data group).
18. For nonlinear analysis, layer-wise mid-point integration is performed assuming that the stresses are constant within each layer. Therefore, for problems involving bending, enough number of layers (say 6 to 8) should be used.

4.24 3-D Laminated Sandwich General Shell Element (NKTP = 33)

Description

This 3-D shell element includes deformation due to membrane, bending, membrane-bending coupling and transverse shear effects and is suited for modeling moderately thick to thin sandwich shells. The element consists of two or more face sheets, which are assumed to be thin and stiff, and one or more cores, which are relatively thick and flexible. The face sheets are assumed to be in a state of plane stress (i.e., they sustain the inplane stresses SXX, SYY, SXY), and the core material sustains only transverse shear stresses SXZ and SYZ. The face sheets themselves may be isotropic, or they may consist of a number of layers of composite materials. For generality, the data input is in the form of lamination sequences (the same as for NKTP = 32 element). The core is identified by a material property input of EX explicitly set to zero. The lamination sequence is between the top and bottom faces of the element with the layer setup starting from the top face. The element has six degrees of freedom per node (UX, UY, UZ, ROTX, ROTY, ROTZ), but it does not possess rotational stiffness about the normal to the shell surface (note 3). The theoretical basis of the element is discussed in [Section 2.8](#).

The element can be shaped as a 4 to 12 node quadrilateral depending on the selected NORDR value. The element configuration, layer setup, node locations and face numbering convention are shown in [Figure 4.25](#). An element reference guide is briefed in [Table 4.52](#) with the available loading listed in [Table 4.53](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.2](#) respectively. [Table 4.52](#) gives the output pertinent to this element

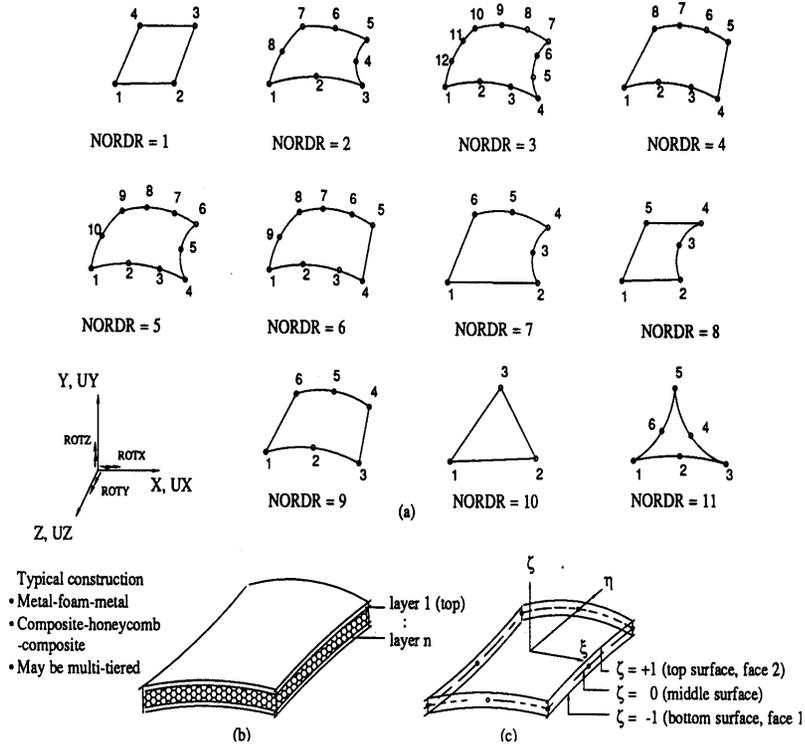


Figure 4.25: 3-D laminated sandwich shell (NKTP = 33), (a) element configuration, (b) layer setup and typical construction, (c) face numbering convention for top and bottom surfaces

Table 4.52: Element reference guide (NKTP = 33).

Element Type	NKTP = 33, 3-D Laminated sandwich general shell element
Analysis Types	Static, Dynamic, Buckling
Degrees of Freedom	6 per node: UX, UY, UZ, ROTX, ROTY, ROTZ
NORDR (Shape / No. of nodes)	- Quadrilateral: 4 to 12 nodes (NORDR = 1 to 9) - Triangular: 3 or 6 nodes (NORDR = 10, 11)
Real Constants (RCTABLE)	4 to 12 nodal layer thicknesses (same as number of nodes), entered in *RCTABLE, see notes 4-6 4 to 12 nodal layer rotation angles (same as number of nodes), entered in *LAMANGLE, see notes 4-6
Material Properties Face sheet Material Core Material	9 properties for each different material (in principal material directions, see note 6): EX, EY, NUXY, GXY, GXZ, GYZ, DENS, ALPX, ALPY 5 strength properties for each different material, see notes 6, 7: FXT, FXC, FYT, FYC, FS 4 properties for each different core material (in principal material directions, see note 6): EX (must be set to 0.0), GXZ, GYZ, DENS 1 strength property for each different core material, see notes 6, 7: FTS
Element Output	- Internal forces (FX, FY, FZ, MX, MY, MZ) and strain energy - Centroidal strains for each layer in material principal directions (for face sheets: EXX, EYY, EXY; for core layers: EXX, DYY, EXY, EYZ, EXZ) - Stresses and stress ratios at the midsurface of each layer, at centroid, Gauss and nodal points in the material principal directions (for face sheets: SXX, SXX/ALLOW, SYX, SYX/ALLOW; for core layers: SXZ, SXZ/ALLOW, SYZ, SYZ/ALLOW TSAI-WU, HILL-MISES, TSAI-HILL (or user defined criterion), See note 12. - Stress resultants at centroid, Gauss and nodal points in the material principal axes associated with the top layer (NXX, NYY, NXY, MXX, MYY, MXY, QX, QY), see note 9

Dynamic Capabilities	<ul style="list-style-type: none"> - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration and shock spectrum analyses - Eigenvalue, transient dynamic, frequency response, random vibration and shock spectrum analyses
-----------------------------	---

Table 4.53: Available loading (NKTP = 33)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X, Y and Z directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal displacements UX, UY, UZ and nodal rotations ROTX, ROTY, ROTZ in global directions (or in local displacements system, if defined), see *SPDISP - Concentrated unit harmonic loading for point-to- point transfer function, see *DRIVER
Pressure Loading	<ul style="list-style-type: none"> - Uniform or non-uniform pressure (force/area) on top or bottom faces of the element, see *PRESSURE or *DPRESSURE and note 2
Thermal Loading	<ul style="list-style-type: none"> - Specified nodal temperature and nodal temperature difference in element thickness direction, see *NDTEMPER and *NDTEMPDIF
Body Force	<ul style="list-style-type: none"> - Gravity loading or body forces due to linear acceleration in global X, Y and Z directions, see *BODYFORCE - Centrifugal loads due to angular velocity about the global X, Y and Z direction, see *BODYFORCE - Tangential loads due to angular acceleration about the global X, Y and Z directions, see *BODYFORCE
Ground Motion	<ul style="list-style-type: none"> - Forces due to ground motion (support excitation) in the global X, Y and Z directions, see *GROUND - Forces due to rotational ground motion in the global XY, YZ and ZX planes about a specified point, see *GROUND

Notes:

1. The element may be oriented anywhere in space. The element connectivity must be given in the order as shown in [Figure 4.25](#), in which the node numbering sequence starts at a corner node and proceeds along the perimeter of the element.

2. The bottom and top faces of the element are numbered as faces 1 and 2 respectively [Figure 4.25 \(c\)](#). Top and bottom faces are determined as follows:
 - (a) Choose a viewpoint so that the element is defined counterclockwise
 - (b) The top face is now closest to you

For pressure loading the pressure (force/area) is integrated over the area of the loaded surface. For curved shells, this area will be somewhat different than the area of the shell midsurface.

3. This element possesses no rotational stiffness about the normal to the shell surface. To avoid any possible difficulties, set `AUTO = ON` in the executive commands. A small rotational stiffness is then added at the nodes where the shell surface has continuous curvature.
4. For most NISA elements, the variable `MATID` in the `*ELEMENTS` data group refers to a material property table defined in `*MATERIAL` data group. However, this is inadequate for sandwich shells, since each layer may be of a different material, and indeed we insist on the core material being different from the face sheets. Therefore, for this element, `MATID` is a pointer to an entry in the `*LAMSQ2` (lamination sequence) data group. The `*LAMSQ2` data group in turn contains pointers to the layer thicknesses, rotation angles, and materials for all layers (`*RCTABLE`, `*LAMANGLE` and `*MATERIAL` data groups, respectively). In this way, we retain the ability to model composite shells which have tapered thickness or variable rotation angles. There is no limitation on the number of layers or the layup sequence. There is no restriction on the number of face sheets or core; however the assumption is made of a common transverse shear angle, for all cores. Note that the number of real constant tables, lamination angle tables, or material property tables may not necessarily be equal to the number of layers, since more than one layer may be of the same material type, thickness type, or lamination angle (orientation) type.
5. There is no need to enter a non-zero value for the variable `IDRC` (real constant ID number) in the `*ELEMENTS` data group for this element. Any value entered for `IDRC` will be reset to zero.
6. For the face sheets, the input of `GXZ`, `GYZ` is optional. The core material is identified by a zero value for `EX`. Note that the material properties are defined in the material principal directions, such that `x` is the first material axis, `y` is the second material axis and is normal to `x`, and `z` is normal to the shell midsurface. The material `xy` plane is tangent to the shell midsurface. Several options are available for the definition of the material principal axes, see `*LAMSQ2` for details.

7. The strength properties (entered in *MATERIAL data group) are used to evaluate the maximum stress failure criterion for each layer of the sandwich shells. For the face sheets, the properties are:

FXT, FXC : x-direction tensile and compressive failure stress, respectively

FYT, FYC : y-direction tensile and compressive failure stress, respectively

FS : inplane (xy) shear failure stress

For the core layers, the strength property is:

FSXZ : transverse shear failure stress along X-Z plane

FSYZ : transverse shear failure stress along Y-Z plane

8. Three inplane components of stress (SXX, SYY, SXY) are computed in the material principal axes at the midsurface of each layer of the face sheets. Two transverse shear stresses (SXZ, SYZ) are computed in the material axes of the core material. The maximum stress failure criterion is implemented in this element, where each of the stress components is compared independently to a unidirectional allowable (failure) stress. The allowable stress for SXX, SYY may be different in tension and compression, and all allowable stresses may be temperature dependent. The ratio of actual stress to allowable is computed at every point where stresses are computed. If the ratio exceeds 1.0, layer failure is presumed to have occurred. The layer stress output may be filtered before printout. The stress ratios will be compared to the threshold value entered on the *SFDCOMP data group. The printout will be suppressed if all the stress ratio are less than the threshold value.
9. Stress resultants will be calculated in the coordinate system associated with layer number 1 (the top layer). The three force resultants (NXX, NYY, NXY) are the integrals through the shell thickness of the inplane stresses (SXX, SYY, SXY, respectively). Similarly, the moment resultants (MXX, MYY, MXY) are the integrals of the inplane stresses weighted by their distance from the middle surface. Finally, the transverse shear stress resultants (QX, QY) are the integrals through the thickness of the transverse (interlaminar) shear stresses (SXZ, SYZ, respectively). To obtain a stress resultant printout in a preferred coordinate system, define an extremely small thickness for layer number 1. The stress printout will be suppressed for any layer having all nodal thicknesses (*RCTABLE data group) less than 1.E-9. Thus, the rotation angles (*LAMANGLE data group) associated with such a layer can be input to control the orientation for stress resultant calculations.

Another use of this feature is to obtain stress printout at the top or bottom of a layer, instead of the midsurface which is the default location. To do this, define thin layers (but greater than 1.E-9) on the top or bottom of the actual layer. Stresses at the midsurface of these thin layers will be essentially at the top and bottom of the actual layer. (Note that the formulation of this element is such that there is no significant computational penalty for increasing the number of layers.)

10. The following special plots are available for this element. These plots may be obtained using the postprocessing module of the DISPLAY program:
 - (a) Stress survey plots: The most critical stress ratio found in each element (see note 8) will be printed on a plot of the undeformed geometry.
 - (b) Layer stress contour plots: The three inplane stresses (SXX, SYY, SXY) in material coordinates can be contoured on any layer of the face sheets.
 - (c) Stress resultant contour plots, see note 9 for the definition of NXX, NYY, NXY, MXX, MYY, MXY, QX and QY.

The following contours are available on an element-by-element basis using the DISPLAY-POST program:

- (a) Unaveraged nodal layer stresses, failure criteria and layer strains.
 - (b) Maximum nodal values over all the layers of an individual stress component (e.g., SXX). The layer number and component number are indicated on the plot.
 - (c) Maximum nodal value over all the layers and all the stress components considered together. The layer number and component number are indicated on the plot.
 - (d) Stress resultants.
11. Interlaminar shear stress computation is available for NORDR = 2, 3, 5, 6, 7, and 11 (see *SFDCOMP data group, [Section 7.5.6](#)).
 12. In linear analysis, the following failure theories are implemented for this element.
 - (a) In *the maximum stress failure theory*, each of the stress components is compared independently to a unidirectional allowable (failure) stress. The allowable stresses for SXX and SYY may be different in tension and compression, and all allowable stresses may

be temperature dependent. The ratio of actual to allowable is computed at every point where stresses are computed. If the ratio exceeds 1.0, layer failure is presumed to have occurred. This theory neglects all interactions among the stress components.

- (b) *The modified Hill-Mises failure theory* [4.5] accounts for interaction of the inplane stress components by defining a failure surface with the functional form,

$$F(SXX, SYY, SXY) = [(SXX/FX)**2 - (SXX/FX)*(SYY/FX) + (SYY/FY)**2 + (SXY/FS)**2]**(1/2)$$

When the value of this function reaches unity, failure of the layer under combined loading is predicted. FX, FY and FS are the failure stresses under unidirectional loading in the axial directions and shear, respectively. Note that this theory does not account for the difference between tensile and compressive failure stresses. Before applying this theory in NISA, the user-input tensile and compressive failure stresses are averaged.

- (c) *The Tsai-Wu failure theory* [4.2] defines the following failure surface,

$$F = F1*SXX + F2*SYY + F11*SXX**2 + F22*SYY**2 + F33*SXY**2 + F12*SXX*SYY$$

Except for the linear terms, which allow the difference between tensile and compressive failure stress to be modeled, this looks much the same as the Hill-Mises equation. However, the definitions of the coefficients F1, F2,... F12 are more general. We have,

$$F1 = 1/FXT - 1/FXC, \quad F2 = 1/FYT - 1/FYC$$

$$F11 = 1/(FXT*FXC), \quad F22 = 1/(FYT*FYC)$$

$$F33 = 1/FS**2$$

The above coefficients are computed from the user-input engineering failure stresses. The calculation of F12 is left to the user since it depends upon the experimental method chosen to perform a biaxial failure test (see [4.3]). Hence, the value of F12 is input directly as a material property in the *MATERIAL data group. The above expression F can be formulated using failure strength ratio, R, in the following manner,

$$(F11*SXX**2 + F22*SYY**2 + F33*SXY**2 + F12*SXX*SYY)R^2 + (F1*SXX + F2*SYY)R - 1 = 0$$

- (d) The Tsai-Hill theory or user defined failure theory (See Appendix D.4): If a user-defined failure theory will be replaced by the user-defined failure theory. The Tsai-Hill theory is defined as follows,

$$F = (SXX/FX)**2 - (SXX/FX)*(SYY/FX) + 0 \\ (SYY/FY)**2 + (SXY/FS)**2$$

FX, FY and FS are the failure stresses under unidirectional loading in the axial and shear directions. This theory does not account for the difference between tensile and compressive failure stresses. NISA applies the average value of the user-input tensile and compressive failure stresses in the formula.

4.25 Axisymmetric Solid with Non-Axisymmetric Loading (NKTP = 34)

4.25.1 Description

This element is used for 2-D modeling of a solid of revolution which is axially symmetric in its geometry and material but not in its loading. The Fourier series method is used to express the non-axisymmetric loading (in the circumferential coordinate) as the sum of several component loadings (harmonics). Since the Fourier harmonics are not coupled, the original 3-D problem is transformed to a series of 2-D problems that do not interact [4.6].

The solution for each harmonic component is obtained separately using the same two-dimensional mesh (a cross-section passing through the axis of revolution). The solution to the original problem is obtained by adding the solutions of the harmonic components. The procedure is automated such that the response (displacements, stresses, etc.) may be provided at any specified angle θ .

The element must lie in the right half of the global Cartesian XY plane (this is also the plane of the circumferential coordinate $\theta = 0$ implied in the model) as shown in [Figure 4.26](#). The global Y axis is the axis of revolution. The global X and Z axes are the radial and tangential directions for the $\theta = 0$ plane, respectively. At any angle, the xyz system is used to express the components of motion and forces, such that x is the radial direction, y (parallel to global Y axis) is the axial direction, and z is the tangential direction. For the $\theta = 0$ plane, the xyz system is parallel to the global Cartesian XYZ system. A positive angle is measured from the global X axis rotating about the negative Y axis. The output for displacements, internal forces and reactions for any angular section is provided in the xyz system (radial, axial, tangential), see note 4.

The element has three degrees of freedom per node: UX (radial, along x), UY (axial, along y), UZ (tangential, along z). The third degree of freedom UZ is present to account for 3-D behavior under the action of non-axisymmetric loading. The state of stress is characterized by six components (Sxx, Syy, Szz, Sxy, Syz, Sxz) referring to the xyz system at any angular section.

The element can be shaped as a 4 to 12 node quadrilateral, or a 3 or 6 node triangle depending on the selected NORDR value. The element configuration, node locations and

Element Library

Axisymmetric Solid with Non-Axisymmetric Loading (NKTP = 34)

face numbering convention are shown in [Figure 4.27](#). An element reference guide is given in [Table 4.54](#) with the available loading given in [Table 4.55](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.54](#) gives the output pertinent to this element.

The only non-axisymmetric loads that may be applied to this element are concentrated forces (ring loads) and pressure loads. Body forces and thermal loading must be axisymmetric.

[Section 4.25.2](#) has some background information regarding the Fourier series representation of non-axisymmetric loads and their classification to symmetric and antisymmetric components about the $\theta = 0$ plane.

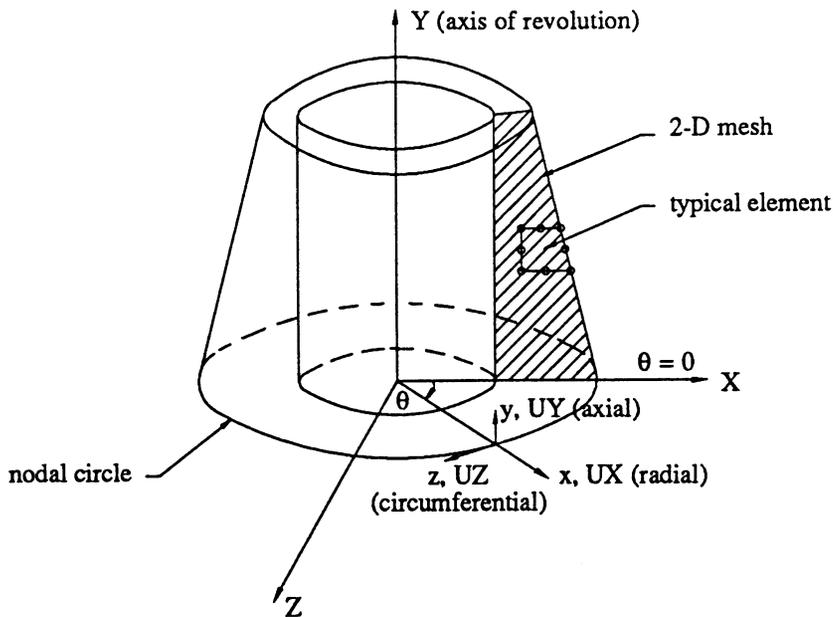


Figure 4.26: Coordinate system for NKTP = 34

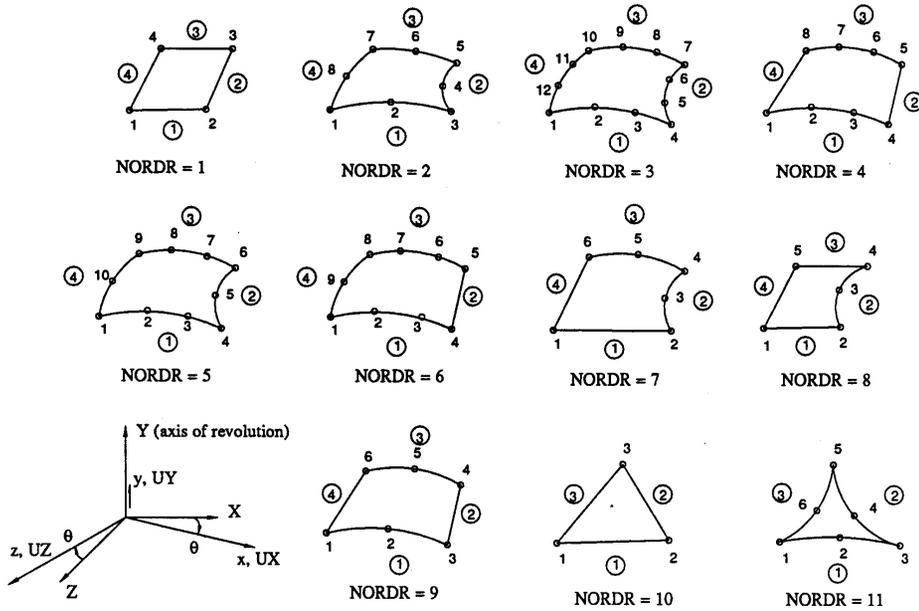


Figure 4.27: Element configuration for available NORDR values, NKTP = 34 (circled numbers indicate face numbers)

Table 4.54: Element reference guide (NKTP = 34)

Element Type	NKTP = 34, Axisymmetric solid with non-axisymmetric loading
Analysis Types	Static
Degrees of Freedom	3 per node: UX (radial), UY (axial), UZ (tangential), see note 4*
NORDR (Shape / No. of nodes)	- Quadrilateral: 4 to 12 nodes (NORDR = 1 to 9) - Triangular: 3 or 6 nodes (NORDR = 10, 11)
Real Constants	None
Material Properties	
Isotropic Elastic	4 properties: EX, NUXY, DENS, ALPX

Element Library

Axisymmetric Solid with Non-Axisymmetric Loading (NKTP = 34)

Element Output	<ul style="list-style-type: none">- Displacement solution at the specified angular sections- Internal forces and reaction forces (both per unit radian) at the specified angular sections- Element Gauss and/or nodal stresses (Sxx, Syy, Szz, Sxy, Syz, Sxz) in the xyz system at the specified angular sections- Element principal stresses (S1, S2, S3), von Mises equivalent stress, octahedral and maximum shear stresses at the specified angular sections- Averaged nodal stresses at the specified angular sections
-----------------------	---

* notes are given after [Section 4.25.1](#)

Table 4.55: Available loading (NKTP = 34)

Nodal Loading	<ul style="list-style-type: none">- Concentrated nodal forces (ring loads) in the xyz system; F_x = radial; F_y = axial; F_z = tangential (or in local displacement system, if defined). The ring load varies harmonically in the circumferential direction θ. See notes 4, 6*- Specified non-zero nodal displacements for a nodal circle (assumed to be constant in the circumferential direction). See *SPDISP and note 3
Pressure Loading	<ul style="list-style-type: none">- Uniform or non-uniform pressure (force/area) on any face of the element. The pressure load varies harmonically in the circumferential direction θ. See note 5
Thermal Loading (axisymmetric)	<ul style="list-style-type: none">- Specified nodal temperature, see *NDTEMPER. The thermal loads are assumed to be axisymmetric
Body Force (axisymmetric)	<ul style="list-style-type: none">- Gravity loading or body forces due to linear acceleration in the x (radial) and Y directions, see *BODYFORCE- Centrifugal loads due to angular velocity about the axis of revolution (global Y axis) see *BODYFORCE

* notes are given after [Section 4.25.2](#)

4.25.2 Fourier Series Representation for Non-Axisymmetric Loads

The Fourier series expansion for a non-axisymmetric load F is given by,

$$F(\theta) = a_0 + \sum_{n=1}^N a_n \cos n\theta + \sum_{n=1}^N b_n \sin n\theta \quad (4.1)$$

where,

$$a_0 = \frac{1}{2\pi} \int_{-\pi}^{\pi} F(\theta) d\theta$$

$$a_n = \frac{1}{\pi} \int_{-\pi}^{\pi} F(\theta) \cos n\theta d\theta \quad (4.2)$$

$$b_n = \frac{1}{\pi} \int_{-\pi}^{\pi} F(\theta) \sin n\theta d\theta$$

where, θ is the circumferential coordinate implied in the model. The number of harmonic terms in the series (N in [Equation 4.1](#)) required for acceptable representation of the loading depends on the complexity of the loading, and should be selected by observing the plot of the function with increasing number of terms.

Cosine terms in [Equation 4.1](#) are called symmetric or even, since $\cos\theta = \cos(-\theta)$. Sine terms are called antisymmetric or odd, since $\sin\theta = -\sin(-\theta)$. This terminology holds for a load applied in the x (radial) or the y (axial) directions, such that: the constant term represents an axisymmetric loading, the cosine terms represent symmetric loading about the $\theta = 0$ plane and the sine terms represent symmetric loading about the $\theta = 0$ plane. The opposite is true for a load applied in the z (tangential) direction, since to achieve symmetry the direction of Fz has to change for $\theta > \pi$ (see [Figure 4.26](#)). Therefore, for the tangential load (Fz) the terms in [Equation 4.1](#) have the following meaning: the constant term represent antisymmetric constant loading about the $\theta = 0$ plane (pure torsion), the sine terms represent symmetric loading about the $\theta = 0$ plane and the *cosine* terms represent antisymmetric loading about the $\theta = 0$ plane. The classification of symmetric and antisymmetric components of the

Element Library

Axisymmetric Solid with Non-Axisymmetric Loading (NKTP = 34)

loading is depicted in [Figure 4.28](#). The Fourier series expansion for non-axisymmetric pressure loading is similar to what is shown for the radial and axial loads in the figure.

Decomposing the non-axisymmetric nodal loads (ring loads acting on nodal circles) and the pressure loading into symmetric and antisymmetric component loadings (about the $\theta = 0$ plane), we have [4.6],

$$F_x = \sum_{n=0}^N \bar{F}_{xn} \cos n\theta + \sum_{n=0}^N \bar{\bar{F}}_{xn} \sin n\theta$$
$$F_y = \sum_{n=0}^N \bar{F}_{yn} \cos n\theta + \sum_{n=0}^N \bar{\bar{F}}_{yn} \sin n\theta$$

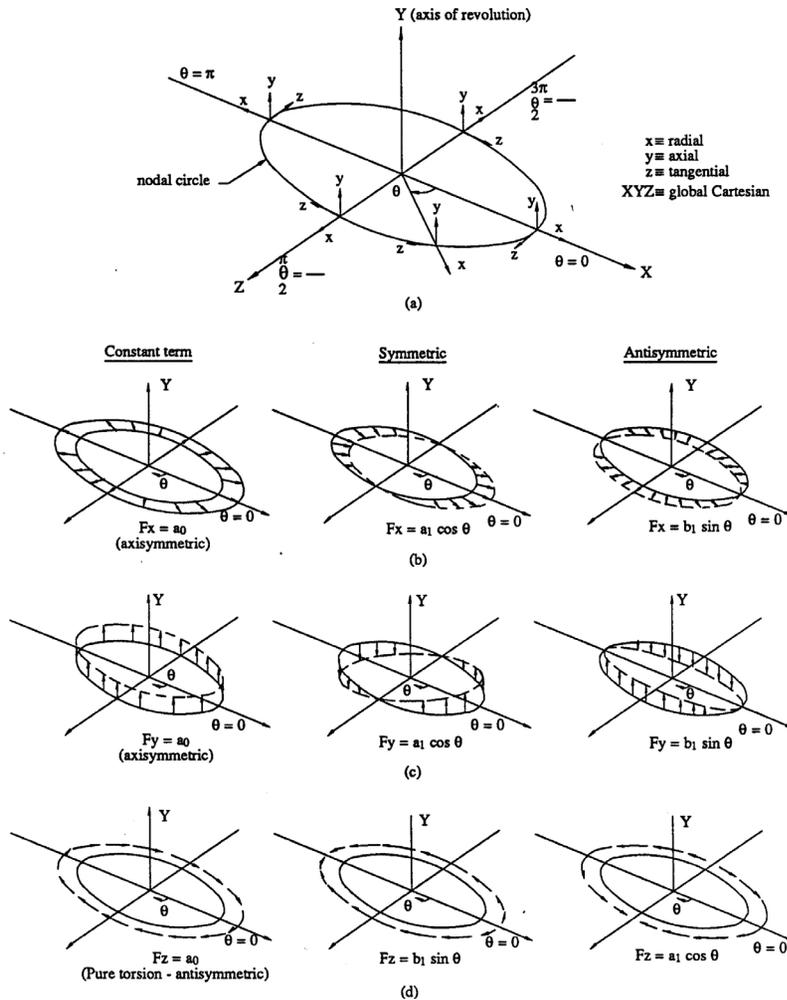


Figure 4.28: Examples of symmetric and antisymmetric loading, (a) xyz coordinate system for different angles, (b) radial loading, (c) axial loading, and (d) Tangential loading

Element Library

Axisymmetric Solid with Non-Axisymmetric Loading (NKTP = 34)

$$F_z = \sum_{n=0}^N \bar{F}_{zn} \sin n\theta + \sum_{n=0}^N \bar{\bar{F}}_{zn} \cos n\theta$$

$$p = \sum_{n=0}^N \bar{p}_n \cos n\theta + \sum_{n=0}^N \bar{\bar{p}}_n \sin n\theta$$

where, F_x , F_y , F_z are concentrated nodal loads (in units of force/length) in the x , y and z directions respectively, and p is the pressure load (force/area). (Input details are given in notes 5, 6.) The barred coefficients in the above equations are used for the symmetric loading, whereas the double-bared coefficients are for the antisymmetric loading, with the understanding that:

$$\bar{\bar{F}}_{xn} = \bar{\bar{F}}_{yn} = \bar{\bar{F}}_{zn} = \bar{\bar{p}}_n = 0, \text{ for } n = 0 \quad (4.3)$$

Similar expansion is used for the displacement field,

$$\begin{aligned} u_x(X, Y, \theta) &= \sum_n \bar{u}_{xn}(X, Y) \cos n\theta + \sum_n \bar{\bar{u}}_{xn}(X, Y) \sin n\theta \\ u_y(X, Y, \theta) &= \sum_n \bar{u}_{yn}(X, Y) \cos n\theta + \sum_n \bar{\bar{u}}_{yn}(X, Y) \sin n\theta \\ u_z(X, Y, \theta) &= \sum_n \bar{u}_{zn}(X, Y) \sin n\theta + \sum_n \bar{\bar{u}}_{zn}(X, Y) \cos n\theta \end{aligned} \quad (4.4)$$

where,

$$\bar{u}_{xn}(X, Y) = [N_1(X, Y), N_2(X, Y), \dots] \bar{\mathbf{u}}_{xn} \quad (4.5)$$

in which $N_i(X, Y)$, $i = 1, 2, \dots$, are the shape functions, and $\bar{\mathbf{u}}_{xn}$ lists the x -components of displacement (amplitudes) at the element nodes for the n th symmetric harmonic. Similar expression can be written for the axial and tangential components of the displacement field. As used before for the loading, the single-bared amplitudes are for the symmetric response whereas the double-bared amplitudes are for the antisymmetric response.

The governing equilibrium equations are obtained as,

$$\bar{\mathbf{K}}_n \bar{\mathbf{u}}_n = \bar{\mathbf{q}}_n, \quad n = 0, 1, \dots, N \quad (4.6)$$

with similar expression for the double-barred (antisymmetric) quantities. The contribution to the r.h.s. vector from concentrated forces Equation is obtained by virtual work as,

$$\begin{aligned} \bar{\mathbf{q}}_{ni} &= \pi r_i [\bar{F}_{xn}, \bar{F}_{yn}, \bar{F}_{zn}]^T & \text{when } n = 1, 2, \dots \\ &= 2\pi r_i [\bar{F}_{xn}, \bar{F}_{yn}, 0]^T & \text{when } n = 0 \end{aligned} \quad (4.7)$$

for the symmetric case, and for the antisymmetric case we have,

$$\begin{aligned} \overline{\overline{\mathbf{q}}}_{ni} &= \pi r_i [\overline{\overline{F}}_{xn}, \overline{\overline{F}}_{yn}, \overline{\overline{F}}_{zn}]^T & \text{when } n = 1, 2, \dots \\ &= 2\pi r_i [0, 0, \overline{\overline{F}}_{zn}]^T & \text{when } n = 0 \end{aligned} \quad (4.8)$$

In the above two equations, i is a typical node where forces are applied, and r_i is the radius of the corresponding nodal circle. The coefficients F_{xn} , etc., are in units of force per length (force per unit arc length). The factor $2\pi r_i$ is to be noted when entering data (see note 6).

Notes:

1. The element connectivity must be given in the order shown in Figure 4.27 in which the node numbering sequence starts at a corner node and proceeds along the perimeter of the element in a counterclockwise direction. The element must lie in the right half of the global Cartesian XY plane with the Y axis as the axis of revolution and the X axis as the radial direction for the $\theta = 0$ plane.
2. This element is suited for modeling solids of revolution which are axially symmetric in their geometries and materials but not in their loading. Use the 2-D axisymmetric solid (NKTP = 3) if the applied loads are axisymmetric.
3. The only non-axisymmetric loads allowed are pressure loads and concentrated nodal forces. Thermal loads and body forces are assumed to be axisymmetric. Specified displacements (zero or non-zero values) are for the entire nodal circle, i.e., they are assumed to be constant in the circumferential direction.

4. If a local displacement coordinate system is specified at a node, the first two axes of the local system must lie in the global Cartesian XY plane (in which the mesh is defined). In this case, the first two axes of the local displacement system are not the radial and the axial directions, respectively. The third local axis, however, is still the tangential direction. Applied nodal loads, specified displacement boundary conditions and kinematic constraints are referred to the nodal local displacement coordinate systems (if specified). Otherwise, they are referred to the xyz system. Similarly, the internal force and reaction output is given in the local displacement systems, if they are specified at the nodes. The displacement output may be provided in the local displacement systems, in the xyz system, or in both (see *LDCASE data group). The stress output is in the xyz system.
5. Pressure loads should be entered in units of force/area. The exact pressure distribution in the circumferential direction is obtained from the product of the reference value specified in the *PRESSURE data group and the Fourier coefficients defined in the *FRCOEF data group.
6. Concentrated nodal forces should be entered for the entire 360 degrees of the model. The nodal force value for the entire nodal circle is obtained from the product of the reference value specified in the *CFORCE data group and the Fourier coefficients defined in the *FRCOEF data group. If the nodal force (ring load) in units of force/length is in the form of Equation 4.1, then the factor $2r$ (where r is the radius of the nodal circle) should be included in the definition of the force. This may be achieved by adjusting either the reference value entered in *CFORCE data group or the Fourier coefficients entered in *FRCOEF data group.
7. The output for displacements, internal forces, reactions, and stresses are only provided for the angular sections (i.e., the angles θ) specified in the *ANGSEC data group.
8. This element may only be combined with the axisymmetric shell with non-axisymmetric loading (NKTP = 37) in the same model.

4.26 Axisymmetric Shell Element (NKTP = 36)

Description

This element is based on an axisymmetric state of stress and strain and is suited for modeling shells with axisymmetric geometry and loading. The element must lie in the right half of the global XY plane. The global Y axis is the axis of revolution and the global X direction is the radial direction. The deformation is characterized by two translational components of displacement (UX, UY) and one component of rotation (ROTZ). The theoretical basis of the element is discussed in [Section 2.8](#).

The element can be shaped as a 2 to 4 node line element depending on the selected NORDR value. The element configuration, node locations and face numbering convention for top and bottom surfaces are shown in [Figure 4.29](#). An element reference guide is briefed in [Table 4.56](#) and the available loadings listed in [Table 4.57](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.56](#) gives the output pertinent to this element.

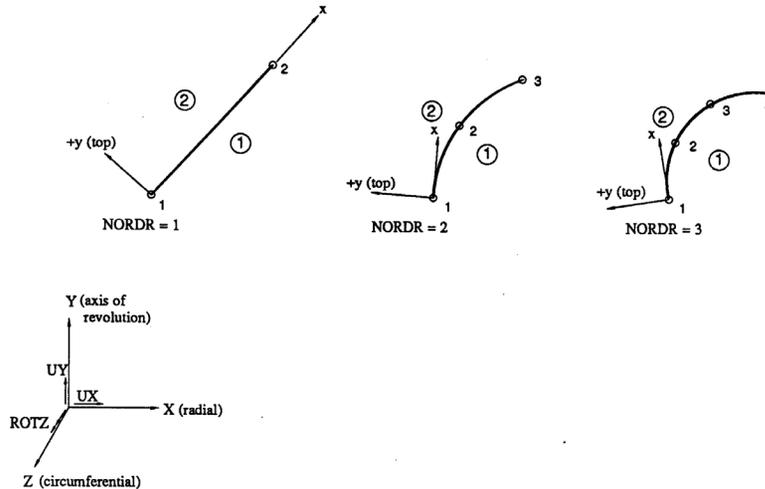


Figure 4.29: Element configuration for available NORDR values, NKTP = 36 (circled numbers indicate face numbers; 1 for bottom; 2 for top)

Table 4.56: Element reference guide (NKTP = 36)

Element Type	NKTP = 36, Axisymmetric shell element
Analysis Types	Static, Dynamic, Buckling
Degrees of Freedom	3 per node: UX, UY, ROTZ
NORDR (Shape / No. of nodes)	- Line: 2 to 4 nodes (NORDR = 1 to 3)
Real Constants	2 to 4 nodal thicknesses (same as number of nodes)
Material Properties	
Isotropic Elastic	4 properties: EX, NUXY, DENS, ALPX
Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY, MZ) and strain energy - Centroidal strains at the top, middle and bottom surface in element local directions (Exx, Ezz, Exy, where x is the tangent to the line element, and z is the circumferential direction), and in global directions (EXX, EYY, EZZ, EXY) - Element stresses at the centroid, Gauss and nodal points for top, middle and bottom surfaces in element local directions (Sxx, Szz, Sxy, where x is the tangent to the line element, and z is the circumferential direction), and in global directions (SXX, SYY, SZZ, SXY) - Element principal stresses (S1, S2, S3), von Mises equivalent stress, octahedral and maximum shear stresses at the centroid, Gauss and nodal points for top, middle, and bottom surfaces.
Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration and shock spectrum analyses

Table 4.57: Available loading (NKTP = 36)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces (FX, FY, MZ) in global X (radial), Y (axis of revolution) and Z (circumferential) directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE and note 1 - Specified non-zero nodal displacements UX, UY and nodal rotations ROTZ in global directions (or in local displacement system, if defined), see *SPDISP - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER and note 1
----------------------	---

Pressure Loading	- Uniform or non-uniform pressure (force/area) on top or bottom faces of the element, see *PRESSURE or *DPRESSURE and note 2
Thermal Loading	- Specified nodal temperature and nodal temperature difference in element thickness direction, see *NDTEMPER and *NDTEMPDIF
Body Force	- Gravity loading or body force due to linear acceleration in global X and Y directions, see *BODYFORCE and note 3 - Centrifugal loads due to angular velocity about the global Y direction, see *BODY-FORCE
Ground Motion	- Forces due to ground motion in global X and/or Y directions, see *GROUND and note 3

Notes:

1. Concentrated loads and reactions are for the entire 360 degree circumference of the model.
2. Pressure loads can be applied to the bottom or top face of the element. The bottom and top faces are numbered as 1 and 2, respectively as shown in [Figure 4.29](#). The pressure should be entered in units of force/area.
3. Caution must be taken that the global X direction is actually the radial direction.

4.27 Axisymmetric Shell with Non-Axisymmetric Loading (NKTP = 37)

4.27.1 Description

This element is used for modeling of a shell of revolution which is axially symmetric in its geometry and material but not in its loading. The Fourier series method is used to express the non-axisymmetric loading (in the circumferential coordinate θ) as the sum of several component loadings (harmonics). Since the Fourier harmonics are not coupled, the original 3-D problem is transformed to a series of 2-D problems that do not interact [4.6].

The solution for each harmonic component is obtained separately using the same two-dimensional mesh (a cross-section passing through the axis of revolution). The solution to the original problem is obtained by adding the solutions of the harmonic components. The procedure is automated such that the response (displacements, stresses, etc.) may be provided at any specified angle θ .

The element must lie in the right half of the global Cartesian XY plane (this is also the plane of the circumferential coordinate $\theta = 0$ implied in the model) as shown in [Figure 4.30](#). The global Y axis is the axis of revolution. The global X and Z axes are the radial and tangential directions for the $\theta = 0$ plane, respectively. At any angle θ , the xyz system is used to express the components of motion and forces, such that x is the radial direction, y (parallel to the global Y axis) is the axial direction, and z is the tangential direction. At the $\theta = 0$ plane, the xyz system is parallel to the global Cartesian XYZ system. A positive angle is measured from the global X axis rotating about the negative Y axis. The output for displacements, internal forces and reactions for any angular section is provided in the xyz system (radial, axial, tangential), see note 4.

The element has six degrees of freedom per node: UX (radial, along x), UY (axial, along y), UZ (tangential, along z), ROTX (rotation about x axis), ROTY (rotation about y axis), and ROTZ (rotation about z axis), but possesses no rotational stiffness about the normal to the shell surface. The degrees of freedom UX, ROTX and ROTY are present to account for 3-D behavior under the action of non-axisymmetric loading. The state of stress is characterized by six components (Sxx, Syy, Szz, Sxy, Syz, Sxz) referring to the xyz system at any angular section, however, stress normal to the shell surface ($\sigma_{\eta\eta}$) is ignored due to shell

assumptions. The element uses a local coordinate system $\xi\eta$ as shown in [Figure 4.31](#) where ξ is tangent to the line element and η is normal to the shell midsurface.

The element can be shaped as a 2 to 4 node line element depending on the selected NORDR value. The element configuration, node locations and face numbering convention are shown in [Figure 4.31](#). An element reference guide is given in [Table 4.58](#) with the available loading given in [Table 4.59](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.58](#) gives the output pertinent to this element.

The only non-axisymmetric loads that may be applied to this element are concentrated loads (ring loads) and pressure loads. Body forces and thermal loading must be axisymmetric.

[Section 4.27.2](#) has some background information regarding the Fourier series representation of non-axisymmetric loads and their classification to symmetric and antisymmetric components about the $\theta = 0$ plane.

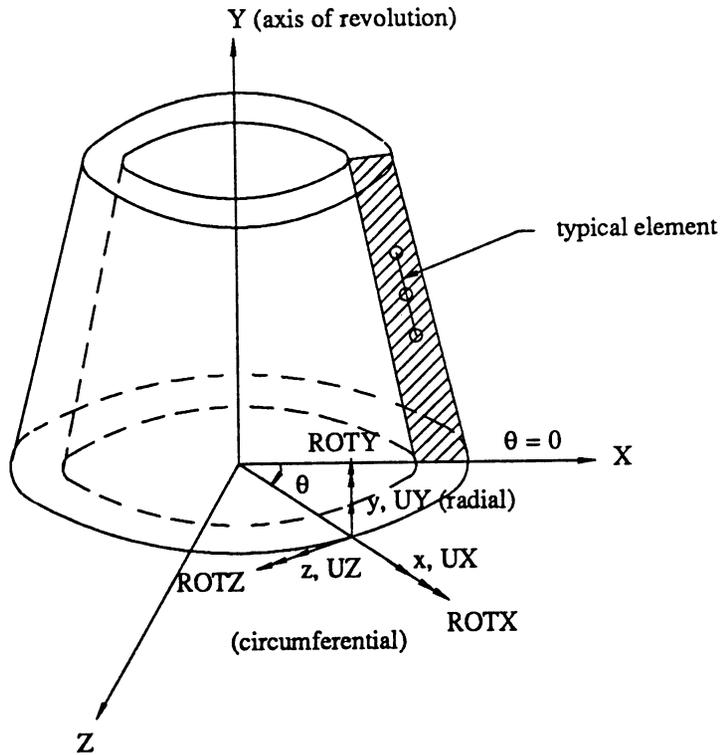


Figure 4.30: Coordinate system for NKTP = 37

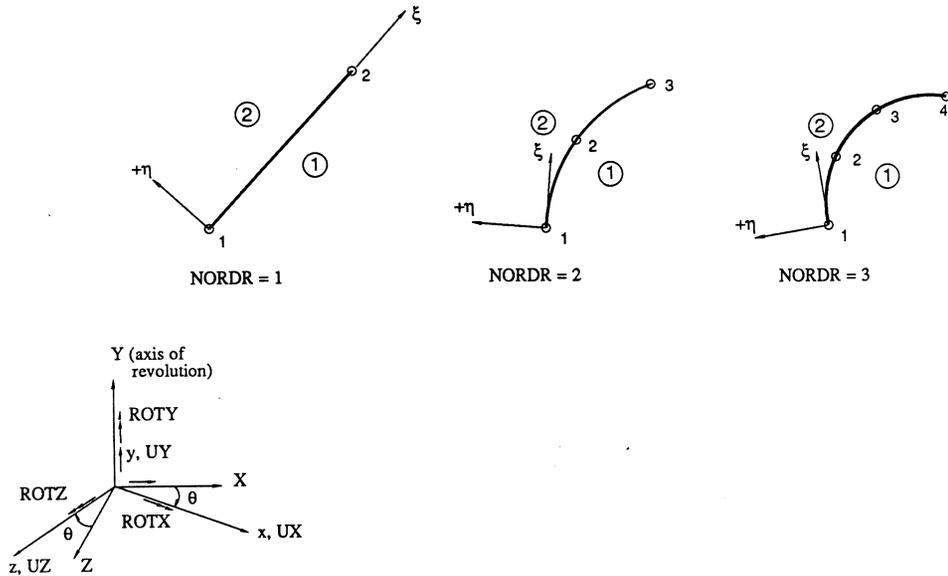


Figure 4.31: Element configuration for available NORDR, NKTP = 37 (circled numbers indicate face numbers; 1 for bottom; 2 for top)

Table 4.58: Element reference guide (NKTP = 37)

Element Type	NKTP = 37, Axisymmetric solid with non-axisymmetric loading
Analysis Types	Static
Degrees of Freedom	6 per node: UX (radial), UY (axial), UZ (tangential), ROTX, ROTY, ROTZ (about x, y and z axes, respectively)
NORDR (Shape / No. of nodes)	- 2 to 4 (NORDR = 1 to 3)
Real Constants	2 to 4 nodal thicknesses (same as number of nodes)
Material Properties	

Element Library

Axisymmetric Shell with Non-Axisymmetric Loading (NKTP = 37)

<p>Isotropic Elastic Output</p>	<p>4 properties: EX, NUXY, DENS, ALPX</p> <ul style="list-style-type: none"> - Displacement solution at the specified angular sections. - Internal forces (Fx, Fy, Fz, Mx, My, Mz) and reaction forces (both per unit radian) at the specified angular sections. - Element stresses at centroid, Gauss and nodal points for top, middle and bottom surfaces in element local directions $\sigma_{\xi\xi}$, σ_{zz}, $\sigma_{\xi\eta}$, $\sigma_{\xi z}$, $\sigma_{\eta z}$ and in the xyz system (Sxx, Syy, Szz, Sxy, Sxz, Syz) at the specified angular sections. - Element principal stresses (S1, S2, S3), von Mises equivalent stress, octahedral and maximum shear stresses for the top, middle and bottom surface at the specified angular sections. - Averaged nodal stresses for top, middle and bottom surface at the specified angular sections.
--	---

* notes are given after [Section 4.27.2](#)

Table 4.59: Available loading (NKTP = 37)

<p>Nodal Loading</p>	<ul style="list-style-type: none"> - Concentrated nodal forces (Fx, Fy, Fz) and moments (Mx, My, Mz) in the xyz system (or in local displacement system, if defined). The ring load varies harmonically in the circumferential direction. see notes 4, 6* - Specified non-zero nodal displacements for a nodal circle (assumed to be constant in the circumferential direction). See *SPDISP and note 3
<p>Pressure Loading</p>	<ul style="list-style-type: none"> - Uniform or non-uniform pressure (force/area) on any face of the element. The pressure load varies harmonically in the circumferential direction. see note 5.
<p>Thermal Loading (axisymmetric)</p>	<ul style="list-style-type: none"> - Specified nodal temperature, see *NDTEMPER. The thermal loads are assumed to be axisymmetric
<p>Body Force (axisymmetric)</p>	<ul style="list-style-type: none"> - Gravity loading or body forces due to linear acceleration in the X (radial) and Y directions, see *BODYFORCE - Centrifugal loads due to angular velocity about the axis of revolution (global Y axis) see *BODYFORCE

* notes are given after [Section 4.27.2](#)

4.27.2 Fourier Series Representation for Non-Axisymmetric Loads

The Fourier series expansion for a non-axisymmetric load F is given by,

$$F(\theta) = a_0 + \sum_{n=1}^N a_n \cos n\theta + \sum_{n=1}^N b_n \sin n\theta \quad (4.9)$$

where,

$$a_0 = \frac{1}{2\pi} \int_{-\pi}^{\pi} F(\theta) d\theta$$

$$a_n = \frac{1}{\pi} \int_{-\pi}^{\pi} F(\theta) \cos n\theta d\theta \quad (4.10)$$

$$b_n = \frac{1}{\pi} \int_{-\pi}^{\pi} F(\theta) \sin n\theta d\theta$$

where, θ is the circumferential coordinate implied in the model. The number of harmonic terms in the series (N in Equation 4.9) required for acceptable representation of the loading depends on the complexity of the loading, and should be selected by observing the plot of the function with increasing number of terms.

The loads and displacements are classified to symmetric and antisymmetric about the $\theta = 0$ plane for NKTP = 37. Details about the classification and the Fourier series expansion for concentrated force (F_x , F_y , F_z), pressure and displacements (U_x , U_y , U_z) are given in Section 4.33.2. For the applied moment M_z , the constant term and the cosine terms represent antisymmetric loading about the $\theta = 0$ plane and the sine terms represent antisymmetric loading about the $\theta = 0$ plane. However, in order to apply a symmetric moments (M_x , M_y), these two moments have to change direction for $\theta > \pi$ (see Figure 4.32). Therefore, the moment (M_x , M_y) terms in Equation 4.9 have the following meaning: the constant term represent antisymmetric constant moment, the sine terms represent symmetric loading about the $\theta = 0$ plane and the cosine terms represent antisymmetric loading about the $\theta = 0$ plane. The classification of symmetric and antisymmetric components of the loading is depicted in Figure 4.32.

Element Library

Axisymmetric Shell with Non-Axisymmetric Loading (NKTP = 37)

Decomposing the non-axisymmetric nodal moments into symmetric and antisymmetric component loadings about the ($\theta = 0$ plane), we have [4.6],

$$\begin{aligned}M_x &= \sum_{n=0}^N \bar{M}_{xn} \sin n\theta + \sum_{n=0}^N \bar{\bar{M}}_{xn} \cos n\theta \\M_y &= \sum_{n=0}^N \bar{M}_{yn} \sin n\theta + \sum_{n=0}^N \bar{\bar{M}}_{yn} \cos n\theta \\M_z &= \sum_{n=0}^N \bar{M}_{zn} \cos n\theta + \sum_{n=0}^N \bar{\bar{M}}_{zn} \sin n\theta\end{aligned}\tag{4.11}$$

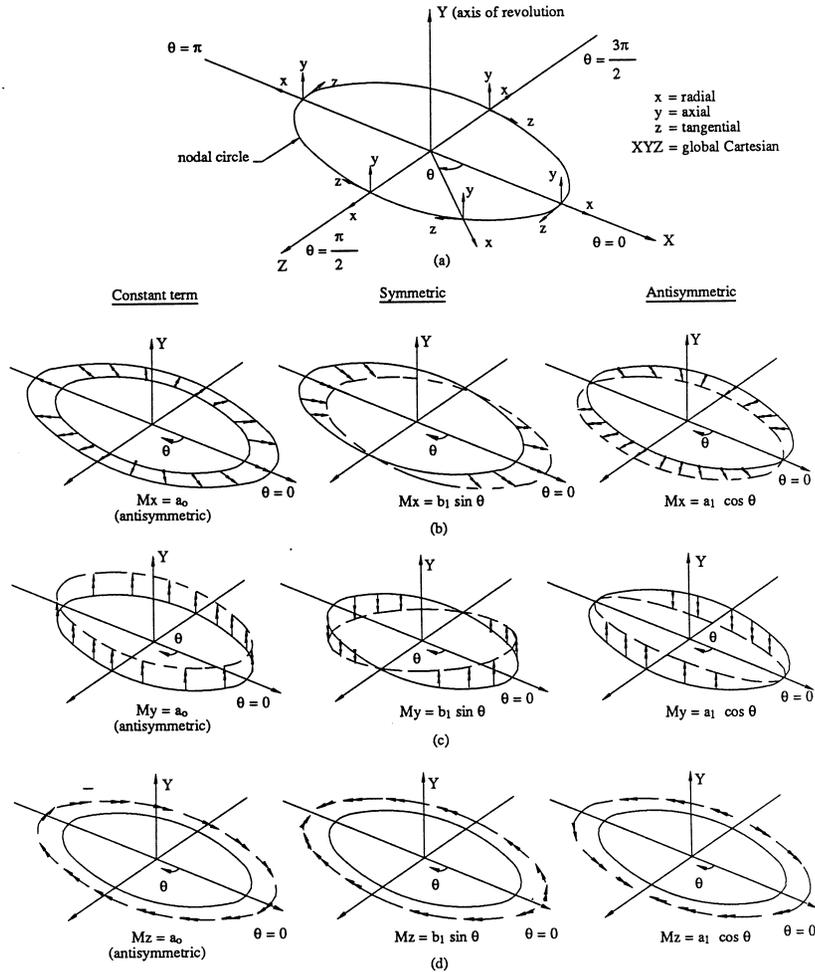


Figure 4.32: Examples of symmetric and antisymmetric loading
 (a) xyz coordinate system for different angle for applied moment;
 (b) moment about x axis (M_x);
 (c) moment about y axis (M_y);
 (d) moment about z axis (M_z)

where, M_x , M_y , M_z are concentrated nodal moments (in unit of moment/length) in the x, y and z directions, respectively. (Input details are given in notes 5, 6). The barred coefficients

in the above equations are used for the symmetric loading, whereas the double-barred coefficients are for the antisymmetric loading, with the understanding that,

$$\bar{M}_{xn} = \bar{M}_{yn} = \bar{\bar{M}}_{z\theta} = 0, \quad \text{for } n = 0 \quad (4.12)$$

Similar expansion is used for the rotations,

$$\begin{aligned} \theta_x(X, Y, \theta) &= \sum_{n=0}^N \bar{\theta}_{xn}(X, Y) \sin n\theta + \sum_{n=0}^N \bar{\bar{\theta}}_{xn}(X, Y) \cos n\theta \\ \theta_y(X, Y, \theta) &= \sum_{n=0}^N \bar{\theta}_{yn}(X, Y) \sin n\theta + \sum_{n=0}^N \bar{\bar{\theta}}_{yn}(X, Y) \cos n\theta \\ \theta_z(X, Y, \theta) &= \sum_{n=0}^N \bar{\theta}_{zn}(X, Y) \cos n\theta + \sum_{n=0}^N \bar{\bar{\theta}}_{zn}(X, Y) \sin n\theta \end{aligned} \quad (4.13)$$

where,

$$\bar{\theta}_x(X, Y) = [N_1(X, Y), N_1(X, Y), \dots] \bar{\theta}_{xn} \quad (4.14)$$

in which $(N_i(X, Y), i = 1, 2, \dots)$ are the shape functions, and θ_{xn} list the x-components of rotations (amplitudes) at the element nodes for the nth symmetric harmonic. Similar expression can be written for the other components of rotations. As used before for the moment, the single-barred amplitudes are for the symmetric response whereas the double-barred amplitudes are for the antisymmetric response.

The governing equilibrium equations are obtained as,

$$\bar{\bar{\mathbf{K}}}_n \bar{\mathbf{u}}_n = \bar{\mathbf{q}}_n \quad (4.15)$$

where,

$$\bar{\mathbf{u}}_n = [\bar{U}_{xn}, \bar{U}_{yn}, \bar{U}_{zn}, \bar{\theta}_{xn}, \bar{\theta}_{yn}, \bar{\theta}_{zn}]^T$$

and,

$$n = 0, 1 \dots N$$

with similar expression for the double-barred (antisymmetric) quantities. The contribution to the r.h.s. vector from concentrated forces and moments (Equation and Equation 4.11.) is obtained by virtual work as,

$$\begin{aligned} \bar{\mathbf{q}}_{ni} &= \pi r_i [\bar{F}_{xn}, \bar{F}_{yn}, \bar{F}_{zn}, \bar{M}_{xn}, \bar{M}_{yn}, \bar{M}_{zn}] \quad \text{when } n = 1, 2, \dots \\ &= 2\pi r_i [\bar{F}_{xn}, \bar{F}_{yn}, \bar{F}_{zn}, 0, 0, \bar{M}_{zn}]^T \quad \text{when } n = 0 \end{aligned} \quad (4.16)$$

for the symmetric case, and for the antisymmetric case we have,

$$\begin{aligned} \bar{\mathbf{q}}_{ni} &= \pi r_i [\bar{\bar{F}}_{xn}, \bar{\bar{F}}_{yn}, \bar{\bar{F}}_{zn}, \bar{\bar{M}}_{xn}, \bar{\bar{M}}_{yn}, \bar{\bar{M}}_{zn}]^T \quad \text{when } n = 1, 2, \dots \\ &= 2\pi r_i [0, 0, \bar{F}_{zn}, \bar{\bar{M}}_{xn}, \bar{\bar{M}}_{yn}, 0]^T \quad \text{when } n = 0 \end{aligned} \quad (4.17)$$

In the above two equations, i is a typical node where loads are applied, and r_i is the radius of the corresponding nodal circle. The coefficients \bar{F}_{xn} (or \bar{M}_{xn}), etc., are in units of force per length (moment per length). It should be noted that the factor $2\pi r_i$ should be included in the input data (see note 6).

Notes:

1. The element connectivity for different NORDR value are given in Figure 4.31. The element must lie in the right half the global Cartesian XY plane with Y axis as the axis of revolution and the X axis as the radial direction for the $\theta = 0$ plane.
2. This element is suited for modeling shells of revolution which are axially symmetric in their geometries but not in their loading. Use the 2-D axisymmetric (NKTP = 36) if the applied loads are axisymmetric.
3. The only non-axisymmetric loads allowed are pressure loads and concentrated nodal loads. Thermal loads and body forces are assumed to be axisymmetric. Specified displacements (zero or non-zero values) are for the entire nodal circle, i.e., they are assumed to be constant in the circumferential direction.

4. If a local displacement coordinate system is specified at a node, the first two axes of the local system must lie in the global Cartesian XY plane (in which the mesh is defined). In this case, the first two axes of the local displacement system are not the radial and the axial directions, respectively. The third local axis, however, is still the tangential direction. Applied nodal loads, specified displacement boundary conditions and kinematic constraints are referred to the nodal local displacement coordinate systems (if specified). Otherwise, they are referred to the xyz system. Similarly, the internal force and reaction output is given in the local displacement systems, if they are specified at the nodes. The displacement output may be provided in the local displacement systems, in the xyz system, or in both (see *LDCASE data group). The stress output is in the xyz system.
5. Pressure loads can be applied to the bottom or top face of the element. The bottom and top faces are numbered as 1 and 2 respectively as shown in [Figure 4.31](#). Pressure loads should be entered in units of force/area. The exact pressure distribution in the circumferential direction is obtained from the product of the reference value specified in the *PRESSURE data group and the Fourier coefficients defined in the *FRCOEF data group.
6. Concentrated nodal forces should be entered for the entire 360 degrees of the model. The nodal force value for the entire nodal circle is obtained from the product of the reference value specified in the *CFORCE data group and the Fourier coefficients defined in the *FRCOEF data group. If the nodal force (ring load) in units of force/length (or moment/length for concentrated moments) is in the form of [Equation 4.9](#), then the factor $2\pi r$ (where r is the radius of the nodal circle) should be included in the definition of the force. This may be achieved by adjusting either the reference value entered in *CFORCE data group or the Fourier coefficients entered in *FRCEOF data group.
7. The output for displacements, internal forces, reactions, and stresses are only provided for the angular sections (i.e., the angles) specified in the *ANGSEC data group.
8. This element may only be combined with the axisymmetric solid with non-axisymmetric loading (NKTP = 34) in the same model.

4.28 3-D General Spring Element (NKTP = 38)

Description

This element is a 2-node 3-D general spring element with six independent spring rates and six degrees of freedom (UX, UY, UZ, ROTX, ROTY, ROTZ) per node. The spring rates are defined in the element local coordinate system. The local x-axis of the spring is along the line joining the two nodes (or along a specified direction if the two nodes are coincident). The local y and z axes are defined using an orientation vector or a third point to define the local xy plane, see *ELEMENTS and note 1 for input details.

The element configuration is shown in Table 4.60. An element reference guide is given in Table 4.60 with the available loading given in Table 4.61. A general description of the element input data and output options are given in Section 4.1.2 and Section 4.1.3 respectively. Table 4.60 gives the output pertinent to this element.

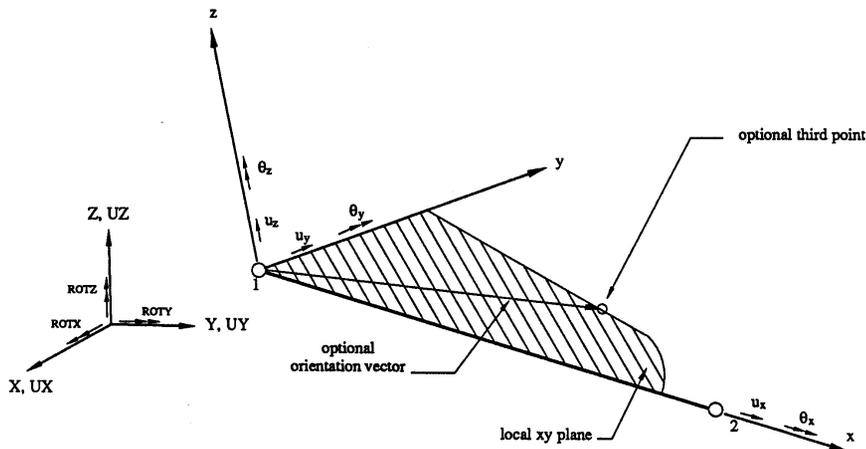


Figure 4.33: 3-D general spring element (NKTP = 38, NORDR = 1)

Element Library

3-D General Spring Element (NKTP = 38)

Table 4.60: Element reference guide (NKTP = 38)

Element Type	NKTP = 38, 3-D General spring element
Analysis Types	Static, Dynamic, Buckling, nonlinear static see note 2
Degrees of Freedom	6 per node: UX, UY, UZ, ROTX, ROTY, ROTZ
NORDR (Shape / No. of nodes)	- Line: 2 nodes (NORDR = 1)
Real Constants (RCTABLE)	<p>Up to 22 constants: KX, KY, KZ, KRX, KRY, KRZ, V1, V2, V3, ISYS, CX, CY, CZ, CRX, CRY, CRZ, IDPLX, IDPLY, IDPLZ, IDPLRX, IDPLRY, and IDPLRZ (only the first 6 constants are required if the two nodes are not coincident and no damper is required; the last six constants are applicable only in linear and non-linear transient analysis)</p> <p>Where,</p> <p>KX, KY, KZ spring constants (force/length) in local x, y and z directions, respectively.</p> <p>KRX, KRY, KRZ torsional spring constants (moment/radian) about local x, y and z axes respectively</p> <p>V1, V2, V3, components of a vector pointing in the spring axis direction (if the two nodes are</p> <p>ISYS Coordinate system in which the components V1, V2, V3, are given</p> <p> = 0 global cartesian system</p> <p> = 1 local displacement coordinate system at first node, see = note 3</p> <p> = 2 local displacement coordinate system at first node, see = note 3</p> <p>CX, CY, CZ damping constants (force-time/length) in local x, y, z direction</p> <p>CRX, CRY, CRZ rotational damping constants (moment-time/radian) about local x, y, z directions, respectively</p> <p>IDPLX, IDPLY, IDPLZ Deformation curve ID defined in *PLSPRING data group used for translation spring in X, Y, and Z directions. These curves can be used in material nonlinear analysis to simulate plastic hinge. If these curves are not given, KX, KY, and KZ will be used.</p>

	<p>IDPLRX, IDPLRY, IDPLRZ</p> <p>Deformation curve ID defined in *PLSPRING data group used for torsion spring in X, Y and Z directions. These curves can be used in material nonlinear analysis to simulate Plastic hinge. If these curves are not given, KRX, KRY, and KRZ will be used.</p>
Material Properties	Material ID (optional); see note 4
Element Output -	Internal forces (FX, FY, FZ, MX, MY, MZ) at the two ends in global coordinate system (or in local displacement system, if defined), and strain energy
Dynamic Capabilities	<ul style="list-style-type: none"> - Null mass matrix is assumed, see note 2 - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration and shock spectrum analyses

Table 4.61: Available loading (NKTP = 38)

<p>Nodal Loading</p>	<ul style="list-style-type: none"> - Concentrated nodal forces and moments in global X, Y and Z directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal rotations UX, UY, UZ and ROTX, ROTY and ROTZ in global X, Y and Z directions (or in local displacement system, if defined), see *SPDISP - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
-----------------------------	--

Notes:

1. The six spring constants are applied in the element local coordinate system, see [Figure 4.33](#). A third point or an orientation vector lying in the local xy plane should be defined, along with the element connectivity, in the *ELEMENTS data group. If neither is defined, the six spring constants are applied directly in the global coordinate system regardless of the direction of the spring local x-axis.
2. This spring element is a massless element (a null mass matrix is assumed). It may be used in dynamic analyses provided that it is connected to other elements with non-zero masses. Similarly, a null geometric stiffness matrix is assumed in buckling analysis and the element must be connected to other elements having non-zero geometric stiffness matrices.
3. If the two nodes of the element are not coincident, only the first 6 real constants, the spring rates, are required. If the two nodes are coincident, a direction vector ($V_{i,i=1,2,3}$) is required. The direction vector is described in the coordinate system indicated by ISYS. If ISYS = 1 or 2, it is assumed that the local displacement coordinate system is defined at the referenced node (node 1 or node 2), otherwise ISYS is reset to zero.
4. Material damping in the modal dynamic analysis can be applied to this spring element. For the elements with material damping, a material ID should be specified in eigen value analysis to be used later in the modal dynamic analysis. Even though material properties are not required for the spring element, a dummy material property table should be given for the material ID used for the spring element. For any other analysis without material damping, the material ID for the spring element is not required.
5. The third point or the orientation vector defining the spring orientation is not updated during the deformation. These spring elements are limited to small deformation or material nonlinear analysis only

4.29 3-D General Beam Element (NKTP = 39)

Description

This is a 2-node prismatic 3-D beam element. The formulation includes stretching, bending and torsion effects and accounts for geometric and material nonlinear behavior. The beam is based on Kirchhoff assumption, therefore no transverse shear deflection effect is considered. The beam vertices may be offset from the corresponding nodal points and the centroid may be offset from the shear center, see [Figure 4.34](#). The element has end release capability, see note 5 below. The deformation is characterized by three translations (UX, UY, UZ) and three rotations (ROTX, ROTY, ROTZ). The local x-axis of the beam is along the centroidal axis. The local y-and z-axes are user defined and not necessary principal axes of the cross section. The theoretical basis of the element is discussed in [Section 2.8](#).

Two different ways are available for defining the beam cross section. These are the general cross section and the standard cross section. In the general cross section description, sectional properties are defined by integrated quantities such as moment of inertia, torsional constant, etc. In the standard cross section description, the user can choose from a library of available known sections.

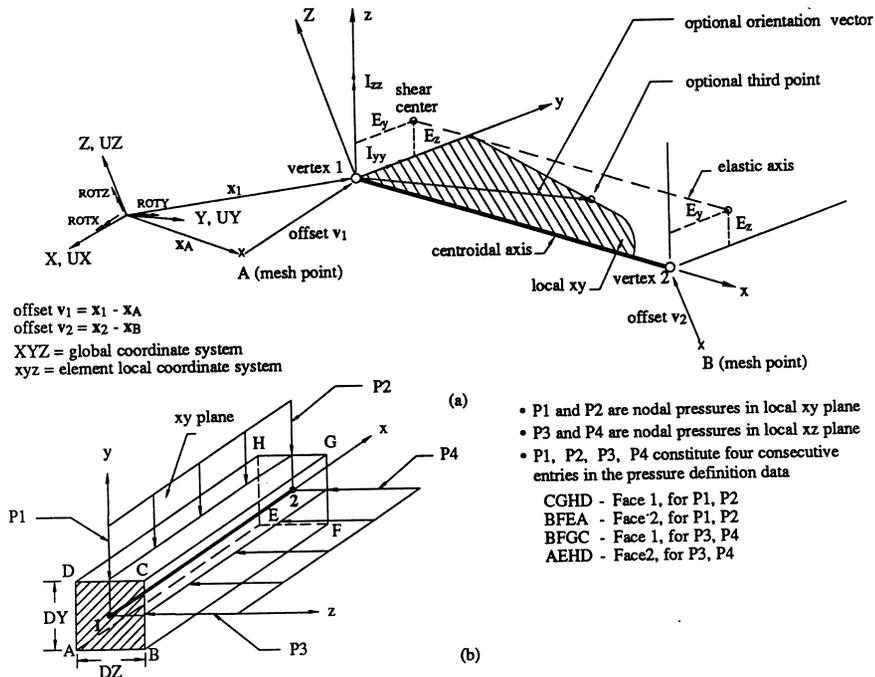


Figure 4.34: 3-D beam element, NKTP = 39, NORDR = 1,
 (a) Element configuration and orientation;
 (b) Face numbering convention for pressure loading

Currently, there are: rectangular, circular, circular pipe, box, Symmetric I, L and trapezoidal sections. The user can choose either cross section (general or standard) for linear and geometrically nonlinear analyses whereas for materially nonlinear analysis, only standard sections may be used. General sections are, however, recommended for linear and geometrically nonlinear analyses.

The element configuration, node locations and face numbering convention are shown in [Figure 4.34](#). An element reference guide is briefed in [Table 4.62](#) with the available loading listed in [Table 4.63](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.62](#) gives the output pertinent to this element. Two other data groups are necessary to complete the definition of this beam element. These are: the *BMSECT group ([Section 6.3.8](#)) for specifying cross section data and the *BMDATA group ([Section 6.3.7](#)) for specifying miscellaneous beam data.

Table 4.62: Element reference guide (NKTP = 39)

Element Type	NKTP = 39, 3-D General Beam element
Analysis Types	Static, Dynamic, Buckling, Nonlinear Static, Nonlinear Transient
Degrees Of Freedom	6 per node: UX, UY, UZ, ROTX, ROTY, ROTZ
NORDR (Shape / No. Of Nodes)	- Line: 2 nodes (NORDR = 1)
Real Constants	No real constant table *RCTABLE is required. Beam data and beam section data are input in *BMDATA (Section 6.3.7) and *BMSECT (Section 6.3.8) data groups, respectively.
Material Properties	
Isotropic Elastic	4 properties: EX, NUXY, DENS, ALPX
Elastoplastic	see *PLASTIC
Creep Law	see *CREEP
Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ, MX, MY, MZ) at beam ends in global coordinate system (or in local displacement system, if defined), and strain energy - Resultant forces and moments at beam ends in beam local coordinate system (FX, FY, FZ, MX, MY, MZ) - For standard cross-section, element local stresses (SXX, SXY, SXZ), principal stresses (S1, S2) and maximum shear stress (TAU) at default points, see note 1
Nonlinear Capabilities	<ul style="list-style-type: none"> - Geometric nonlinearity: large displacements, rotations, small strain; updated Lagrangian formulation; deformation dependent loads - Material nonlinearity: elastoplastic material models
Dynamic Capabilities	Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration and shock spectrum analyses

Table 4.63: Available loading (NKTP = 39)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X, Y and Z directions and concentrated nodal moments about global X, Y and Z axes (or in local displacement system, if defined), see *CFORCE - Specified non-zero nodal displacements UX, UY, UZ and nodal rotations ROTX, ROTY, ROTZ in global directions (or in local displacement system, if defined), see *SPDISP
Pressure and Partially Distributed Loading	<ul style="list-style-type: none"> - Uniform or non-uniform pressure (force/length) on top or bottom face of the element; see *PRESSURE, *BEAMLOAD and notes 3,4. - Uniform or non-uniform follower (deformation dependent) pressure (force/length) on top or bottom face of the element; see *PRESSURE, *BEAMLOAD and notes 3, 4.
Thermal Loading	<ul style="list-style-type: none"> - Specified nodal temperature and nodal temperature difference in element local y and z directions, see *NDTEMPER, *NDTEMPDIF and note 2.
Body Force	<ul style="list-style-type: none"> - Gravity loading or body forces due to linear acceleration in global X, Y and Z directions, see *BODYFORCE - Centrifugal loads due to angular velocity about the global X, Y and Z axes, see *BODYFORCE - Tangential loads due to angular acceleration about the global X, Y, and Z axes, see *BODYFORCE

Notes:

1. The stress resultants in the beam local coordinate system (normal force, shear forces, bending moments and twisting moment) are computed at the beam ends. If general cross-section is used, stress output is available for the current release, while if standard cross section is used, the stresses are output at the two end of the beam. The stresses at the cross section are calculated certain point where the extreme stresses might occur.
2. Thermal loads can cause axial and/or bending deformation of the beam element. Nodal temperatures (specified in *NDTEMPER data group) are used to compute axial expansion or contraction. temperature differences (*NDTEMPDIF data group) will be used in conjunction with the beam in both the y-direction and z-direction (DY and DZ, see *BMSECT) to compute beam bending local xy and xz planes, respectively. If the temperature gradient loading is to be applied in only direction, enter zero value for the depth in the other direction.
3. Pressure loading can be applied to any or all of the four faces of the beam element. The numbering is shown in [Figure 4.34](#). Positive pressure loading is applied toward the face

and parallel the local axes of the beam cross-section. The unit of this loading must be force/length. reference to [Figure 4.34\(b\)](#), it should be noted that P1, P2, P3, and P4 constitute four consecutive entries in the pressure definition data. Therefore, if the pressure is applied in only one direction, appropriate entries must be set to zero.

4. A partially distributed load, concentrated force or concentrated moment (all defined in the beam coordinate system) at arbitrary points along the length of the beam element can be specified *BEAMLOAD data group.
5. End release can be specified in the *BMDATA data group to release up to five of the six degrees freedom (DOF), defined in the beam local coordinate system, at either end of the element. A released degree of freedom means that the connection between this DOF and the corresponding point removed (e.g., to simulate a hinge connection). Singularity problems may arise if the assembled stiffness associated with a released DOF is not properly compensated for from neighboring elements. Therefore, the end release option should be used with caution. For example, if the same DOF released at the common node of two colinear elements, the released DOF should be constrained zero. Internally, automatic constraints are applied if all beam elements connected to a common have all translations and/or all rotations released (at the common node).

4.30 3-D Thin Shell Element (NKTP = 40)

Description

This shell element includes membrane and bending deformations and is suited for modeling thin shells. The element is formulated by applying the Kirchhoff assumption of zero transverse shear strains at some discrete points. Thus, transverse shear effects are not included in the formulation. The 3-D general shell element (NKTP = 20) should be used if the effect of transverse shear deformation is considered to be significant. The element has six degrees of freedom per node (UX, UY, UZ, ROTX, ROTY, ROTZ), but it possesses no rotational stiffness about the normal to the shell surface (note 3). The theoretical basis of the element is discussed in [Section 2.8](#). An element reference guide is given in [Table 4.64](#) with the available loading listed in [Table 4.65](#).

The element can be shaped as a 4 node quadrilateral or a 3 node triangle depending on the selected NORDR value. The element configuration, node locations and face numbering convention for top and bottom surfaces are shown in [Figure 4.35](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.64](#) gives the output pertinent to this element.

Element Library

3-D Thin Shell Element (NKTP = 40)

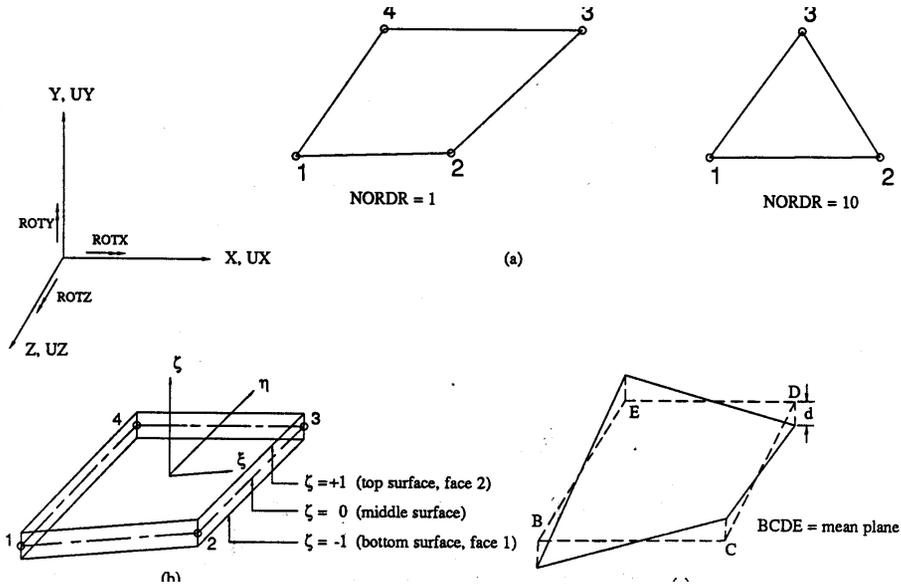


Figure 4.35: 3-D thin shell (NKTP = 40), (a) element configuration,
(b) face numbering convention for top and bottom surfaces,
(c) warped quadrilateral

Element Library

3-D Thin Shell Element (NKTP = 40)

Table 4.64: Element reference guide (NKTP = 40)

Element Type	NKTP = 40, 3-D Thin shell element
Analysis Types	Static, Dynamic, Buckling, Nonlinear static
Degrees of Freedom	6 per node: UX, UY, UZ, ROTX, ROTY, ROTZ
NORDR (Shape / No. of nodes)	- Quadrilateral: 4 nodes (NORDR = 1)a Triangle: 3 nodes (NORDR = 10)
Real Constants (RCTABLE)	13th entry:0-No offset 1 - Bottom surface is offset to the mid-plane 2 - Top surface is offset to the mid-plane 3 - User defined offset If the 13th entry is '3', then the next (up to) 4 entries correspond to the nodal offset values. It should be noted that the offset option should not be used for nonlinear problems.
Material Properties Isotropic Elastic	4 properties: EX, NUXY, DENS, ALPX
Orthotropic Elastic	7 properties: EX, EY, NUXY, GXY, DENS, ALPX, ALPY, (see note 4)
Elastoplastic	For nonlinear static only, see *PLASTIC and note 7

Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ, MX, MY, MZ) and strain energy - Centroidal strains at the top, middle and bottom surface in element local directions (EXX, EYY, EXY) and/or in global directions (EXX, EYY, EZZ, EXY, EYZ, EXZ)^a - Element stresses at the centroid, Gauss and nodal points for top, middle, and bottom surfaces in element local directions (SXX, SYX, SXY) and in global directions (SXX, SYX, SZZ, SXY, SYZ, SXZ)
	<ul style="list-style-type: none"> - Element principal stresses (S1, S2, S3), von Mises equivalent stress, octahedral and maximum shear stresses at the centroid, Gauss and nodal points for top, middle, and bottom surfaces - Equivalent plastic strain (EPS), and effective yield stress (YLD), for nonlinear static only, at Gauss and nodal points, see note 6 - Stress resultants (NXX, NYY, NXY, MXX, MYX, MXY, QYZ, QXZ) at nodal points. (see note 8)
Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration, and shock spectrum analyses
Nonlinear Capabilities	<ul style="list-style-type: none"> - Geometric nonlinearity, large displacement and rotation; updated Lagrangian formulations; deformation dependent loads - Material nonlinearity; elastoplastic material model

a. Not available for nonlinear static

Table 4.65: Available loading (NKTP = 40)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X, Y and Z directions (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal displacements UX, UY, UZ and nodal rotations ROTX, ROTY, ROTZ in global directions (or in local displacement system, if defined), see *SPDISP - Concentrated nodal follower (deformation dependent) forces perpendicular to the middle surface of the element, for nonlinear static only, see *CFOLLOWER - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
----------------------	--

Element Library

3-D Thin Shell Element (NKTP = 40)

Pressure Loading	<ul style="list-style-type: none">- Uniform or non-uniform pressure (force/area) on top or bottom faces of the element, see *PRESSURE or *DPRESSURE and note 2- Uniform or non-uniform pressure and deformation dependent pressure (force/area) on top or bottom faces of the element, for nonlinear static only, see *PRESSURE
Thermal Loading	<ul style="list-style-type: none">- Specified nodal temperature and nodal temperature difference in element thickness direction, see *NDTEMPER and *NDTEMPDIF
Body Force	<ul style="list-style-type: none">- Gravity loading or body forces due to linear acceleration in global X, Y and Z directions, see *BODYFORCE- Centrifugal loads due to angular velocity about the global X, Y and Z direction, see *BODYFORCE- Tangential loads due to angular acceleration about the global X, Y and Z directions, see *BODYFORCE
Ground Motion	<ul style="list-style-type: none">- Forces due to ground motion (support excitation) in the global X, Y and Z directions, see *GROUND- Forces due to rotational ground motion in the global XY, YZ and ZX planes about a specified point, see *GROUND

Notes:

1. The element may be oriented anywhere in space. The element connectivity must be given in the order shown in [Figure 4.35](#), in which the node numbering sequence starts at a corner node and proceeds along the perimeter of the element.
2. The bottom and top faces of the element are numbered as faces 1 and 2, respectively ([Figure 4.35\(b\)](#)). Top and bottom faces are determined as follows:
 - (a) Choose a viewpoint so that the element is defined counterclockwise.
 - (b) The top face is now closest to you.

Pressure loading (force/area) is integrated over the area of the shell middle surface.

3. This element possesses no rotational stiffness about the normal to the shell surface. To avoid any possible difficulties, set AUTO = ON in the executive commands. A small rotational stiffness is then added at the nodes where the shell surface has continuous curvature.
4. For orthotropic materials, the material principal axes should be defined in such a way that the material Z-axis is normal to the middle surface of the element. If the X and Y

material axes are not tangent to the middle surface, the local material axes are chosen such that the material Z-axis is normal to the shell and the revised system is closest to the user defined system. For geometrically complex structures, it may be advantageous to use laminated composite shell elements (NKTP = 32) even for the case of single layer shells, because of the ease with which material directions can be specified.

For the 4-node element (NORDR = 1), the 4 nodes need not lie in the same plane. That is, the element may be warped. However, excessive warping is not recommended. A warning message is printed whenever the warping index exceeds 0.2. This corresponds to a warping angle of about 11° . The warping index is defined as,

$$WI = \frac{2d}{\sqrt{A}}$$

where, d is the perpendicular distance between any node of the element and the mean plane, and A is the area of the mean plane, see [Figure 4.35\(c\)](#). The mean plane is the plane passing through the midpoints of the 4 edges of the element.

5. Stresses can be calculated in both local and global systems, at the centroid, Gauss and/or nodal points for the top, middle, and bottom surfaces. The direction cosines of the local directions, which are tangent to the shell midsurface, are also printed along with the local stress components. The three inplane stress components (SXX, SYY, SXY) vary linearly through the thickness, and reach their maximum values at either the top or bottom surfaces. The transverse shear stresses (SXZ, SYZ) are not computed.
6. For nonlinear static analysis, the nodal equivalent plastic strains (EPS) are approximated by extrapolating the Gauss point equivalent plastic strains.
7. The elasto-plastic model for NORDR = 10 in NISA is based on the Ilyushin yield condition (see [Section 2.8](#)).
8. Eight components of stress resultants are defined as,

$$N_{XX} = \int \sigma_{xx} dz, \quad N_{YY} = \int \sigma_{yy} dz, \quad N_{XY} = \int \sigma_{xy} dz,$$

$$M_{XX} = \int \sigma_{yy} z dz, \quad M_{YY} = \int \sigma_{xx} z dz, \quad M_{XY} = \int \sigma_{xy} z dz,$$

$$Q_{YZ} = \int \sigma_{yz} dz, \quad Q_{XZ} = \int \sigma_{xz} dz$$

These resultants are computed in the element local coordinate system specified in NRCS of *ELEMENT data group. The positive resultant moments MXX and MYY are chosen in the direction of positive rotation about X and Y axes. The averaged nodal stress resultants are the average of all local stress resultants at the node.

4.31 2-D or Axisymmetric Gap Element (NKTP = 42)

Description

This element is a 2-node line element which may be used to represent three different physical situations:

- (a) Gaps, i.e., an initial separation (or interference) between two parts of a 2-D model, which may be closed (or opened) during deformation.
- (b) 2-D tension-only members, such as cables.
- (c) 2-D compression-only members, such as a loose soil foundation.

The element may be used in a 2-D or axisymmetric model. It must lie in the global XY plane (right half of XY plane if used as axisymmetric gap). The element has two translational degrees of freedom per node (UX, UY).

The element configuration is shown in [Figure 4.36](#). An element reference guide is given in [Table 4.66](#). A general description of the element input data is given in [Section 4.1.2](#). The output pertinent to this element is given in [Table 4.66](#).

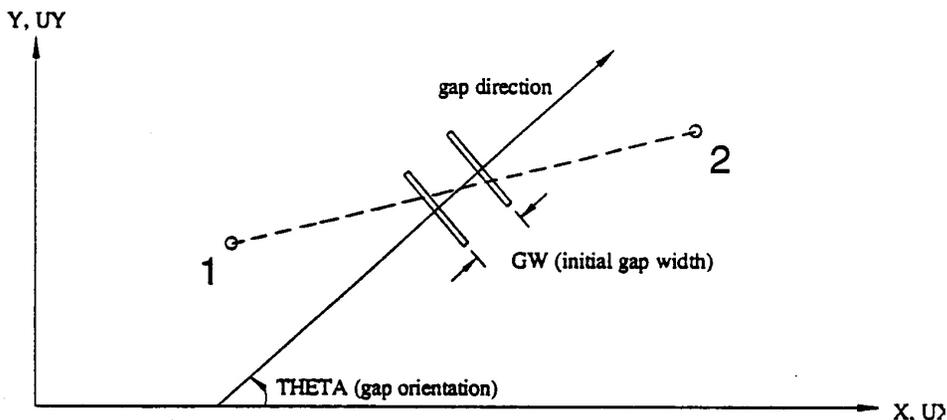


Figure 4.36: 2-D or axisymmetric gap element (NKTP = 42, NORDR = 1)

Table 4.66: Element reference guide (NKTP = 42)

Element Type	NKTP = 42, 2-D or Axisymmetric Gap element
Analysis Types	Static
Degrees of Freedom	2 per node: UX, UY
NORDR (Shape / No. of nodes)	- Line: 2 nodes (NORDR = 1)
Real Constants (RCTABLE)	6 constants: K, F, GW, GTOL, THETA, FL where,
Entry no.	
1	K Axial stiffness for rigid body motion, see note 1
2	F flexibility coefficient or flexibility coefficient per unit length, see FL and note 2 = 0 for gap elements > 0 global cartesian system < 0 local displacement coordinate system at first node, see note 2
3	GW initial gap width, see notes 3, 4
4	GTOL convergence tolerance on gap closure (default = 1.E-8).
5	THETA element orientation angle (in degrees) between the global X-axis and the element axis in counterclockwise direction, see note 4
6	FL type of flexibility coefficient given in F = 0 the value given in F is flexibility coefficient. = 1 the value given in F is flexibility coefficient per unit length.
Material Properties	None
Element Output	- Load factor: force transmitted in the gap element (upon closure), or load carried by tension-only or compression-only members - Gap width status: specified width (input), gap width prior to adjustment (i.e., for the initial solution in which the element is inactive), and final gap width

Notes:

1. A provision is made for solving problems where the structure is unstable (in the sense of having unrestrained rigid body translations or rotations) for an initial configuration in which all the gaps are open (and/or no load is carried by tension-only or compression-only members).

If the model has no unrestrained rigid body modes even with all the gaps open, then the initial spring stiffness (K, the first real constant) can be set to zero.

If rigid body modes are present, then enter a positive value for K. In this case, to calculate the deformation under initial loading, gap elements will be modeled as springs having stiffness K. Hence, the chosen value for K should be small compared to the characteristic stiffness of other elements in the model, but not so small as to cause ill-conditioning of the stiffness matrix.

2. The member flexibility coefficient (F, the second real constant) should be set to zero when the element is being used as a gap element.

When the element is being used as a tension-only member, then F should be a positive number equal to the flexibility (or flexibility per unit length) of the member when it is acting in tension. Considering the tension-only member as a spring, its flexibility is the reciprocal of the spring constant. Considering the member as a rod or spar, its flexibility is $L/(AE)$, where L, A, and E are length, cross-sectional area, and Young's Modulus, respectively.

For elements that have the same values of A and E but different element length, it is recommended to define F in flexibility per unit length ($FL = 1$).

Otherwise each element with different length requires a separate real constant table.

The above comments for tension-only members pertain to compression-only members as well, except that the flexibility coefficient is entered as a negative number.

3. When the element is being used as a gap, the initial gap width (GW --the third real constant) must be defined. The gap width is *not* inferred from the difference in nodal coordinates. In fact, the two nodes of the gap may be coincident. The initial width of the gap is measured in the direction specified by THETA (the fifth real constant). If GW is positive, the gap is initially open. If GW is negative, there is initial interference across the gap.

When the element is being used as a tension-only or compression-only member, then the input of GW is optional. If GW is entered as zero there is no initial deformation in the member. If GW is positive, it is interpreted as the initial deformation of a tension-

only member. If GW is negative, it is the initial deformation of a compression-only member. Initial deformations are in the axial direction of the element, which is defined by THETA.

- Neither the initial width of the gap nor the orientation of the element is inferred from the location of the two nodes defining these 2-D gap elements. The initial gap width is given as GW, the third real constant. The orientation of the element is given by THETA, the fifth real constant. THETA is a rotation about the global Z-axis, in degrees, counterclockwise positive, measured from the global X-axis to the element axis. Zero degrees is aligned with the global X-axis. (Note that, by contrast, the orientation of 3D gap elements (NKTP = 43) is inferred from the node locations.)
- The 2-D gap element can be used along with the following element types: plane stress (NKTP = 1), plane strain (NKTP = 2), axisymmetric (NKTP = 3), 2-D beam (NKTP = 13), 2-D spars (NKTP = 16), 2-D translational and torsional springs (NKTP = 18, 22), 2D point masses (NKTP = 25, 27 and 29), and the axisymmetric shell (NKTP = 36).

As is always the case for 2-D models, these elements must exist in the global XY plane. When used with axisymmetric (NKTP = 3) elements, the entire model must be in the right half of the global XY plane.

(Use element type NKTP = 43 if your model contains 3-D elements.)

- Problems including gaps are solved iteratively. The first solution is of the entire model with the actual static loads applied. All gaps are assumed to be open for this initial solution. If any non-zero spring constants (see note 1) are defined, they are incorporated into the stiffness of this solution. Additional load cases (equal to the number of gap elements) are then solved internally. Unit loads applied along the axis of the gap elements are used to generate flexibility coefficients for the structure. This matrix of flexibility coefficients has a rank equal to the number of gap elements in the model. This matrix is iterated on to find the values of gap displacement and axial force which solve the problem, i.e., which provide no interference at any of the gap elements. Interference is checked for only in the direction along the axis of the gap element. There is no restriction to motion normal to the gap axis.
- Gap (compression-only or tension-only) elements are not recommended to be edges or faces of finite elements with midside nodes. Use only the first order elements (NORDR = 1, 10) in regions connected to the gap elements.
- The degrees of freedom (UX, UY) at the two nodes of the element must not be dependent degrees of freedom in any MPC equation or rigid link. In addition, displacement boundary conditions, if specified, must not be such that the relative displacement in the element axis direction is specified, i.e., the gap status must not be predetermined.

4.32 3D Gap Element (NKTP = 43)

Description

This element is a 2-node 3-D line element which may be used to represent three different physical situations:

- (a) Gaps, i.e., an initial separation (or interference) between two parts of a 3-D model, which may be closed (or opened) during deformation.
- (b) 3-D tension-only members, such as cables.
- (c) 3-D compression-only members, such as a loose soil foundation.

The element may be oriented anywhere in space and it has three translational degrees of freedom per node (UX, UY, UZ). The two nodes of the element must not be coincident. The element configuration is shown in [Figure 4.37](#). An element reference guide is given in [Table 4.67](#). A general description of the element input data is given in [Section 4.1.2](#). The output pertinent to this element is given in [Table 4.67](#).

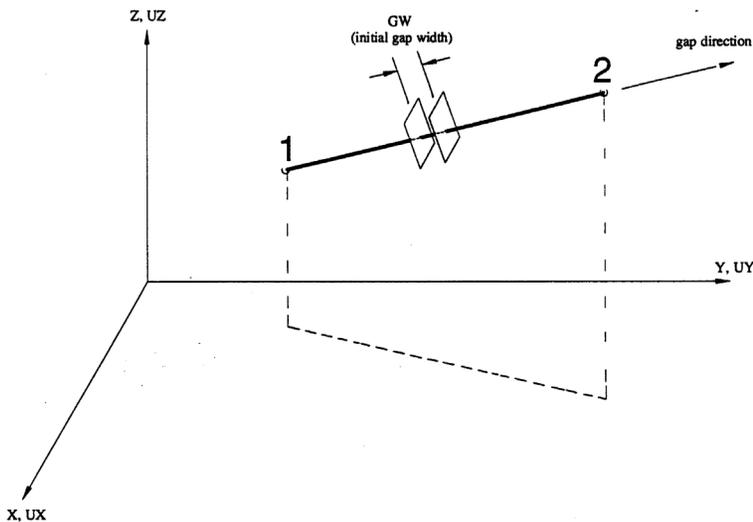


Figure 4.37: 3-D gap element (NKTP = 43, NORDR = 1)

Table 4.67: Element reference guide (NKTP = 43)

Element Type	NKTP = 43, 3-D Gap element
Analysis Types	Static
Degrees of Freedom	3 per node: UX, UY, UZ
NORDR (Shape / No. of nodes)	- Line: 2 nodes (NORDR = 1)
Real Constants (RCTABLE)	5 constants: K, F, GW, GTOL, FL where,
Entry No.	
1	K Axial stiffness for rigid body motion, see note 1
2	F flexibility coefficient or flexibility coefficient per unit length, see FL and note 2 = 0 for gap elements > 0 flexibility coefficient of a tension-only member < 0 flexibility coefficient of a compression-only member
3	GW initial gap width, see notes 3, 4
4	GTOL convergence tolerance on gap closure (default = 1.E-8).
5	FL type of flexibility coefficient given in F = 0 the value given in F is flexibility coefficient. = 1 the value given in F is flexibility coefficient per unit length.
Material Properties	None
Element Output	- Load factor: force transmitted in the gap element (upon closure), or load carried by tension-only or compression-only members - Gap width status: specified width (input), gap width prior to adjustment (i.e., for the initial solution in which the element is inactive), and final gap width

Notes:

1. A provision is made for solving problems where the structure is unstable (in the sense of having unrestrained rigid body translations or rotations) for an initial configuration in which all the gaps are open (and/or no load is carried by tension-only or compression-only members).

If the model has no unrestrained rigid body modes even with all the gaps open, then the initial spring stiffness (K, the first real constant) can be set to zero.

If rigid body modes may be present, then enter a positive value for K. In this case, to calculate the deformation under initial loading, gap elements will be modeled as springs having stiffness K. Hence, the chosen value for K should be small compared to the characteristic stiffness of other elements in the model, but not so small as to cause ill-conditioning of the stiffness matrix.

2. The member flexibility coefficient (F, the second real constant) should be set to zero when the element is being used as a gap element.

When the element is being used as a tension-only member, then F should be a positive number equal to the flexibility (or flexibility per unit length) of the member when it is acting in tension. Considering the tension-only member as a spring, its flexibility is the reciprocal of the spring constant. Considering the member as a rod or spar, its flexibility is $L/(AE)$, where L, A, and E are length, cross-sectional area, and Young's Modulus, respectively.

For elements that have the same values of A and E but different element length, it is recommended to define F in flexibility per unit length ($FL = 1$). Otherwise each element with different length requires a separate real constant table.

The above comments for tension-only members pertain to compression-only members as well, except that the flexibility coefficient is entered as a negative number.

3. When the element is being used as a gap, the initial gap width (GW, the third real constant) must be defined. The gap width is *not* inferred from the difference in nodal coordinates.

If GW is positive, the gap is initially open. If GW is negative, there is initial interference across the gap.

When the element is being used as a tension-only or compression-only member, then the input of GW is optional. If GW is entered as zero there is no initial deformation in the member. If GW is positive, it is interpreted as the initial deformation of a tension-only member. If GW is negative, it is the initial deformation of a compression-only

member. Initial deformations are in the axial direction of the element, which is defined by THETA.

4. The initial width of the gap is not inferred from the nodal locations. But the orientation of the element is defined by the nodal locations, such that the element axis is directed from node 1 to node 2. Therefore, the nodes must not be coincident (note that by contrast, the orientation of the 2-D gap element (NKTP = 42) is not inferred from the node locations).
5. Problems including gaps are solved iteratively. The first solution is of the entire model with the actual static loads applied. All gaps are assumed to be open for this initial solution. If any non-zero spring constants, see note 1, are defined, they are incorporated into the stiffness of this solution. Additional load cases (equal to the number of gap elements) are then solved internally. Unit loads applied along the axis of the gap elements are used to generate flexibility coefficients for the structure. This matrix of flexibility coefficients has a rank equal to the number of gap elements in the model. This matrix is iterated on to find the values of gap displacement and axial force which solve the problem, i.e., which provide no interference at any of the gap elements. Interference is checked for only in the direction along the axis of the gap elements. There is no restriction to motion normal to the gap axis.
6. Gap (compression-only or tension-only) elements are not recommended to be used at edges or faces of finite elements with midside nodes. Use only the first order elements (NORDR = 1, 10) in regions connected to the gap elements.
7. The degrees of freedom (UX, UY, UZ) at the two nodes of the element must not be dependent degrees of freedom in any MPC equation or rigid link. In addition, displacement boundary conditions, if specified, must not be such that the relative displacement in the element axis direction is specified, i.e., the gap status must not be predetermined.

4.33 3-D Cable Element (NKT = 45)

Description

This element is a 2-node uniaxial tension-only 3-D line element and is used to model space cables. The element may be oriented anywhere in space. The element has three translational degrees of freedom per node (UX, UY, UZ). The theoretical basis of the element is similar to that of 3-D spar element (NKTP = 14) except that this element cannot take compressive loads.

The element configuration is shown in Figure 4.38. An element reference guide is given in Table 4.68 with the available loading listed in Table 4.69. A general description of the element input data and output options are given in Section 4.1.2 and Section 4.1.3 respectively. Table 4.68 gives the output pertinent to this element.

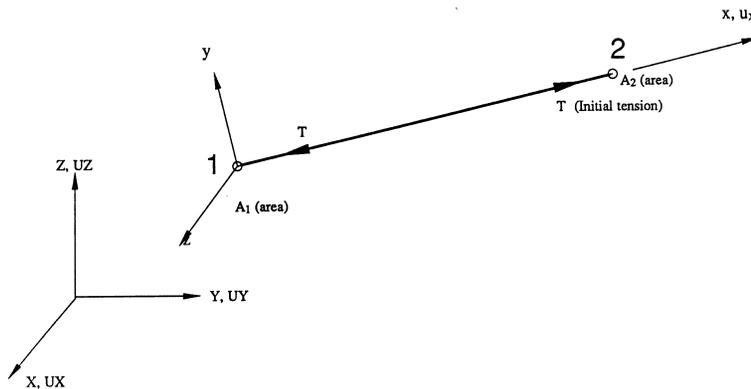


Figure 4.38: Element configuration (NKTP = 45, NORDR = 1)

Table 4.68: Element reference guide (NKTP = 45)

Element Type	NKTP = 45,3-D Cable element
Analysis Types	Nonlinear Static, Nonlinear Transient
Degrees of Freedom	3 per node: UX, UY, UZ
NORDR (Shape / No. of nodes)	- Line: 2 nodes (NORDR = 1)

<p>Real Constants (RCTABLE)</p>	<p>5 constants: A1, A2, TENSIN, TKEY, C where,</p> <p>A1, A2 : Cross sectional area at node 1 and 2</p> <p>TENSIN : Initial tension in the element</p> <p>TKEY : Option for artificial stiffness when cable under compression</p> <p>= 0 (default) using $C = 1.0 \times 10^{-6}$</p> <p>= 1 C is given by user</p> <p>C : Artificial stiffness for cable under compression</p>
<p>Material Properties</p> <p>Isotropic Elastic</p> <p>Elastoplastic</p>	<p>3 properties: EX, DENS, ALPX</p> <p>For nonlinear static only, see *PLASTIC</p>
<p>Element Output</p>	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ) at the two ends in the global coordinate system (or in local displacement system if defined), and strain energy - Axial force (FX) and axial stress in element local coordinate system - Element stresses at nodal points in element local coordinate system - Equivalent plastic strain (EPS), and effective yield stress, for nonlinear static only, at nodal points - Second Piola-Kirchhoff or Cauchy stresses, for nonlinear static only, at nodal points
<p>Nonlinear Capabilities</p>	<ul style="list-style-type: none"> - Geometric nonlinearity: Large displacements, rotations and strains; total and updated Lagrangian Formulation; deformation dependent loads - Material nonlinearity: elastoplastic material model

Table 4.69: Available loading (NKTP = 45)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X, Y, Z directions (or in local displacement system, if defined), see *CFORCE - Specified non-zero nodal displacements UX, UY, UZ in global X, Y and Z directions (or in local displacement system, if defined), see *SPDISP - Concentrated nodal follower (deformation dependent) forces perpendicular to the axis of the element, for nonlinear static only. See *FOLLOWER and note 3.
Thermal Loading	<ul style="list-style-type: none"> - Specified nodal temperature, see *NDTEMPER
Body Force	<ul style="list-style-type: none"> - Gravity loading or body forces due to linear acceleration in global X, Y,Z directions, see *BODYFORCE - Centrifugal loads due to angular velocity about the global X, Y and Z directions, see *BODYFORCE - Tangential loads due to angular acceleration about the global X, Y and Z axes, see *BODYFORCE

Notes:

1. This element has three real constants which must be entered in the *RCTABLE data group. They are the cross-sectional areas at the first and second nodes of the element, respectively, and the initial tension in the cable element.
2. If there is no initial tension in the physical model, it is recommended that a small value of initial tension be supplied in the finite element model to enhance convergence.
3. To calculate the direction of the concentrated follower force, the following approximation is assumed:
 - The third point required in *CFOLLOWER will define the local y- and z-axes in the initial undeformed configuration such that nodes 1, 2 and the third point will be in the xy plane.
 - In any subsequent deformed configuration, the local z-axis will be defined as the cross product of two unit vectors aligned with the current and the previous x-axis, and the local y-axis is the cross product of two unit vectors aligned with the local z- and x-axes.

4.34 3-D Straight Pipe Element (NKTP = 46)

Description

This element is a 2-node straight pipe element with circular cross section. The formulation includes stretching, bending and torsion effect. The transverse shear deformation effect is included as an option. The element axial stiffness can be overrided by user's input (see note 4). The flexibility factor of the element and the stress intensity factors at nodes can be specified in the *RCTABLE (see note 2). In addition, the mass effect of the internal fluid and the insulation material can be included (see note 3). The deformation is characterized by three translations (UX, UY, UZ) and three rotations (ROTX, ROTY, ROTZ). The local x-axis of the pipe is along the centroidal axis. The local y and z axes are defined by the position of the third point, see note 1. The theoretical basis of the element is discussed in [Section 2.8](#).

The element configuration, node location, local coordinate system and face numbering convention for line pressure loading are shown in [Figure 4.39](#). An element reference guide is briefed in [Table 4.70](#) with the available loading listed in [Table 4.71](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.70](#) gives the output pertinent to this element. The required real constants for this element are given in [Table 4.72](#). For a given problem, not all of the entries in the real constant table are needed. Entries not specified will be set to default values.

Element Library

3-D Straight Pipe Element (NKTP = 46)

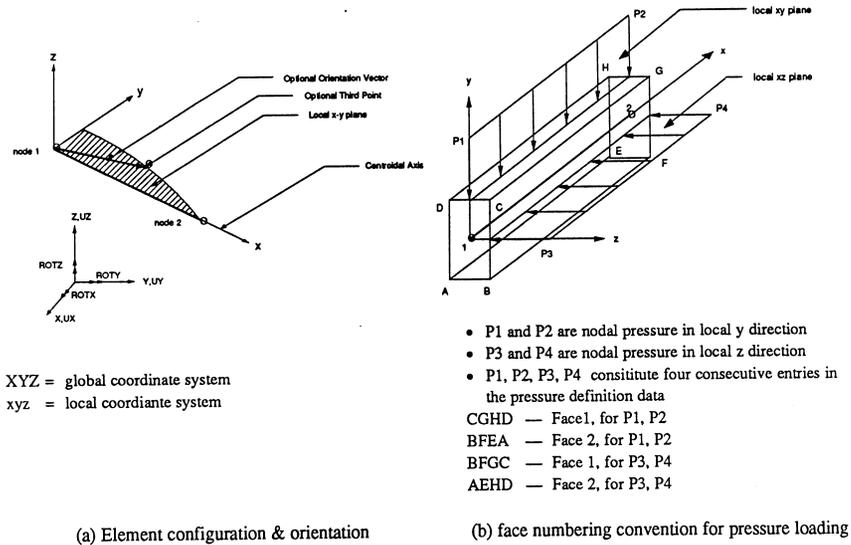


Figure 4.39: 3-D Straight Pipe Element (NKTP = 46, NORDR = 1)

Table 4.70: Element reference guide (NKTP = 46)

Element Type	NKTP = 46, 3-D Straight pipe element
Analysis Types	Static, Dynamic
Degrees of Freedom	6 per node: UX, UY, UZ, ROTX, ROTY, ROTZ
NORDR (Shape / No. of nodes)	- Line: 2 nodes (NORDR = 1)
Real Constants	Up to 10 constants, see Table 4.72
Material Properties Isotropic Elastic	4 properties: EX, NUXY, DENS, ALPX

Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ, MX, MY, MZ) at pipe ends in global coordinate system (or in local displacement system, if defined), and strain energy - Resultant forces and moments at the two ends in the element local coordinate system (FX, FY, FZ, MX, MY, MZ) - Element local stresses (normal stress, hoop stress, shear stress), principal stresses (S1, S2), see 1
Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration, and shock spectrum analyses

Table 4.71: Available loading (NKTP = 46)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X, Y and Z directions and concentrated nodal moments about global X, Y and Z axes (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal displacements UX, UY, UZ and nodal rotations ROTX, ROTY, ROTZ in global directions (or in local displacement system, if defined), see *SPDISP - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
Internal Pressure and Transverse Pressure	<ul style="list-style-type: none"> - Uniform or non-uniform transverse line pressure (force/length) see *PRESSURE, *DPRESSURE and note 7 - Uniform internal pressure; see *PRESSURE and note 8
Thermal Loading	<ul style="list-style-type: none"> - Specified nodal temperature and nodal temperature difference in pip wall see *NDTEMPER, *NDTEMPDIF, and note 9
Body Force	<ul style="list-style-type: none"> - Gravity loading or body forces due to linear acceleration in global X, Y and Z axes, see *BODYFORCE - Centrifugal loads due to angular velocity about the global X, Y and Z axes, *BODYFORCE - Tangential loads due to angular acceleration about the global X, Y, and Z axes, see *BODYFORCE

Element Library

3-D Straight Pipe Element (NKTP = 46)

Ground Motion	<ul style="list-style-type: none">- Forces due to ground motion (support excitation) in the global X, Y and Z directions, see *GROUND- Forces due to rotational ground motion in the global XY, YZ and ZX planes about a specified point, see *GROUND
----------------------	--

Table 4.72: List of real constants for 3-D beam (NKTP = 46)

Entry	Variable	Description
1	D	outer diameter
2	T	wall thickness
3	ST	Flexial stiffness (see note 4)
4	FLX	flexibility factor (see note 2)
5	SF1	stress intensity factor at node 1 (see note 2)
6	SF2	stress intensity factor at node 2 (see note 2)
7	FLDENS	internal fluid density per unit volume ($force \cdot time^2 / length^4$) (see note 3)
8	AMS	additional mass per unit length ($force \cdot time^2 / length^2$) (see note 3)
9	KSH	key for transverse shear coefficient (see note 5)
10	ALPHA	transverse shear coefficient

Notes:

1. The element local coordinate system is defined by the two nodal points and a third node. The third point is defined in *ELEMENTS data group either explicitly as a third node or implicitly using an orientation vector, see [Figure 4.39\(a\)](#). The local x-axis is along the line joining node 1 and node 2. The local xy plane is defined by node 1, node 2 and the third point. The local z-axis is normal to the xy plane, see [Figure 4.39\(b\)](#).
2. The flexibility factor (FLX) is used to adjust the bending stiffness of the pipe element for various pipe analysis. The default value of FLX is 1. The stress intensity factor (SF1 and SF2) is used to modify the bending stress predicted by the beam theory. The bending stress will multiplied by the stress intensity factor before being added to the axial stress. The value of SF1 and SF2 can be specified in *RCTABLE data group. These two factors will default to 1 if the given value is less than 1.

3. In addition to the pipe mass, two types of material density can be specified in *RCTABLE data group to model mass effect other than the pipe mass. The mass effect of internal fluid can be included by specifying the internal fluid density (FLDENS) in the 7th entry in the *RCTABLE. The total mass of the internal material will be the internal volume times FLDENS. The additional mass density AMS can be used to model any distributed mass such as insulation material.
4. Any positive non-zero value of STF will override the computed element axial stiffness in the element stiffness matrix.
5. The transverse shear coefficient (ALPHA) accounts for the non-uniformity of the shear strain over the cross-section. It defines an effective area for shear as the actual cross-sectional area divided by ALPHA. The parameter KSH is the key to control the definition of the transverse shear coefficient. If KSH = 0, the input value in ALPHA will be ignored and the correction factor will default to 1.887 (the value for circular tube cross section). For KSH = 1, the value specified in ALPHA will be used as the transverse shear coefficient.
6. The stress results in the pipe local coordinate system (normal force, shear forces, bending moments and twisting moment) are computed at the pipe ends. In addition, stresses are computed at four locations where maximum bending stress and maximum shear stresses due to bending occur, see [Figure 4.40](#) and [Figure 4.41](#)
7. Line pressure loading can be applied for this element along the local y and/or z axes using *PRESSURE data group. Four imaginary faces, see [Figure 4.39\(b\)](#), are defined to direct the line loads. Positive pressure loading is applied toward to the face. P1, P2, P3 and P4 constitute four consecutive entries in *PRESSURE data group for pressure applied in local y (P1, P2) and local z (P3, P4) directions, see [Figure 4.39\(b\)](#). If the pressure is applied in only one direction, the appropriate entries must be set to zero.

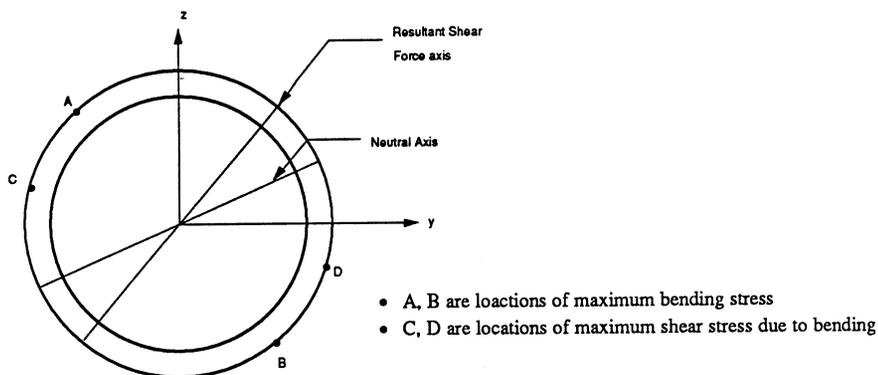


Figure 4.40: Element Stress Output Locations

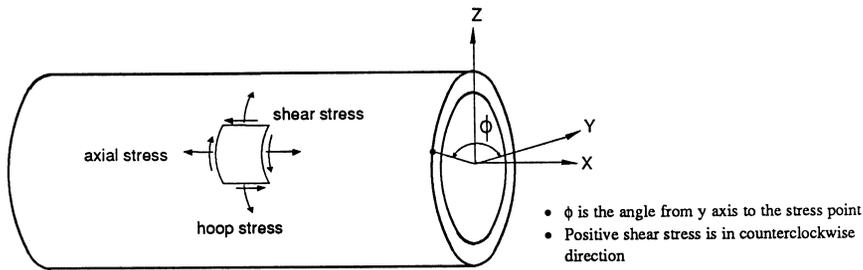


Figure 4.41: Element Stress Definition

8. The element is assumed as a closed ends pipe. Axial load will be produced by the internal pressure (force/area). The internal pressure is assumed constant in the element. Only one entry is needed to specify the value of the internal pressure in the *PRESSURE data group and the internal pressure is identified by a face number which is greater than 2.
9. The temperature given in *NDTEMPER data group are used to compute the thermal load due to axial thermal expansion and contraction. The temperature difference at nodes given in *NDTEMPDIF data group specifies the temperature difference between the outer and inner pipe wall. The data given in *NDTEMPDIF is only used for stress recovery to include thermal stresses due to temperature difference through the pipe wall.

4.35 3-D Elbow Element (NKTP = 47)

Description

This element is a 2-node 3-D single curve elbow with constant curvature and circular cross section. The subtended angle of the elbow must be less than 180° . The elbow is assumed to have closed ends. The formulation includes stretching, bending and torsion effect. The transverse shear deformation effect is included as an option. The flexibility factor of the element and the stress intensity factors at the nodes can be specified in the *RCTABLE (see note 2 and 3). In addition, the mass effect of the internal fluid and the insulation material can be included (see note 5). The deformation is characterized by three translations (UX, UY, UZ) and three rotations (ROTX, ROTY, ROTZ). At any point along the centroidal circular arc, the local x-axis of the element is tangent to the centroidal arc and the local y-axis is a vector directed from the center of curvature to the centroid of the cross section. The local z-axis is normal to the local xy plane, see note 1. The theoretical basis of the element is discussed in [Section 2.8](#).

The element configuration, node location, local coordinate system and face numbering convention for line pressure loading are shown in [Figure 4.42](#). An element reference guide is briefed in [Table 4.73](#) with the available loading listed in [Table 4.74](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.73](#) gives the output pertinent to this element. The required real constants for this element are given in [Table 4.75](#). For a given problem, not all of the entries in the real constant table are needed. Entries not specified will be set to default values.

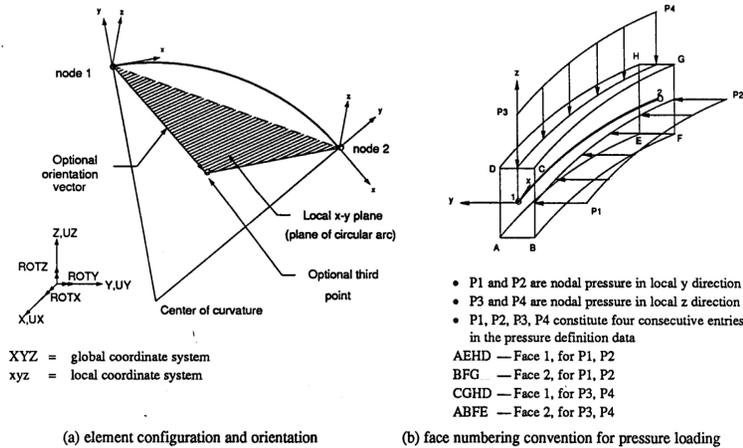


Figure 4.42: 3-D Elbow Element (NKTP = 47, NORDR = 1)

Table 4.73: Element reference guide (NKTP = 47)

Element Type	NKTP = 47, 3-D Elbow element
Analysis Types	Static, Dynamic
Degrees of Freedom	6 per node: UX, UY, UZ, ROTX, ROTY, ROTZ
NORDR (Shape / No. of nodes)	- Circular arc: 2 nodes (NORDR = 1)
Real Constants	Up to 10 constants, see Table 4.35.3
Material Properties	
Isotropic Elastic	4 properties: EX, NUXY, DENS, ALPX
Element Output	<ul style="list-style-type: none"> - Internal forces (FX, FY, FZ, MX, MY, MZ) at elbow ends in global coordinate system (or in local displacement system, if defined), and strain energy - Resultant forces and moments at elbow ends in the element local coordinate system (FX, FY, FZ, MX, MY, MZ) - Element local stresses (normal stress, hoop stress, shear stress), principal stresses (S1, S2), see Figure 4.44c

Dynamic Capabilities	<ul style="list-style-type: none"> - Consistent or lumped mass - Eigenvalue, transient dynamic, direct frequency response, frequency response, random vibration, and shock spectrum analyses
-----------------------------	--

Table 4.74: Available loading (NKTP = 47)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal forces in global X, Y and Z directions and concentrated nodal moments about global X, Y and Z axes (or in local displacement system, if defined), see *CFORCE or *DCFORCE - Specified non-zero nodal displacements UX, UY, UZ and nodal rotations ROTX, ROTY, ROTZ in global directions (or in local displacement system, if defined), see *SPDISP - Concentrated unit harmonic loading for point-to-point transfer function, see *DRIVER
Internal Pressure and Transverse Pressure	<ul style="list-style-type: none"> - Uniform or non-uniform transverse line pressure (force/length); see *PRESSURE, *DPRESSURE and note 7 - Uniform internal pressure (force/area); see *PRESSURE and note 8
Thermal Loading	<ul style="list-style-type: none"> - Specified nodal temperature and nodal temperature difference in wall thickness; see *NDTEMPER, *NDTEMPDIF, and note 9
Body Force	<ul style="list-style-type: none"> - Gravity loading or body forces due to linear acceleration in global X, Y and Z axes, see *BODYFORCE - Centrifugal loads due to angular velocity about the global X, Y and Z axes, see *BODYFORCE - Tangential loads due to angular acceleration about the global X, Y, and Z axes, see *BODYFORCE
Ground Motion	<ul style="list-style-type: none"> - Forces due to ground motion (support excitation) in the global X, Y and Z directions, see *GROUND - Forces due to rotational ground motion in the global XY, YZ and ZX planes about a specified point, see *GROUND

Table 4.75: List of real constants for 3-D beam (NKTP = 47)

Entry	Variable	Description
1	D	outer diameter
2	T	wall thickness
3	R	radius of curvature (distance between center of curvature and the centroidal arc of the element)
4	FLX	flexibility factor (see note 2)
5	SF1	stress intensity factor at node 1 (see note 2)
6	SF2	stress intensity factor at node 2 (see note 2)
7	FLDENS	internal fluid density per unit volume ($force \cdot time^2 / length^4$) (see note 3)
8	AMS	additional mass per unit length ($force \cdot time^2 / length^2$) (see note 3)
9	KSH	key for transverse shear coefficient (see note 5)
10	ALPHA	transverse shear coefficient

Notes:

1. The subtended angle of the circular arc should be less than 180°. The element local x-axis is tangent to the centroidal arc. The orientation of the element (local xy plane) is defined by the two node points and a third point. The third point is defined in the *ELEMENTS data group either explicitly as a third node or implicitly using an orientation vector, see [Figure 4.42](#). The third point must lie on the center of curvature side of the line connecting node 1 and node 2.
2. The flexibility factor (FLX) is used to correct the bending stiffness of the elbow from that predicted by beam theory. The value of the flexibility factor can be specified by the user (by specifying a positive nonzero number for FLX) or calculated internally by the program using formulae given in [4.7] and [4.8]. Six options are available:
 - (a) FLX = 0 (Default)

flexibility factor = 1.0

- (b) FLX = -1 (ASME flexibility factor with pressure term)

$$\text{flexibility factor} = 1.65/h[1 + (6P/Eh)(R/t)^{1/3}]$$

- (c) FLX = -2 (ASME flexibility factor without pressure term)

$$\text{flexibility factor} = 1.65/h$$

- (d) FLX = -3 (Empirical flexibility factor with pressure term)

$$\text{flexibility factor} = 1.66/\lambda[1 + 1.75\lambda^{-4/3}\exp(-1.15\Psi^{-1/4})]$$

- (e) FLX = -4 (Empirical flexibility factor without pressure term)

$$\text{flexibility factor} = 1.66/\lambda$$

$$h = \frac{tR}{r^2}$$

$$\lambda = \frac{1}{h\sqrt{1-\nu^2}}$$

$$\Psi = \frac{PR}{Ert}$$

$$\text{flexibility factor} = 1.66/\lambda$$

- (f) FLX = -5 (KARMAN flexibility factor)

$$\text{flexibility factor} = (10 + 12h^2)/(1 + 12h^2)$$

where,

t = the wall thickness

r = the mean radius of the pipe cross-section

R = radius of curvature

P = internal pressure

E = Young's modulus

ν = Poisson's ratio

3. The stress intensity factor is used to account for the difference in bending stress between the elbow and the beam theory. The bending stress obtained by beam theory is multiplied by the stress intensity factor before being added to the axial stress. The value of the stress intensity factor should be greater than 1.0. Any value less than 1.0 defaults to 1.0 except -1.0. If the SIF1 or SIF2 is specified as -1.0, the stress intensity factor at that node is calculated by the formula $0.9(r^2/Rt)^{2/3}$ (ASME code, from [4.7]).
4. The mass effect of the internal fluid can be included by specifying the fluid density (FLDENS) in the 7th entry in the *RCTABLE. The total mass of the fluid will be the internal volume times FLDENS. The additional mass density AMS can be used to model any distributed mass such as insulation material.
5. The transverse shear coefficient (ALPHA) accounts for the non-uniformity of the shear strain over the cross-section. It defines an effective area for shear as the actual cross-sectional area divided by ALPHA. The parameter KSH is the key to control the definition of the transverse shear coefficient. If KSH = 0, the input value in ALPHA will be ignored and the correction factor will default to 1.887 (the value for circular tube cross section). For KSH = 1, the value specified in ALPHA will be used as the transverse shear coefficient.
6. The stress resultants in the elbow local coordinate system (normal force, shear forces, bending moments and twisting moment) are computed at the elbow ends. In addition, stresses are computed at four locations where maximum bending stress and maximum shear stresses due to bending occur, see Figure 4.43 and Figure 4.44.

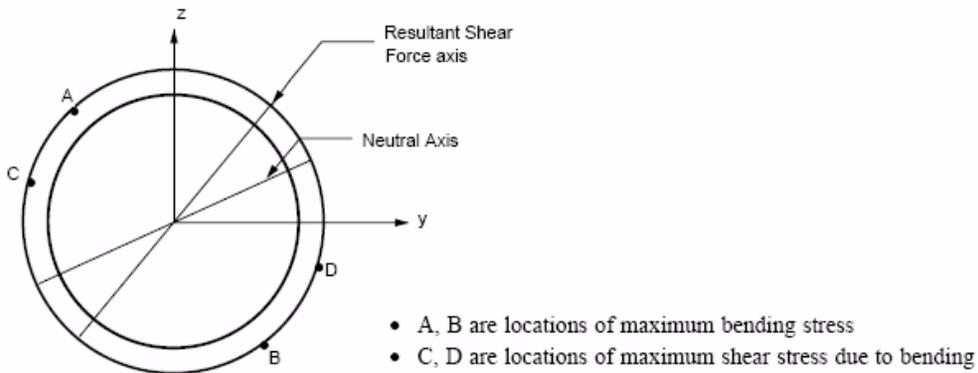


Figure 4.43: Element Stress Output Locations

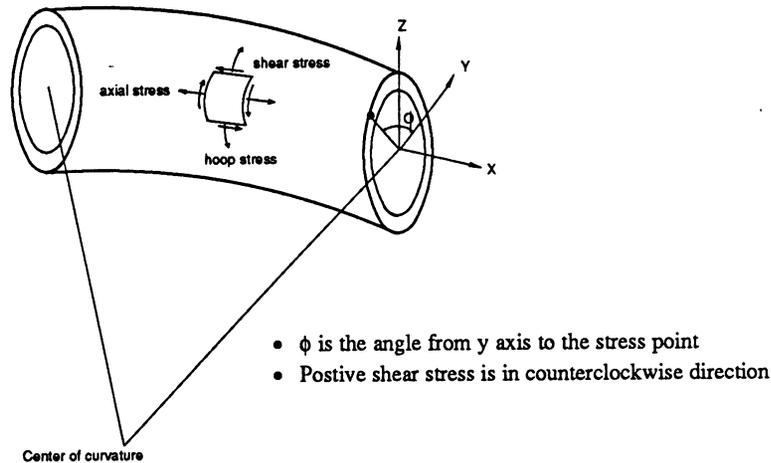


Figure 4.44: Element Stress Definition

7. Line pressure loading can be applied for this element along the local y and/or local z axes using *PRESSURE data group. Four imaginary faces, see [Figure 4.42\(b\)](#), are defined to direct the line loads. Positive pressure loading is applied toward to the face. P1, P2, P3 and P4 constitute four consecutive entries in *PRESSURE data group for pressure applied in local y (P1, P2) and local z (P3, P4) direction, see [Figure 4.42\(b\)](#). If the pressure is applied in only one direction, the appropriate entries must be set to zero.
8. The element is assumed as a closed ends elbow. Axial load will be produced by non-zero internal pressure value. The internal pressure should be in force/area. The internal pressure effect may be included or excluded from the flexibility factor (see note 2). The pressure is assumed constant in the element. Only one entry is needed to specify the value of the internal pressure in *PRESSURE data group and the internal pressure is identified by a face number which is greater than 2.
9. The temperature given in *NDTEMPER data group are used to compute the thermal load due to axial thermal expansion and contraction. The nodal temperature difference given in *NDTEMPDIF data group specifies the temperature difference between the outer and inner pipe wall. The data given in *NDTEMPDIF is only used for stress recovery to include thermal stresses due to temperature difference through the pipe wall.

4.36 3-D Translational Damper Element (NKTP = 48)

Description

This element is a 2-node uniaxial massless damper in three dimensions. The element may be oriented anywhere in space. The element has three translational degrees of freedom per node (UX, UY, UZ). The two nodes defining the damper may be coincident, in which case a direction vector is specified indicating the direction of the damper axis.

An element configuration is shown in [Figure 4.45](#). An element reference guide is given in [Table 4.76](#) with the available loading given in [Table 4.77](#). A general description of the element input data is given in [Section 4.1.2](#).

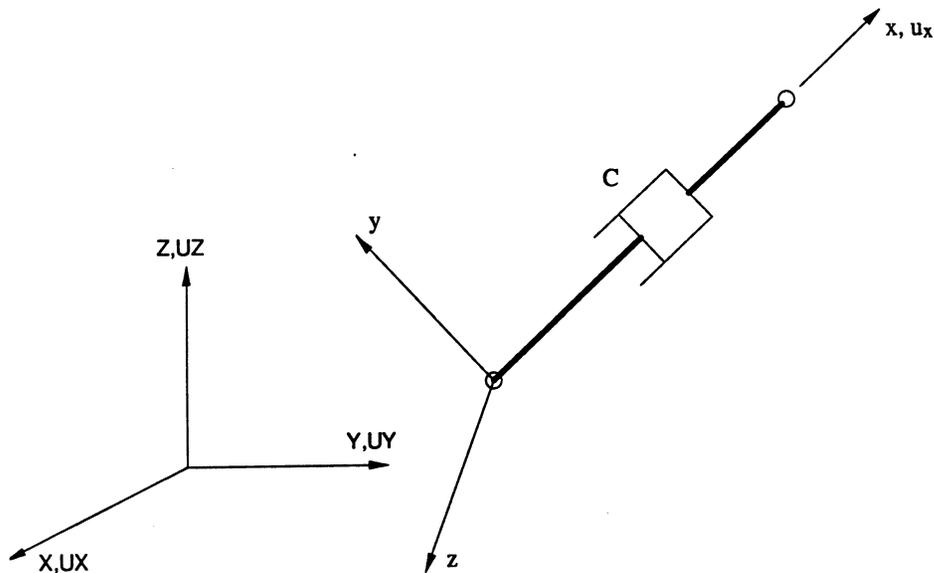


Figure 4.45: Element configuration (NKTP = 48, NORDR = 1)

Table 4.76: Element reference guide (NKTP = 48)

Element Type	NKTP = 48, 3-D Translational damper element
Analysis Types	Linear direct transient dynamics, direct frequency response, Nonlinear Transient, Direct frequency analysis see note 1
Degrees of Freedom	3 per node: UX, UY, UZ
NORDR (Shape / No. of nodes)	- Line: 2 nodes (NORDR = 1)
Real Constants (RCTABLE)	Up to 5 constants: C, V1, V2, V3, ISYS, IDSP where, C : Axial damping constant (force-time/length) V _i : i = 1, 2, 3; required only if the two nodes are coincident; components of a vector pointing in the damper axis direction ISYS: Coordinate system in which the components V _i , i = 1, 2, 3, are given = 0 global cartesian system = 1 local displacement coordinate system at first node, see note 2 = 2 local displacement coordinate system at second node, see note 2 IDSP: Frequency function ID of the frequency dependent damping constant for direct frequency analysis
Material Properties	None (damping coefficient is treated as a real constant)
Element Output	None

Table 4.77: Available loading (NKTP = 48)

Nodal Loading	- Concentrated nodal forces in global X, Y and Z directions (or in local displacement system, if defined), see *CFORCE
----------------------	--

Element Library

3-D Translational Damper Element (NKTP = 48)

Notes:

1. The damper element is a massless element. It may be used in direct transient dynamic and direct frequency response analysesdynamic analyses, provided that it is connected to other elements with non-zero masses.
2. If the two nodes of the element are not coincident, only one real constant, the axial damping constant, is required. If the two nodes are coincident, a direction vector (V_i , $i = 1, 2, 3$) is required. The direction vector is described in the coordinate system indicated by ISYS. If ISYS = 1 or 2, it is assumed that the local displacement coordinate system is defined at the referenced node (node 1 or node 2), otherwise ISYS is reset to zero.

4.37 2-D Gap/Friction Element (NKTP = 49)

Description

This element is a 2-node interface element, which may be used in 2-D, or axisymmetric problems to model contact at discrete points between two bodies. The element has two translational degrees of freedom (UX,UY) at each node. The element is nonlinear, it may have an open or closed status. If closed, the element resists normal compressive force and tangential shear force (Coulomb friction, if friction effect is activated). The theoretical basis of the element is described in [Section 2.8.6](#).

The element configuration is shown in [Figure 4.46](#). As an option, the element orientation and the initial gap width may be inferred from the location of the two nodes of the element. An element reference guide is given in [Table 4.78](#). A general description of element input data is given in [Section 4.1.2](#). The output pertinent to the element is given in [Table 4.78](#).

The element is represented by a pair of coupled nonlinear orthogonal springs in the normal and tangent directions to the interface. The load-displacement curves are shown in [Figure 4.47](#).

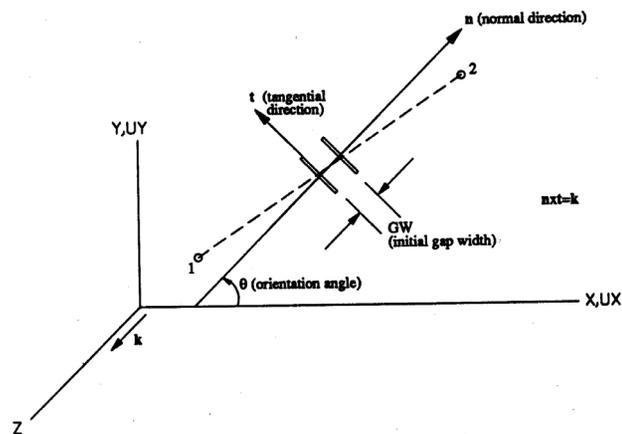


Figure 4.46: 2-D gap/friction element (NKTP=49, NORDR=1),
n = normal direction, **t** = tangential direction

Table 4.78: Element reference guide (NKTP = 49)

Element Type	NKTP = 49, 2-D gap/friction element
Analysis Types	Nonlinear Static, see notes 2, 13
Degrees of Freedom	2 per node: UX, UY
NORDR (Shape / No. of nodes)	- Line: 2 nodes (NORDR = 1)
Real Constants (RCTABLE)	9 constants: Kn, Kt, Sn, St, RGKEY, GW, RTKEY, THETA, TOL where,
Entry no.	
1	Kn Axial stiffness (in the normal direction), see note 3
2	Kt Tangential stiffness (stiffness in sticking status), see note 3, [default = Kn]
3	Sn small stiffness in normal direction for rigid body motion, $Sn \ll Kn$, see note 4
4	St small stiffness in tangential direction for rigid body motion, $St \ll Kt$, see note 4
5	RGKEY Key for method of gap width definition, see notes 5, 6 = 0.0 gap width is defined by next entry in the table = 1.0 determine gap width from nodal coordinates, next entry in the table is ignored.
6	GW initial gap width, used only if RGKEY = 0.0, see notes 5, 6
7	RTKEY key for method of definition of local coordinate system, see notes 7, 8 = 0.0 orientation of normal axis n is the direction from node 1 to node 2 in the initial configuration. < 1.0 orientation of normal axis n is the direction from node 2 to node 1 in the initial configuration. > 1.0 orientation of normal axis n is defined by the global components of the vector n (entries 8, 9, 10)

<p>8</p>	<p>THETA angle in degrees from the global X-axis to the n axis measured about the global Z-axis, used only if RTKEY = 0.0, see notes 7, 8</p>
<p>9</p>	<p>TOL convergence tolerance on shear force when the element oscillates between sticking and sliding status, see note 9, [default = 1.E-3].</p> <p>> 0.0 value for convergence tolerance on shear force</p> <p>< 0.0 convergence should be based on unchanging element status only</p>
<p>Material Properties</p>	<p>1 property: coefficient of friction, <i>entered using the label NUXY</i> in the *MATERIAL data group, see note 10</p>
<p>Element Output</p>	<p>the element output includes (see note 11):</p> <ul style="list-style-type: none"> - initial element parameters including values of normal and transverse stiffness coefficients, initial gap width, initial status, friction coefficient and local axes (n, t1, t2) definition - current gap width = $(u_2 - u_1) \cdot n + GW$ - Gap status at current and previous time steps, the status is given as a number (see note 12) <p>0 = Open Gap</p> <p>1 = Sticking mode (resultant shear force $< \mu f_n$)</p> <p>2 = Sliding mode (resultant shear force $= \mu f_n$),</p> <ul style="list-style-type: none"> - sliding distance between the two nodes projected on the local y and z axes, see note 12 - Gap forces in element local coordinate system (see note 11), Fx is normal force, Fy and Fz are shear forces.

Notes:

1. The element must lie in the global XY plane (right half of XY plane in axisymmetric case).
2. The element is available in nonlinear static analysis with various nonlinear analysis options, i.e., material and/or geometric. A nonlinear analysis type (NLTYPE = GAP, [Section 5.3.3](#)) is available to analyze contact problems wherein the only nonlinearity is

introduced by this interface element. In this case, all other elements remain linear elastic throughout the analysis.

- The axial stiffness of the gap (K_n , see Figure 4.47) should be based on the stiffnesses of the adjacent elements. For most problems, it may be estimated as one to three order of magnitude higher than the stiffness of the adjacent elements. The tangential stiffness K_t defaults to K_n if not specified. It represents resistance in the tangential direction when the gap is in sticking mode. For frictionless contact problems (with zero coefficient of friction), the tangential stiffness K_t is not used.

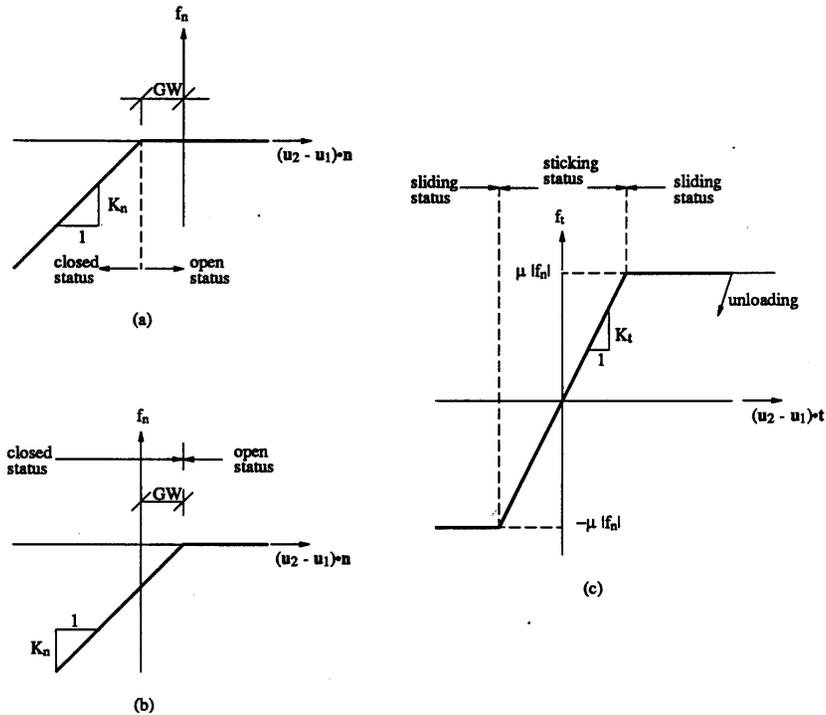


Figure 4.47: Force-displacement relationships for the gap/friction element (without soft springs, $S_n = S_t = 0$), (a) Normal direction for positive gap width, (b) Normal direction for negative gap width, (c) Tangential direction for a closed gap and initial loading.

- Small stiffnesses should be specified for S_n and S_t (entries no. 3,4 in RCTABLE) in cases where rigid body motion may occur initially, upon break in contact or during sliding motion. The small stiffnesses S_n and/or S_t are assumed to be much smaller than K_n and/or K_t (say $S_n = 1.E-8 * K_n$). These soft springs provide small stiffnesses and

transmit small forces even if the element is in an open status. The effect of these soft springs on the solution accuracy is negligible. The forces transmitted in these soft springs are included in the output of the normal and shear forces of the element.

5. The initial gap width may be defined by two methods using the key RGKEY (entry 5 in RCTABLE)
 - (a) RGKEY = 0.0 - the gap width is defined directly in entry 6 of the real constant table.
 - (b) RGKEY > 1.0 - the gap width is computed internally from the nodal coordinates,

$$GW = (\mathbf{x}_2 - \mathbf{x}_1) \bullet \mathbf{n}$$

where, \mathbf{x}_2 and \mathbf{x}_1 are the position vectors of nodes 2 and 1, respectively, and \mathbf{n} is the gap normal direction (defined using entries 7 and 8 in the real constant table).

6. Negative gap width, GW, (either specified or calculated internally) indicates interference between the two contacting bodies, whereas positive gap width indicates clearance or separation. The initial gap width may also be specified (or calculated) as zero. In fact, the two nodes of the element may be coincident. The initial gap status is determined as follows:

GW < 0.0 - closed status (sticking mode for frictional case)

GW > 0.0 - open status (no contact initially)

Initial interference condition implies an initial load, which is automatically accounted for by the program.

7. The element orientation is the direction from node 1 to node 2 when the gap is open. The element orientation may be defined by one of two methods. Either method defines the direction of the normal axis to the interface, \mathbf{n} . The axis tangent to the interface, \mathbf{t} , is obtained from the vector cross product,

$$\mathbf{n} \times \mathbf{t} = \mathbf{k}$$

where, \mathbf{k} is a unit vector along the global Z-axis. The two methods are:

- (a) RTKEY = 0.0

the orientation of the normal axis \mathbf{n} is defined by the angle THETA (entry 8 in RCTABLE) which is measured from the global X-axis and about the global Z-axis.

(b) RTKEY = 1,2

the orientation of normal axis \mathbf{n} is computed internally from initial nodal coordinates,

$$\mathbf{n} = (\mathbf{x}_2 - \mathbf{x}_1) / \|\mathbf{x}_2 - \mathbf{x}_1\| \text{ for RTKEY} = 1.0$$

$$\mathbf{n} = (\mathbf{x}_1 - \mathbf{x}_2) / \|\mathbf{x}_1 - \mathbf{x}_2\| \text{ for RTKEY} = 2.0$$

RTKEY = 1 is used for gap element without modeling interference. RTKEY = 2 should be used whenever interference is modeled in the model. In this case, the direction from node 1 to node 2 in the initial configuration is in the opposite direction of an open gap element. The two nodes should not be coincident in this case, otherwise \mathbf{n} is assumed to be in the direction of the global x-axis.

8. The gap closure is measured in the normal direction \mathbf{n} , which is a specified fixed direction in space. The element orientation remains the same throughout the analysis and does not change due to deformation. This is applicable to all nonlinear analysis types, including geometric nonlinearity.
9. The element is assumed to have converged when its status does not change between two successive iterations. However, in some cases of frictional contact, the element status may change between sticking and sliding or vice versa but with no appreciable changes in the resisting forces. In this case, convergence is still assumed when,

$$\frac{|f_n^{(i)} - f_n^{(i-1)}|}{|f_n^{(i)}|} < TOL \quad \text{and} \quad \frac{Abs(|f_p^{(i)}| - \mu |f_n^{(i)}|)}{\mu |f_n^{(i)}|} < TOL$$

where, $f_n^{(i)}$ and $f_n^{(i-1)}$ are the normal forces at iterations i and $(i-1)$, respectively, and $f_p^{(i)}$ is the predicted elastic shear force, and TOL is entry number 9 in the RCTABLE, which is defaulted to 1.E-3. If the convergence is to be based only on unchanging status, the parameter TOL should be entered as a negative number. The above convergence criteria are required to be satisfied for all nonlinear analysis types (i.e., for all NLTYPE options). For geometric and/or material nonlinear analysis, additional convergence requirements, namely the displacement, energy and force criteria should also be satisfied, as specified in the command TOLERances in the *EVENT data group.

10. The coefficient of friction *must be* specified in the *MATERIAL data group, *using the label NUXY*. Frictionless problems can be analyzed by specifying the coefficient of friction as zero. The coefficient of friction may be temperature-dependent. In this case, it is computed at the average temperature of the two nodes.
11. The element output may be obtained if element stress output is requested. The element forces are also included in the stress resultant printout for beam-like elements, which is

obtained when the average nodal stress output is requested. The element forces are output in the element local coordinate system. They represent the forces exerted by the grid points on the element vertices.

12. The sliding distance is measured from the most recent time contact was established. For the frictionless case, the status of a closed gap is +2, and the relative tangential displacement between the two nodes is printed under the sliding displacement header.
13. If the element is used with the option "NLTYPE = LINEAR" (i.e., the linear analysis option is selected in nonlinear static analysis), it behaves as two orthogonal linear springs with the following stiffnesses in the normal and tangential directions: (Kn, Kt) if $GW < 0.0$, or (Sn, St) if $GW > 0.0$.
14. The 2-D and 3-D gap elements available in linear static analysis (NKTP = 42,43) have similar capability to this element. However, unlike the current element, they neither have frictional capability, nor can they be used in conjunction with other types of nonlinearities.

4.38 3-D Gap/Friction Element (NKTP = 50)

Description

This element is a 2-node interface element which may be used in 3-D problems to model contact at discrete points between two bodies. The element has three translational degrees of freedom (UX, UY, UZ) at each node. The element is nonlinear, it may have an open or closed status. If closed, the element resists normal compressive force and tangential shear forces (Coulomb friction, if friction effect is activated). The theoretical basis of the element is similar to the 2-D gap element (NKTP = 49) which is described in [Section 2.8.6](#).

The element configuration is shown in [Figure 4.48](#). As an option, the element orientation and the initial gap width may be inferred from the location of the two nodes of the element or may be specified directly. An element reference guide is given in [Table 4.79](#). A general description of element input data is given in [Section 4.1.2](#). The output pertinent to the element is given in [Table 4.79](#).

The element is represented by coupled nonlinear orthogonal springs, one in the normal direction and two in orthogonal tangential directions to the interface. The load-displacement curves are shown in [Figure 4.49](#).

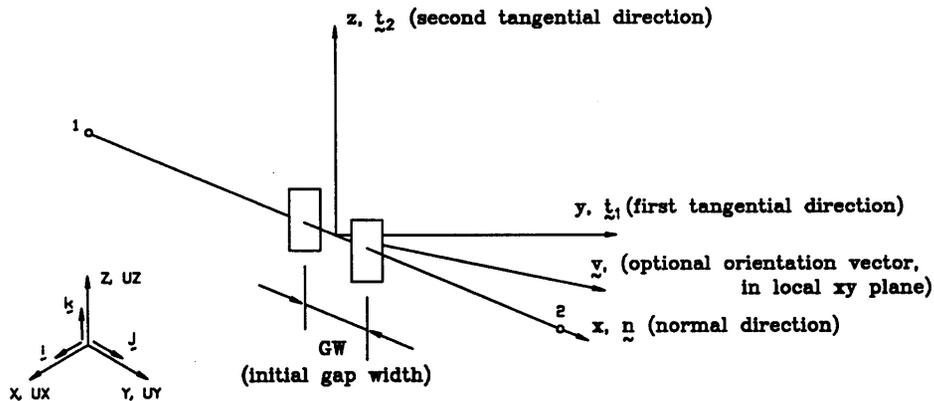


Figure 4.48: 3-D gap/friction element (NKTP = 50, NORDR = 1)

Table 4.79: Element reference guide (NKTP = 50)

Element Type	NKTP = 50, 3-D gap/friction element	
Analysis Types	Nonlinear Static, see notes 2, 13	
Degrees of Freedom	3 per node: UX, UY, UZ	
NORDR (Shape/No.of nodes)	- Line: 2 nodes (NORDR = 1)	
Real Constants (RCTABLE)	14 constants: Kn, Kt, Sn, St, RGKEY, GW, RTKEY, n1, n2, n3, v1, v2, v3, TOL where,	
Entry no.		
1	Kn	Axial stiffness (in the normal direction), see note 3
2	Kt	Tangential stiffness in y (t1) and z (t2) directions (stiffness in sticking status), see note 3, [default = Kn]
3	Sn	small stiffness in normal direction for rigid body motion, Sn << Kn, see note 4
4	St	small stiffness in tangential direction for rigid body motion, St << Kt, see note 4
5	RGKEY	Key for method of gap width definition, see notes 5, 6 = 0.0 gap width is defined by next entry in the table > 1.0 determine gap width from nodal coordinates, next entry in the table is ignored.
6	GW	initial gap width, used only if RGKEY = 0.0, see notes 5, 6
7	RTKEY	key for method of definition of local coordinate system, see notes 7, 8 = 0.0 orientation of normal axis n is the direction from node 1 to node 2 in the initial configuration. < 1.0 orientation of normal axis n is the direction from node 2 to node 1 in the initial configuration. > 1.0 orientation of normal axis n is defined by the global components of the vector n (entries 8, 9, 10)

Element Library

3-D Gap/Friction Element (NKTP = 50)

<p>8, 9, 10</p> <p>11, 12, 13</p> <p>14</p>	<p>n_1, n_2, n_3 global components of the gap axial direction n, used only if RTKEY > 1.0, see notes 7,8</p> <p>v_1, v_2, v_3 Global components of an optional orientation vector that lies in the local xy plane, see note 7</p> <p>TOL convergence tolerance on shear force when the element oscillates between sticking and sliding status, see note 9, [default = 1.E-3].</p> <p>> 0.0 value for convergence tolerance on shear force</p> <p>< 0.0 convergence should be based on unchanging element status only</p>
<p>Material Properties</p>	<p>1 property: coefficient of friction, entered using the label NUXY in the *MATERIAL data group, see note 10</p>
<p>Element Output</p>	<p>the element output includes (see note 11):</p> <ul style="list-style-type: none"> - initial element parameters including values of normal and transverse stiffness coefficients, initial gap width, initial status, friction coefficient and local axes (n, t1, t2) definition - current gap width = $(\mathbf{u2} - \mathbf{u1}) \cdot \mathbf{n} + \text{GW}$ - Gap status at current and previous time steps, the status is given as a number (see note 12) <p>0 = Open Gap</p> <p>1 = Sticking mode (resultant shear force $< \mu f_n$)</p> <p>2 = Sliding mode (resultant shear force = μf_n),</p> <ul style="list-style-type: none"> - sliding distance between the two nodes projected on the local y and z axes, see note 12 - Gap forces in element local coordinate system (see note 11), Fx is normal force, Fy and Fz are shear forces.

Notes:

1. A 2-D version of this element is NKTP = 49 which is described in [Section 2.8.6](#) and [Section 4.37](#). The Friction law for this element has the form,

$$\|\mathbf{f}_t\| = \mu|\mathbf{f}_n|$$

where, μ is the coefficient of friction, \mathbf{f}_n is the compressive normal force, and $\|\mathbf{f}_t\|$ is the,

$$\|\mathbf{f}_t\| = (\mathbf{f}_{t1}^2 + \mathbf{f}_{t2}^2)^{1/2}$$

where, \mathbf{f}_{t1} and \mathbf{f}_{t2} are the tangential forces along local y and z axes.

2. The element is available in nonlinear static analysis with various nonlinear analysis options, i. e., material and/or geometric. A nonlinear analysis type (NLTYPE = GAP, [Section 5.3.3](#)) is available to analyze contact problems wherein the only nonlinearity is introduced by this interface element. In this case, all other elements remain linear elastic throughout the analysis.
3. The axial stiffness of the gap (K_n , see [Figure 4.49](#)) should be based on the stiffnesses of the adjacent elements. For most problems, it may be estimated as one to three order of magnitude higher than the stiffness of the adjacent elements. The tangential stiffness K_t defaults to K_n if not specified. It represents resistance in the tangential directions when the gap is in sticking mode. For frictionless contact problems (with zero coefficient of friction), the tangential stiffness K_t is not used.

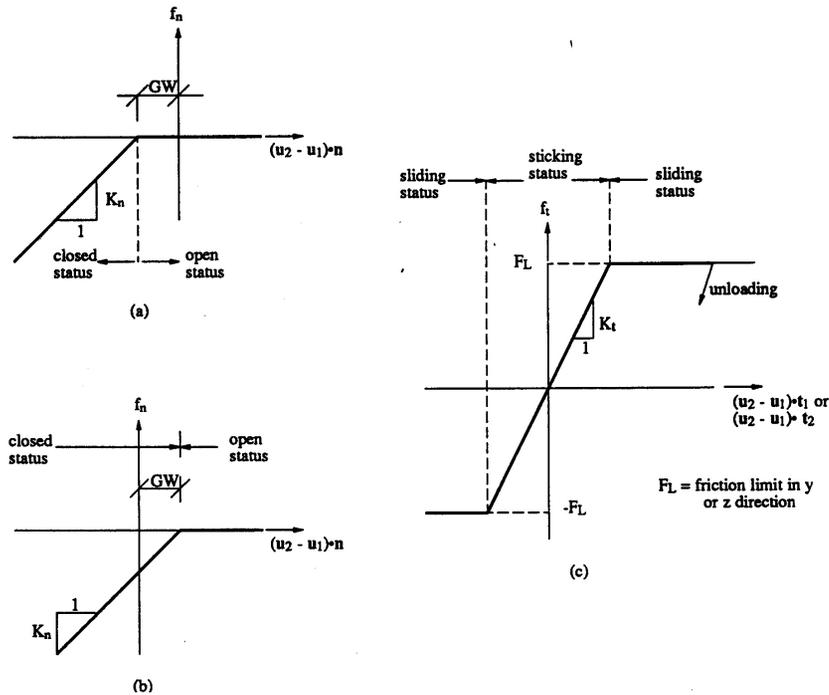


Figure 4.49: Force-displacement relationships for the gap/friction element (without soft springs, $S_n = S_t = 0$), (a) Normal direction for positive gap width, (b) Normal direction for negative gap width, (c) Tangential direction for a closed gap and initial loading.

4. Small stiffnesses should be specified for S_n and S_t (entries no. 3,4 in RCTABLE) in cases where rigid body motion may occur initially, upon break in contact or during sliding motion. The small stiffnesses S_n and/or S_t are assumed to be much smaller than K_n and/or K_t (say $S_n = 1.E-8 * K_n$). These soft springs provide small stiffnesses and transmit small forces even if the element is in an open status. The effect of these soft springs on the solution accuracy is negligible. The forces transmitted in these soft springs are included in the output of the normal and shear forces of the element.
5. The initial gap width may be defined by two methods using the key RGKEY (entry 5 in RCTABLE)
 - (a) RGKEY = 0.0 - the gap width is defined directly in entry 6 of the real constant table.
 - (b) RGKEY = 1.0 - the gap width is computed internally from the nodal coordinates:

$$GW = (\mathbf{x}_2 - \mathbf{x}_1) \bullet \mathbf{n}$$

where, \mathbf{x}_2 and \mathbf{x}_1 are the position vectors of nodes 2 and 1, respectively, and \mathbf{n} is the gap normal direction (defined using entry 7 in the real constant table).

6. Negative gap width, GW , (either specified or calculated internally) indicates interference between the two contacting bodies, whereas positive gap width indicates clearance or separation. The initial gap width may also be specified (or calculated) as zero. In fact, the two nodes of the element may be coincident. The initial gap status is determined as follows:

$GW = 0.0$ - closed status (sticking mode for frictional case)

$GW > 0.0$ - open status (no contact initially)

Initial interference condition implies an initial load which is automatically accounted for by the program.

7. The element local coordinate system $(x,y,z) = (n,t_1,t_2)$ is defined as follows:

- (a) The normal direction n (local x axis):

The normal direction of a gap element is the direction from node 1 to node 2 when the gap is open. The n axis can be defined by one of the three methods using the key RTKEY (entry 7 in RCTABLE)

- (i) $RTKEY < 0.0$ - the normal axis is inferred from the initial nodal coordinates,

$$\mathbf{n} = (\mathbf{x}_2 - \mathbf{x}_1) / \|\mathbf{x}_2 - \mathbf{x}_1\| \quad \text{for } RTKEY = 0$$

$$\mathbf{n} = (\mathbf{x}_1 - \mathbf{x}_2) / \|\mathbf{x}_1 - \mathbf{x}_2\| \quad \text{for } RTKEY < 0$$

where, \mathbf{x}_2 and \mathbf{x}_1 are the position vectors of nodes 2 and 1, respectively. The two nodes should not be coincident in this case, otherwise \mathbf{n} is assumed to be in the direction of the global X -axis.

- (ii) $RTKEY > 1.0$ – the global components of the vector \mathbf{n} are given in entries 8, 9 and 10 of the RCTABLE

- (b) The tangential directions t_1 and t_2 (local y and z axis):

The local y and z axes are defined by an optional orientation vector \mathbf{v} that lies in the local xy plane, using entries 11, 12, 13 in RCTABLE.

- (i) if a null vector is specified, $\mathbf{v} = 0$:

$$\mathbf{t}_1 = \mathbf{k} \times \mathbf{n}, \mathbf{t}_2 = \mathbf{n} \times \mathbf{t}_1$$

where, \mathbf{k} is a unit vector along global Z axis. If this fails because \mathbf{n} is aligned with \mathbf{k} , then

$$\mathbf{t}_1 = \mathbf{n} \times \mathbf{i}, \mathbf{t}_2 = \mathbf{n} \times \mathbf{t}_1, \text{ where, } \mathbf{i} \text{ is a unit vector along global X-axis.}$$

- (ii) If a non-null vector is specified, $\mathbf{v} \neq 0$:

$$\mathbf{d} = \frac{\mathbf{v}}{\|\mathbf{v}\|}, \quad \mathbf{t}_2 = \mathbf{n} \times \mathbf{d}, \quad \mathbf{t}_1 = \mathbf{t}_2 \times \mathbf{n}$$

8. The gap closure is measured in the normal direction \mathbf{n} , which is a specified fixed direction in space. The element orientation remains the same throughout the analysis and does not change due to deformation. This is applicable to all nonlinear analysis types, including geometric nonlinearity.
9. The element is assumed to have converged when its status does not change between two successive iterations. However, in some cases of frictional contact, the element status may change between sticking and sliding or vice versa but with no appreciable changes in the resisting forces. In this case, convergence is still assumed when,

$$\frac{|f_n^{(i)} - f_n^{(i-1)}|}{|f_n^{(i)}|} < TOL \quad \text{and} \quad \frac{Abs(|f_p^{(i)}| - \mu |f_n^{(i)}|)}{\mu |f_n^{(i)}|} < TOL$$

where, $f_n^{(i)}$ and $f_n^{(i-1)}$ are the normal forces at iterations i and $(i-1)$, respectively, and $f_p^{(i)}$ is the predicted resultant elastic shear force, and TOL is entry number 9 in the RCTABLE, which is defaulted to 1.E-3. If the convergence is to be based only on unchanging status, the parameter TOL should be entered as a negative number. The above convergence criteria are required to be satisfied for all nonlinear analysis types (i.e., for all NLTYPE options). For geometric and/or material nonlinear analysis, additional convergence requirements, namely the displacement, energy and force criteria should also be satisfied, as specified in the command TOLERances in the *EVENT data group.

10. The coefficient of friction must be specified in the *MATERIAL data group, using the label NUXY. An isotropic friction law is used. Frictionless problems can be analyzed by specifying the coefficient of friction as zero. The coefficient of friction may be temperature-dependent. In this case, it is computed at the average temperature of the two nodes.

11. The element output may be obtained if element stress output is requested. The element forces are also included in the stress resultant printout for beam-like elements, which is obtained when the average nodal stress output is requested. The element forces are output in the element local coordinate system. They represent the forces exerted by the grid points on the element vertices.
12. The sliding distance is measured from the most recent time contact was established. For the frictionless case, the status of a closed gap is 2, and the relative tangential displacement between the two nodes is printed under the sliding displacement header.
13. If the element is used with the option "NLTYPE = LINEAR" (i.e, the linear analysis option is selected in nonlinear static analysis), it behaves as three orthogonal linear springs with the following stiffnesses in the normal and tangential directions: (Kn, Kt, Kt) if GW = 0.0, or (Sn, St, St) if GW > 0.0.
14. The 2-D and 3-D gap elements available in linear static analysis (NKTP = 42,43) have similar capability to this element. However, unlike the current element, they neither have frictional capability, nor can they be used in conjunction with other types of nonlinearities.

4.39 2-D Planar Element for Heat Transfer (NKTP = 101)

Description

This element is based on the assumption of a 2-D state of heat flow and is suitable for modeling thin flat structures lying in the global XY plane. The element has the temperature (T) as the only degree of freedom at each node. The theoretical basis of the element is discussed in Section 2.8. An element referenced guide is given in Table 4.80 with the available loading given in Table 4.81.

The element can be shaped as a 4 to 12 node quadrilateral or a 3 or 6 node triangle depending on the selected NORDR value. The element configuration, node locations and face numbering convention are shown in Figure 4.50. A general description of the element input data and output options are given in Section 4.1.2 and Section 4.1.3 respectively. Table 4.80 gives the output pertinent to this element.

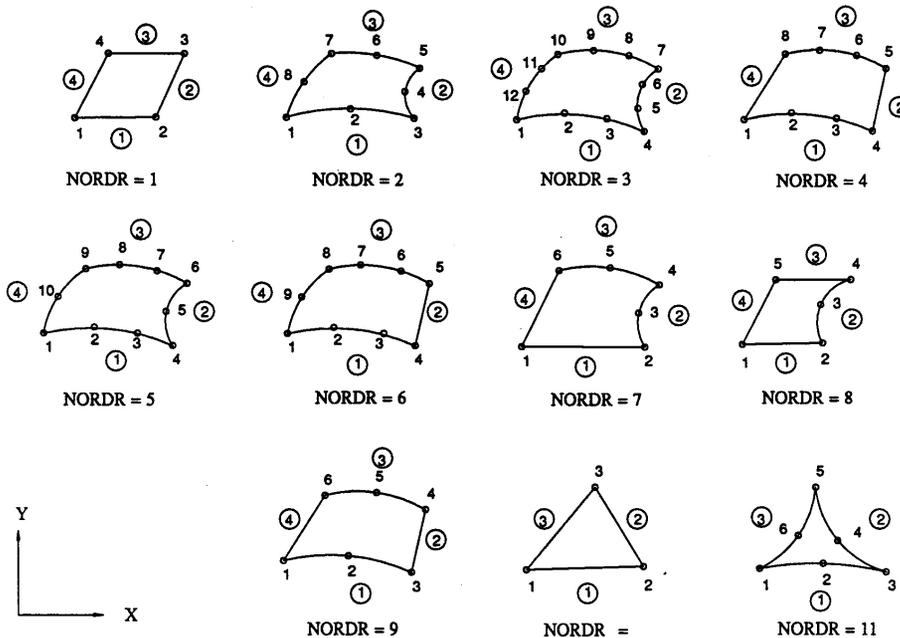


Figure 4.50: Element configuration for available NORDR values, NKTP = 101
(Circled numbers indicate face numbers)

Table 4.80: Element reference guide (NKTP = 101)

Element Type	NKTP = 101, 2-D Planar element for heat transfer.
Analysis Types	Steady state and Transient heat transfer.
Degrees of Freedom	1 per node: T (temperature)
NORDR (Element shape, nodes)	<ul style="list-style-type: none"> - Quadrilateral: 4 to 12 nodes (NORDR = 1 to 9) - Triangle: 3 or 6 nodes (NORDR = 10, 11).
Real Constants	3 to 12 nodal thicknesses (same as no. of nodes).
Material Properties	
Isotropic	3 properties: KXX, DENS, C, see note 1.
Orthotropic	4 properties: KXX, KYY, DENS, C, see note 1.
Element Output	<ul style="list-style-type: none"> - Nodal point heat flow at nodes with specified temperature. - Heat flow output on element boundary faces on which convection or radiation boundary conditions are specified.
Transient Capabilities	<ul style="list-style-type: none"> - Variable, user defined, or equal time step increments with various time integration schemes. - Time dependent heat flux, and heat generation rates - Time dependent film coefficient and emissivity for convection and radiation boundary conditions, respectively. - Time dependent ambient temperature for convection and radiation boundary conditions. - Time dependent ambient temperature for surface radiation
Nonlinear Capabilities	<ul style="list-style-type: none"> - Temperature dependent material properties. - Nonlinear radiation and convection boundary conditions with temperature dependent emissivity and film coefficient. - Surface radiation with view factor computation. - Temperature dependent heat flux and heat generation rate. - Phase change

Table 4.81: Available loading (NKTP = 101)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal heat flow values, see *CFLUX and note 2 - Specified non-zero nodal temperature values, see *SPTEMP - Specified nodal heat generation rate, see *NDHEATGEN and note 3
Distributed Loading	<ul style="list-style-type: none"> - Uniform or non-uniform distributed flux (per unit area) on any face of the element, see *DFLUX and note 2 - Uniform heat generation rate, see *ELHEATGEN and note 3 - Uniform or non-uniform convection on any face of the element, see *CONVBC - Uniform or non-uniform radiation on any face of the element, see *RADBC - Surface radiation with view factor computation, *RADSURFACE

Notes:

1. For steady state heat transfer analysis, the values of DENS and C are not required.
2. Positive heat fluxes indicate that heat is flowing into the element. Distributed flux should be specified per unit area, i.e., Heat/(time. area).
3. Positive heat generation rates indicate that heat is flowing out of the element (i.e. the element is a source of heat flow).
4. The element connectivity must be given in the order shown in [Figure 4.50](#) in which the node numbering sequence starts at a corner node and proceeds along the perimeter of the element in a counter clockwise direction.
5. For orthotropic material property input, the principal material axes must be confined to the global XY plane.

4.40 2-D Planar Element of Unit Thickness for Heat Transfer (NKTP = 102)

Description

This element is based on the assumption of a 2-D state of heat flow and is suitable for modeling thick flat structures with a constant cross-section lying in the global XY plane. The element must lie in the global XY plane. The element has a unit thickness and the temperature (T) is the only degree of freedom at each node. The theoretical basis of the element is discussed in [Section 2.8](#). An element reference guide is given in [Table 4.82](#) with the available loading given in [Table 4.83](#).

The element can be shaped as a 4 to 12 node quadrilateral, or a 3 or 6 node triangle depending on the selected NORDR value. The element configuration, node locations and face numbering convention are shown in [Figure 4.51](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.82](#) gives the output pertinent to this element.

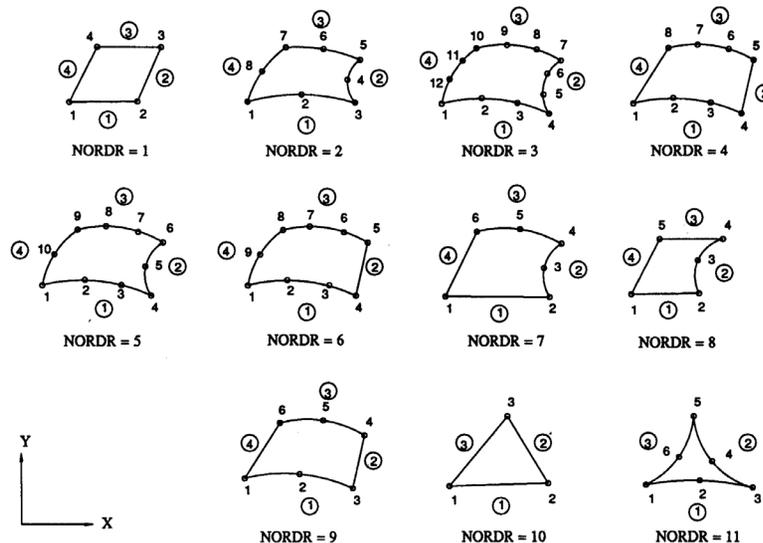


Figure 4.51: Element configuration for available NORDR values, NKTP = 102 (circled numbers indicate face numbers)

Table 4.82: Element reference guide (NKTP = 102)

Element Type	NKTP = 102, 2-D Planar element of unit thickness for heat transfer
Analysis Types	Steady state and Transient heat transfer
Degrees of Freedom	1 per node: T (temperature)
NORDR (Element shape, nodes)	<ul style="list-style-type: none"> - Quadrilateral: 4 to 12 nodes (NORDR = 1 to 9) - Triangle: 3 or 6 nodes (NORDR = 10, 11)
Real Constants	None (unit thickness is assumed)
Material Properties	
Isotropic	3 properties: KXX, DENS, C, see note 1
Orthotropic	4 properties: KXX, KYY, DENS, C, see note 1
Element Output	<ul style="list-style-type: none"> - Nodal point heat flow at nodes with specified temperature - Heat flow output on element boundary faces on which convection or radiation boundary conditions are specified
Transient Capabilities	<ul style="list-style-type: none"> - Variable, user defined, or equal time step increments with various time integration schemes - Time dependent heat flux, and heat generation rates - Time dependent film coefficient and emissivity for convection and radiation boundary conditions, respectively - Time dependent ambient temperature for convection and radiation boundary conditions - Time dependent ambient temperature for surface radiation
Nonlinear Capabilities	<ul style="list-style-type: none"> - Temperature dependent material properties - Nonlinear radiation and convection boundary conditions with temperature dependent emissivity and film coefficient - Surface radiation with view factor computation - Temperature dependent heat flux and heat generation rate - Phase change

Table 4.83: Available loading (NKTP = 102)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal heat flow values, see *CFLUX and note 2 - Specified non-zero nodal temperature values, see *SPTMP - Specified nodal heat generation rate, see *NDHEATGEN and note 3
Distributed Loading	<ul style="list-style-type: none"> - Uniform or non-uniform distributed flux (per unit area) on any face of the element, see *DFLUX and note 2 - Uniform heat generation rate, see *ELHEATGEN and note 3 - Uniform or non-uniform convection on any face of the element, see *CONVBC - Uniform or non-uniform radiation on any face of the element, see *RADBC - Surface radiation with view factor computation, *RADSURFACE

Notes:

1. For steady state heat transfer analysis, the values of DENS and C are not required.
2. Positive heat fluxes indicate that heat is flowing *into* the element. Concentrated flux is per unit depth. Distributed flux should be specified per unit area, i.e., $Heat/(Time \cdot Area)$. Note that a unit depth is assumed.
3. Positive heat generation rates indicate that heat is flowing *out* of the element (i.e. the element is a *source* of heat flow).
4. The element connectivity must be given in the order shown in [Figure 4.50](#) in which the node numbering sequence starts at a corner node and proceeds along the perimeter of the element in a counterclockwise direction.
5. For orthotropic material property input, the principal material axes must be confined to the global XY plane.

4.41 Axisymmetric Solid Element for Heat Transfer (NKTP = 103)

Description

This element is based on the assumption of axisymmetric state of heat flow and is suitable for modeling axisymmetric structures with a section lying in the *right* half of the global XY plane. The global X-axis is the radial direction and the global Y-axis is the axis of revolution or axis of symmetry. The element has the temperature (T) as the only degree of freedom at each node. The theoretical basis of the element is discussed in [Section 2.8](#). An element reference guide is given in [Table 4.84](#) with the available loading given in [Table 4.85](#).

The element can be shaped as a 4 to 12 node quadrilateral, or a 3 or 6 node triangle depending on the selected NORDR value. The element configuration, node locations and face numbering convention are shown in [Figure 4.52](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.84](#) gives the output pertinent to this element.

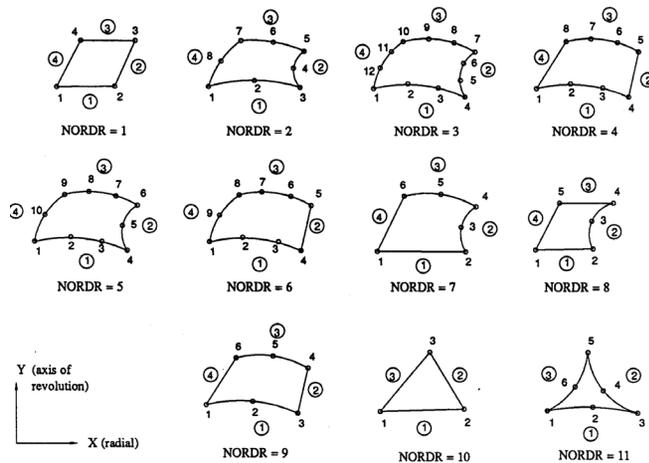


Figure 4.52: Element configuration for available NORDR values, NKTP = 103 (circled numbers indicated face numbers)

Table 4.84: Element reference guide (NKTP = 103)

Element Type	NKTP = 103, Axisymmetric solid element for heat transfer
Analysis Types	Steady state and Transient heat transfer
Degrees of Freedom	1 per node: T (temperature)
NORDR (Element shape, nodes)	<ul style="list-style-type: none"> - Quadrilateral: 4 to 12 nodes (NORDR = 1 to 9) - Triangle: 3 or 6 nodes (NORDR = 10, 11)
Real Constants	None
Material Properties	
Isotropic	3 properties: KXX, DENS, C, see note 1
Orthotropic	4 properties: KXX, KYY, DENS, C, see note 1
Element Output	<ul style="list-style-type: none"> - Nodal point heat flow at nodes with specified temperature. - Heat flow output on element boundary faces on which convection or radiation boundary conditions are specified
Transient Capabilities	<ul style="list-style-type: none"> - Variable, user defined, or equal time step increments with various time integration schemes. - Time dependent heat flux, and heat generation rates. - Time dependent film coefficient and emissivity for convection and radiation boundary conditions, respectively. - Time dependent ambient temperature for convection and radiation boundary conditions. - Time dependent ambient temperature for surface radiation
Nonlinear Capabilities	<ul style="list-style-type: none"> - Temperature dependent material properties. - Nonlinear radiation and convection boundary conditions with temperature dependent emissivity and film coefficient. - Surface radiation with view factor computation. - Temperature dependent heat flux and heat generation rate. - Phase change

Table 4.85: Available loading (NKTP = 103)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal heat flow values, see *CFLUX and note 2. - Specified non-zero nodal temperature values, see *SPTMP - Specified nodal heat generation rate, see *NDHEATGEN and note 3
Distributed Loading	<ul style="list-style-type: none"> - Uniform or non-uniform distributed flux (per unit area) on any face of the element, see *DFLUX and note 2 - Uniform heat generation rate, see *ELHEATGEN and note 3 - Uniform or non-uniform convection on any face of the element, see *CONVBC - Uniform or non-uniform radiation on any face of the element, see *RADBC - Surface radiation with view factor computation, *RADSURFACE

Notes:

1. For steady state heat transfer analysis, the values of DENS and C are not required.
2. Concentrated nodal heat flux values are specified for the entire circumference whereas distributed fluxes are specified per unit area. Positive heat fluxes indicate that heat is flowing into the element.
3. Heat generation rates are specified per unit volume. Positive heat generations indicate that the heat is flowing *out* of the element (i.e. the element is a source of heat flow).
4. The element connectivity must be given in the order shown in [Figure 4.52](#) in which the node numbering sequence starts at a corner node and proceeds along the perimeter of the element in a counterclockwise direction.
5. For orthotropic material property input, the principal material axes must be confined to the global XY plane.

4.42 3-D Solid Element for Heat Transfer (NKTP = 104)

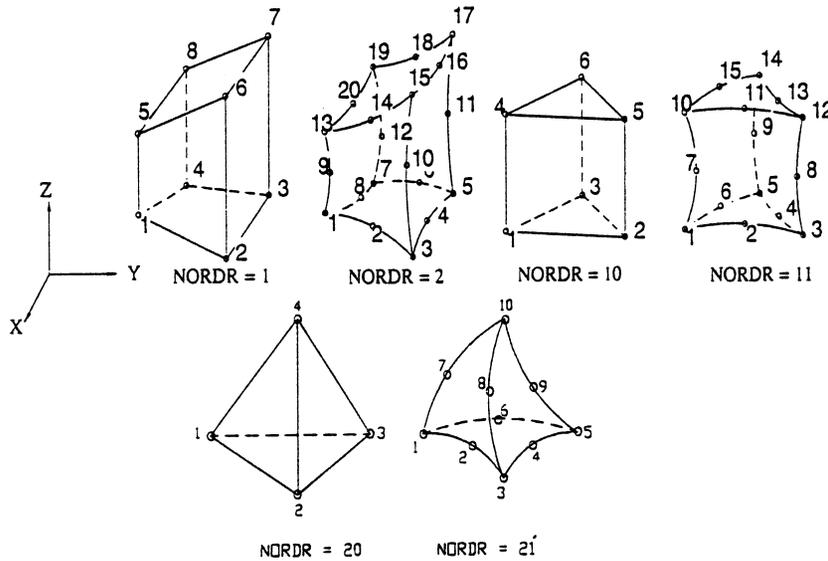
Description

This element models the three-dimensional state of heat flow and is suitable for modeling heat transfer in solid structures. The element has the temperature (T) as the only degree of freedom at each node. The theoretical basis of the element is discussed in [Section 2.8](#). An element reference guide is given in [Table 4.86](#) with the available loading given in [Table 4.87](#).

The element can be shaped as an 8 or 20 node hexahedron (brick), or a 6 or 15 node wedge, or a 4 or 8 noded tetrahedron, depending on the selected NORDR value. The element configuration, node locations and face numbering convention are shown in [Figure 4.53](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.86](#) gives the output pertinent to this element.

Element Library

3-D Solid Element for Heat Transfer (NKTP = 104)



Face No.	NORDR					
	1	2	10	11	20	21
1	1,2,3,4	1,2,3,4,5,6,7,8	1,2,3	1,2,3,4,5,6	1,2,3	1,2,3,4,5,6
2	5,6,7,8	13,14,15,16,17,18,19,20	4,5,6	10,11,12,13,14,15	1,2,4	1,2,3,8,10,7
3	1,2,6,5	1,2,3,10,15,14,13,9	1,2,5,4	1,2,3,8,12,11,10,7	2,3,4	3,4,5,9,10,8
4	2,3,7,6	3,4,5,11,17,16,15,10	2,3,6,5	3,4,5,9,14,13,12,8	3,1,4	5,6,1,7,10,9
5	3,4,8,7	5,6,7,12,19,18,17,11	3,1,4,6	5,6,1,7,10,15,14,9	—	—
6	4,1,5,8	7,8,1,9,13,20,19,12	—	—	—	—

Figure 4.53: Element configurations and face numbering convention for available NORDR values, NKTP = 104

Table 4.86: Element reference guide (NKTP = 104)

Element Type	NKTP = 104, 3-D Solid element for heat transfer
Analysis Types	Steady state and Transient heat transfer
Degrees of Freedom	1 per node: T (temperature)
NORDR (Element shape, nodes)	<ul style="list-style-type: none"> - Hexahedron (brick): 8 or 20 nodes (NORDR = 1, 2) - Wedge: 6 or 15 nodes (NORDR = 10, 11) - Tetrahedron: 4 or 8 nodes (NORDR = 20,21)
Real Constants	None
Material Properties	
Isotropic	3 properties: KXX, DENS, C, see note 1
Orthotropic	5 properties: KXX, KYY, KZZ, DENS, C, see note 1
Element Output	<ul style="list-style-type: none"> - Nodal point heat flow at nodes with specified temperature - Heat flow output on element boundary faces on which convection or radiation boundary conditions are specified
Transient Capabilities	<ul style="list-style-type: none"> - Variable, user defined, or equal time step increments with various time integration schemes - Time dependent heat flux, and heat generation rates - Time dependent film coefficient and emissivity for convection and radiation boundary conditions, respectively - Time dependent ambient temperature for convection and radiation boundary conditions - Time dependent ambient temperature for surface radiation
Nonlinear Capabilities	<ul style="list-style-type: none"> - Temperature dependent material properties - Nonlinear radiation and convection boundary conditions with temperature dependent emissivity and film coefficient - Surface radiation with view factor computation - Temperature dependent heat flux and heat generation rate - Phase change

Table 4.87: Available loading (NKTP = 104)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal heat flow values, see *CFLUX and note 2 - Specified non-zero nodal temperature values, see *SPTMP - Specified nodal heat generation rate, see *NDHEATGEN and note 3
Distributed Loading	<ul style="list-style-type: none"> - Uniform or non-uniform distributed flux (per unit area) on any face of the element, see *DFLUX and note 2 - Uniform heat generation rate, see *ELHEATGEN and note 3 - Uniform or non-uniform convection on any face of the element, see *CONVBC - Uniform or non-uniform radiation on any face of the element, see *RADBC - Surface radiation with view factor computation, *RADSURFACE

Notes:

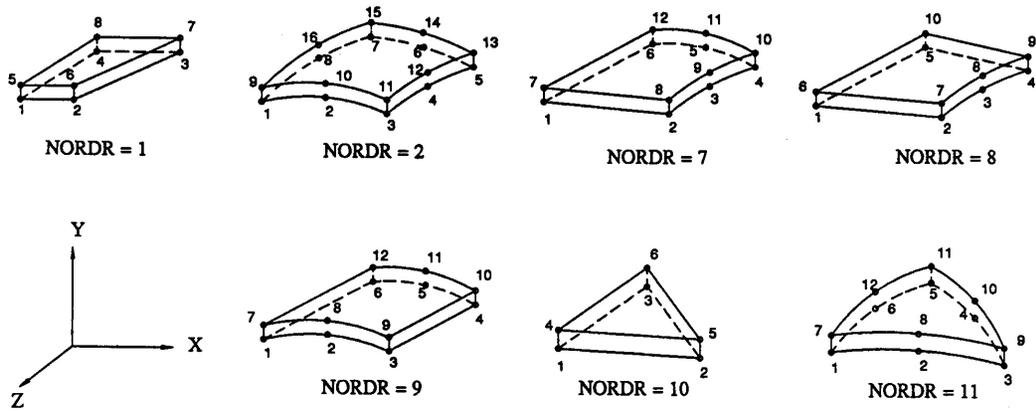
1. For steady state heat transfer analysis, the values of DENS and C are not required.
2. Positive heat fluxes indicate that heat is flowing *into* the element.
3. Positive heat generation rates indicate that the heat is flowing *out* of the element (i.e. the element is a *source* of heat flow).
4. The element connectivity must be given in the order shown in [Figure 4.53](#) in which the node numbering sequence starts at a corner node and proceeds along the perimeter of a face on the element. This face will be considered face-1 of the element. Following the right hand rule, the midside nodes of the edges are then defined with the first edge midside node above the first node of face-1. Face-2 of the element follows similarly. For wedge elements, face-1 must be a triangular face.
5. For orthotropic material property input, the principal material axes may coincide with global axes or be specified independently.

4.43 3-D Thick Shell Element for Heat Transfer (NKTP = 105)

Description

This element is derived by specializing the 3-D solid element (NKTP = 104) to have a linear variation of temperature through the thickness. It is suited for modeling heat transfer in thick shell structures. The element has the temperature (T) as the only degree of freedom at each node. The theoretical basis of the element is discussed in [Section 2.8](#). An element reference guide is given in [Table 4.88](#) with the available loading given in [Table 4.89](#).

The element can be shaped as an 8 to 16 node hexahedron (brick), or 6 or 12 node wedge depending on the selected NORDR value. The element configuration, node locations and face numbering convention are shown in [Figure 4.54](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.88](#) gives the output pertinent to this element.



Face No.	NORDR						
	1	2	7	8		10	11
1	1,2,3,4	1,2,3,4,5,6,7,8	1,2,3,4,5,6	1,2,3,4,5,	1,2,3,4,5,6	1,2,3	
2	5,6,7,8	9,10,11,12, 13,14,15,16	7,8,9,10,11,12	6,7,8,9,10	7,8,9,10,11,12	4,5,6	7,8,9,10, 11,12
3	1,2,6,5	1,2,3,11,10,9	1,2,8,7	1,2,7,6	1,2,3,9,8,7	1,2,5,4	1,2,3,9,8,7
4	2,3,7,6	3,4,5,13,12,11	2,3,4,10,9,8	2,3,4,9,8,7	3,4,10,9	2,3,6,5	3,4,5,11,10,9
5	3,4,8,7	5,6,7,15,14,13	4,5,6,12,11,10	4,5,10,9	4,5,6,12,11,10	3,1,4,6	5,6,1,7,12,11
6	4,1,5,8	7,8,1,9,16,15	6,1,7,12	5,1,6,10	6,1,7,12	—	—

Figure 4.54: Element configuration and face numbering convention for available NORDR values, NKTP = 105

Table 4.88: Element reference guide (NKTP = 105)

Element Type	NKTP = 105, 3-D Thick shell element for heat transfer
Analysis Types	Steady state and Transient heat transfer
Degrees of Freedom	1 per node: T (temperature)
NORDR (Element shape, nodes)	<ul style="list-style-type: none"> - Hexahedron (brick): 8 to 16 nodes (NORDR = 1, 2, 7 to 9) - Wedge: 6 or 12 nodes (NORDR = 10, 11)
Real Constants	None
Material Properties	
Isotropic	3 properties: KXX, DENS, C, see note 1 5
Orthotropic	5 properties: KXX, KYY, KZZ, DENS, C, see note 1
Element Output	<ul style="list-style-type: none"> - Nodal point heat flow at nodes with specified temperature - Heat flow output on element boundary faces on which convection or radiation boundary conditions are specified
Transient Capabilities	<ul style="list-style-type: none"> - Variable, user defined, or equal time step increments with various time integration schemes - Time dependent heat flux, and heat generation rates - Time dependent film coefficient and emissivity for convection and radiation boundary conditions, respectively - Time dependent ambient temperature for convection and radiation boundary conditions - Time dependent ambient temperature for surface radiation
Nonlinear Capabilities	<ul style="list-style-type: none"> - Temperature dependent material properties - Nonlinear radiation and convection boundary conditions with temperature dependent emissivity and film coefficient - Surface radiation with view factor computation - Temperature dependent heat flux and heat generation rate - Phase change

Table 4.89: Available loading (NKTP = 104)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal heat flow values, see *CFLUX and note 2 - Specified non-zero nodal temperature values, see *SPTMP - Specified nodal heat generation rate, see *NDHEATGEN and note 3
Distributed Loading	<ul style="list-style-type: none"> - Uniform or non-uniform distributed flux (per unit area) on any face of the element, see *DFLUX and note 2 - Uniform heat generation rate, see *ELHEATGEN and note 3 - Uniform or non-uniform convection on any face of the element, see *CONVBC - Uniform or non-uniform radiation on any face of the element, see *RADBC - Surface radiation with view factor computation, *RADSURFACE

Notes:

1. For steady state heat transfer analysis, the values of DENS and C are not required.
2. Positive heat fluxes indicate that heat is flowing *into* the element.
3. Positive heat generation rates indicate that the heat is flowing *out* of the element (i.e. the element is a *source* of heat flow).
4. The element connectivity must be given in the order shown in [Figure 4.54](#) in which the node numbering sequence starts at a corner node and proceeds along the perimeter of a face on the element. This will be considered face-1 of the element. Following the right hand rule, the nodes of face-2 of the element are then defined with the first node being above the first node of face-1. It should also be noted that the direction from face-1 to face-2 has to be the thickness direction, through which temperature variation is restricted to be linear.
5. For orthotropic material property input, the principal material axes may coincide with global axes or be specified independently.

4.44 3-D Bar Element for Heat Transfer (NKTP = 112)

Description

This element is a 2-node bar element in three dimensions for heat transfer analysis. The element may be oriented anywhere in space. The element has the temperature (T) as the only degree of freedom at each node and can incorporate heat flux, convection and radiation on any face. The theoretical basis of the element is discussed in [Section 2.8](#). An element reference guide is given in [Table 4.90](#) with the available loading given in [Table 4.91](#).

The element configuration and face number convention are shown in [Figure 4.55](#). A general description of element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.90](#) gives the output pertinent to this element.

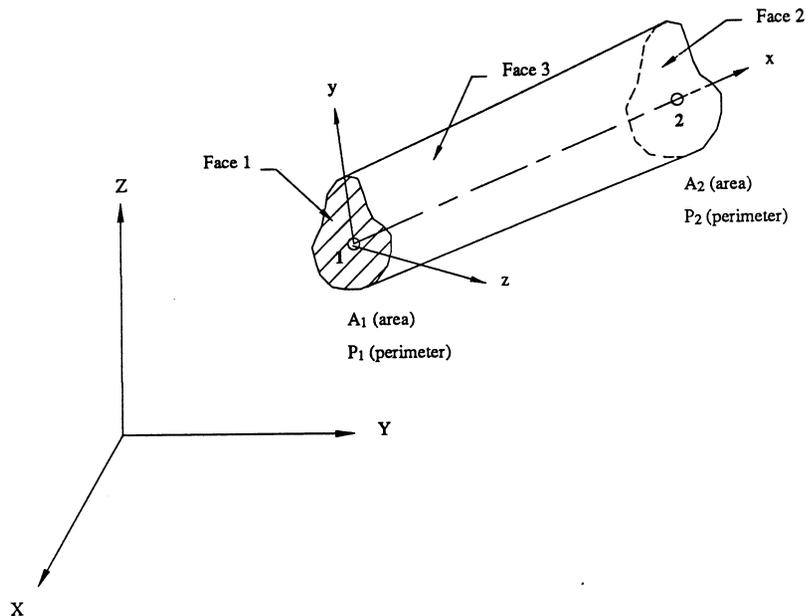


Figure 4.55: Element configuration and face number convention
(NKTP = 112, NORDR = 1)

Table 4.90: Element reference guide (NKTP = 112)

Element Type	NKTP = 112, 3-D Bar element for heat transfer
Analysis Types	Steady state and Transient heat transfer
Degrees of Freedom	1 per node: T (temperature)
NORDR (Element shape, nodes)	- Line, 2 nodes (NORDR = 1)
Real Constants	4; cross sectional areas at node 1 and 2 and perimeters at node 1 and 2, respectively (A1, A2, P1, P2).
Material Properties	3 constants: KXX, DENS, C, see note 1
Element Output	<ul style="list-style-type: none"> - Nodal point heat flow output at nodes with specified temperature - Heat flow output on element boundary face on which convection or radiation boundary conditions are specified
Transient Capabilities	<ul style="list-style-type: none"> - Variable, user defined, or equal time step increments with various time integrations schemes - Time dependent heat flux, and heat generation rates - Time dependent film coefficient and emissivity for convection and radiation boundary conditions, respectively - Time dependent ambient temperature for convection and radiation boundary conditions
Nonlinear Capabilities	<ul style="list-style-type: none"> - Temperature dependent material properties - Nonlinear radiation and convection boundary conditions with temperature dependent emissivity and film coefficient - Temperature dependent heat flux and heat generation rate

Table 4.91: Available loading (NKTP = 112)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal heat flow values, see *CFLUX and note 2 - Specified non-zero nodal temperature values, see *SPTEMP - Specified nodal heat generation rate, see *NDHEATGEN and note 3
----------------------	---

Distributed Loading	<ul style="list-style-type: none"> - Uniform or non-uniform distributed flux (<i>per unit area</i>) on any face of the element, see *DFLUX and note 2 - Uniform heat generation rate, see *ELHEATGEN and note 3 - Uniform or non-uniform convection on any face of the element, see *CONVBC - Uniform or non-uniform radiation on any face of the element, see *RADBC
----------------------------	---

Notes:

1. For steady state heat transfer analysis, the values of DENS and C are not required.
2. The element has three faces. Area and perimeter are specified at the two end nodes (input in *RCTABLE data group). The cross-sectional area at each node is required for transient heat transfer analysis. For steady state analysis, however, if the areas are not input, the effects of conduction and heat generation are ignored. The input of the perimeter at each node is required if distributed heat flux (*DFLUX), convection (*CONVBC), or radiation (*RADBC) boundary conditions are specified on face number 3. If perimeter values are not specified, these effects are ignored for face 3. The surface area of face number 3, and the volume of the element are calculated by integrating the perimeter and the cross-sectional area over the length, respectively.
3. Positive heat fluxes indicate that heat is flowing into the element. Distributed flux should be specified per unit area, i.e., $Heat/(Time \cdot Area)$.
4. Positive heat generation rates indicate that heat is flowing *out* of the element (i.e., the element is a *source* of heat flow).

4.45 3-D Curved Thin Shell Element For Heat Transfer (NKTP = 120)

Description

This element is a curved thin shell element oriented arbitrarily in space and is suitable for modeling heat transfer in thin shell structures. The element has the temperature (T) as the only degree of freedom at each node. The theoretical basis of the element is discussed in [Section 2.8](#). An element reference guide is given in [Table 4.92](#) with the available loading given in [Table 4.93](#).

The element can be shaped as a 4 to 12 node quadrilateral or a 3 or 6 node triangle depending on the selected NORDR value. The element configuration, node locations and face numbering convention are shown in [Figure 4.56](#). A general description of the element input data and output options are given in [Section 4.1.2](#) and [Section 4.1.3](#) respectively. [Table 4.92](#) gives the output pertinent to this element.

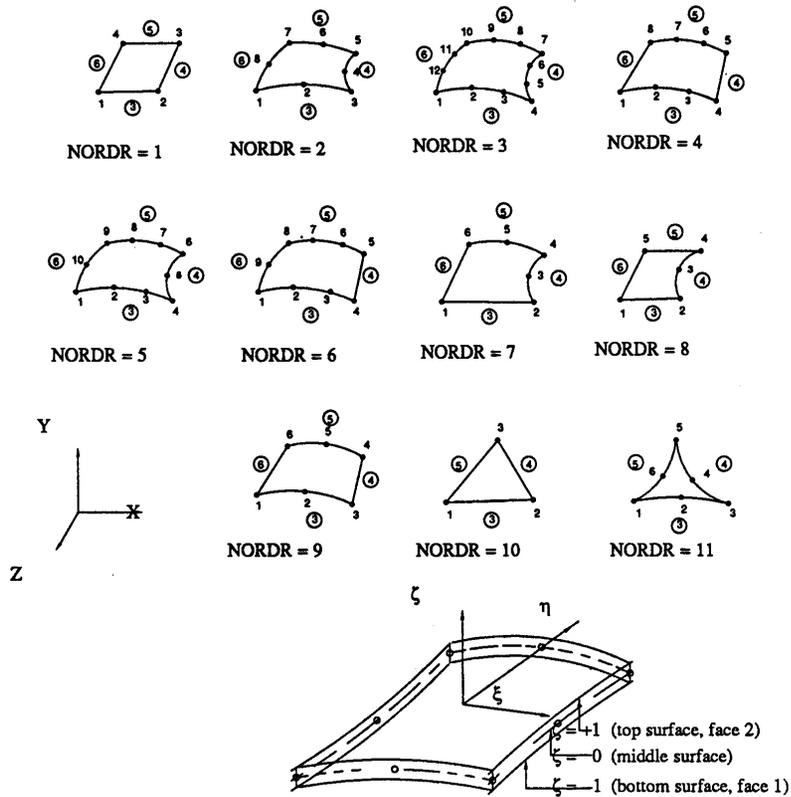


Figure 4.56: Element configuration for available NORDR values, NKTP = 120

Table 4.92: Element reference guide (NKTP = 120)

Element Type	NKTP = 120, 3-D Curved thin shell element for heat transfer
Analysis Types	Steady state and Transient heat transfer
Degrees of Freedom	1 per node: T (temperature)
NORDR (Element shape, nodes)	<ul style="list-style-type: none"> - Quadrilateral: 4 to 12 nodes (NORDR = 1 to 9) - Triangle: 3 or 6 nodes (NORDR = 10, 11)
Real Constants	3 - 12 nodal thicknesses (same as number of nodes)
Material Properties	
Isotropic	3 properties: KXX, DENS, C, see note 1
Orthotropic	5 properties: KXX, KYY, KZZ, DENS, C, see notes 1 and 5
Element Output	<ul style="list-style-type: none"> - Nodal point heat flow at nodes with specified temperature - Heat flow output on element boundary faces on which convection or radiation boundary conditions are specified - Nodal point temperature difference between top and bottom surfaces, see note 7
Transient Capabilities	<ul style="list-style-type: none"> - Variable, user defined, or equal time step increments with various time integration schemes - Time dependent heat flux, and heat generation rates - Time dependent film coefficient and emissivity for convection and radiation boundary conditions, respectively - Time dependent ambient temperature for convection and radiation boundary conditions - Time dependent ambient temperature for surface radiation
Nonlinear Capabilities	<ul style="list-style-type: none"> - Temperature dependent material properties - Nonlinear radiation and convection boundary conditions with temperature dependent emissivity and film coefficient - Surface radiation with view factor computation - Temperature dependent heat flux and heat generation rate - Phase change

Table 4.93: Available loading (NKTP = 120)

Nodal Loading	<ul style="list-style-type: none"> - Concentrated nodal heat flow values, see *CFLUX and note 2 - Specified non-zero nodal temperature values, see *SPTMP - Specified nodal heat generation rate, see *NDHEATGEN and note 3
Distributed Loading	<ul style="list-style-type: none"> - Uniform or non-uniform distributed flux (per unit area) on any face of the element, see *DFLUX and note 2 - Uniform heat generation rate, see *ELHEATGEN and note 3 - Uniform or non-uniform convection on any face of the element, see *CONVBC - Uniform or non-uniform radiation on any face of the element, see *RADBC - Surface radiation with view factor computation, *RADSURFACE

Notes:

1. For steady state heat transfer analysis, the values of DENS and C are not required.
2. Positive heat fluxes indicate that heat is flowing into the element. Distributed heat fluxes on all faces (top, bottom and side faces) should be specified per unit area, i.e., $Heat/(Time \cdot Area)$.
3. Positive heat generation rates indicate that heat is flowing *out* of the element (i.e. the element is a *source* of heat flow).
4. The element connectivity must be given in the order shown in [Figure 4.56](#) in which the node numbering sequence starts at a corner node and proceeds along the perimeter of the element.
5. For orthotropic material property input, the material directions must be defined in such a way that the material z-axis is normal to the middle surface of the element. The material property KZZ (thermal conductivity in the shell thickness direction) is used for the calculation of the nodal temperature difference between the top and bottom faces of the element. If KZZ is not specified, the KXX value is substituted for KZZ.
6. The element is suitable for modeling curved thin shell structures. Heat flux, convection and radiation, if any, may be applied to the top and bottom faces as well as the in-plane faces. Top and bottom are determined as follows:
 - (a) Choose a viewpoint so that the element connectivities are defined counterclockwise.
 - (b) The top face of the element is now closest to the observer. The bottom and top faces are numbered 1 and 2, respectively.

Element Library

3-D Curved Thin Shell Element For Heat Transfer (NKTP = 120)

The inplane faces of quadrilateral elements are numbered 3 to 6 (3 to 5 for triangles), as shown in [Figure 4.56](#).

7. The nodal point temperature difference output is obtained for nodes connected to elements for which distributed flux, convection or radiation boundary conditions are applied on top or bottom surfaces.

4.46 3-D Convection Link Element (NKTP = 149)

Description

This element is a 2-node convection link element in three dimensions and it may be used for heat convection between two nodes. The element can be used in two-dimensional (including axisymmetric) or in three-dimensional applications. The element has the temperature (T) as the only degree of freedom at each node and can be oriented anywhere in space. The theoretical basis of the element is discussed in [Section 2.8](#). An element reference guide is given in [Table 4.94](#) with the available loading given in [Table 4.95](#).

The element configuration is shown in [Figure 4.57](#).

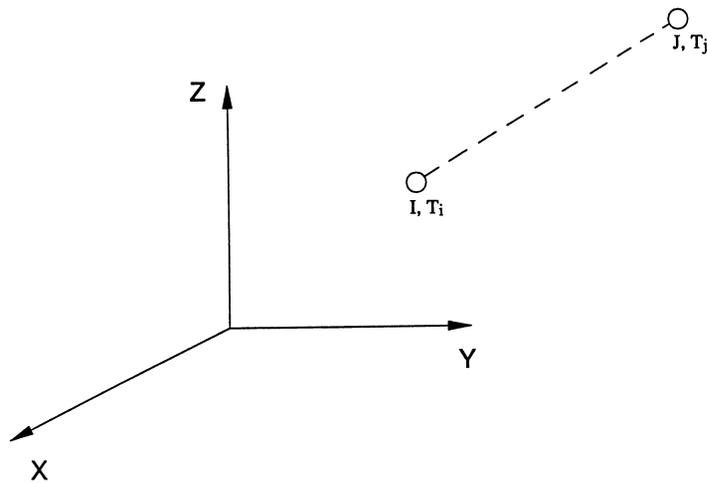


Figure 4.57: Element configuration for convection link (NKTP = 149, NORDR = 1)

Table 4.94: Element reference guide (NKTP = 149)

Element Type	NKTP = 149, 3-D convection link element for heat transfer
Analysis Types	Steady state and Transient heat transfer
Degrees of Freedom	1 per node: T (temperature)
NORDR (Element shape, nodes)	- 2 nodes (NORDR = 1), see note 3
Real Constants	2; convection surface area and film coefficient (A, h) respectively, see notes 1 and 2
Material Properties	none
Element Output	- Nodal point heat flow at nodes with specified temperature
Transient Capabilities	- Variable, user defined, or equal time step increments with various time integration schemes
Nonlinear Capabilities	- Temperature dependent film coefficient

Table 4.95: Available loading (NKTP = 149)

Nodal Loading	- Specified non-zero nodal temperature values at any one of the element nodes, see *SPTEMP
----------------------	--

Notes:

1. If the film coefficient is temperature dependent, then the curve identification number is given in *RCTABLE data group and the corresponding curve is input in *TEMPFN data group.
2. The convection surface area, A, in *RCTABLE data group, for this element is the surface area over which heat is convected. In an axisymmetric analysis, the convection area must be the total circumferential area.
3. Nodal points may or may not be coincident.

4.47 3-D Radiation Link Element (NKTP = 150)

Description

This element is a 2-node radiation link element in three dimensions and may be used for heat radiation between two nodes. The element can be used in two-dimensional (including axisymmetric) or in three-dimensional applications. The element has the temperature (T) as the only degree of freedom at each node and can be oriented anywhere in space. The theoretical basis of the element is discussed in [Section 2.8](#). An element reference guide is given in [Table 4.96](#) with the available loading given in [Table 4.97](#).

The element configuration is shown in [Figure 4.57](#).

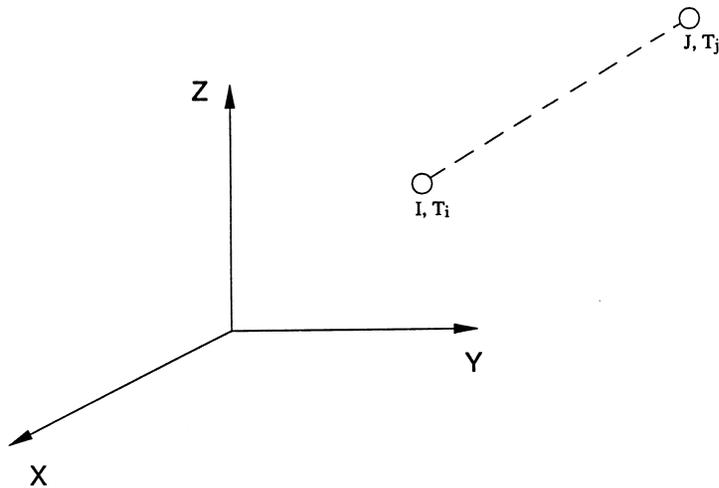


Figure 4.58: Element configuration for radiation link (NKTP = 150, NORDR = 1)

Table 4.96: Element reference guide (NKTP = 150)

Element Type	NKTP = 150, 3-D radiation link element for heat transfer
Analysis Types	Steady state and Transient heat transfer
Degrees of Freedom	1 per node: T (temperature)
NORDR (Element shape, nodes)	- 2 nodes (NORDR = 1), see note 3
Real Constants	4; surface area for radiation (A), surface emissivity (ϵ), shape factor (F), and Stefan-Boltzman constant (σ), see notes 1 and 2
Material Properties	none
Element Output	- Nodal point heat flow output at nodes with specified temperature
Transient Capabilities	- Variable, user defined, or equal time step increments with various time integration schemes

Table 4.97: Available loading (NKTP = 150)

Nodal Loading	- Specified non-zero nodal temperature values at any one of the element nodes, see *SPTMP
----------------------	---

Notes:

1. If the emissivity is temperature dependent, then the curve identification number is given in *RCTABLE data group and the corresponding curve is input in *TEMPFN data group.
2. The input quantity, A in *RCTABLE data group, for this element is the surface area over which heat is radiating. In an axisymmetric analysis, the radiation area must be the total circumferential area.
3. Nodal points may or may not be coincident.

Input Setup and Executive Commands

5.1 Introduction and Input Data Setup

The input data setup for a typical NISA analysis run, as well as examples for input data setup for various analysis types, are described in this chapter. The analysis types that may be performed with the following data setup are: linear static analysis (STATIC); nonlinear static analysis (NLSTATIC); vibration eigenvalue analysis (EIGENVALUE); eigenvalue buckling analysis (BUCKLING); direct transient dynamic analysis (LTRANSIENT); direct frequency response analysis (DFREQ); steady state, and transient heat transfer analyses (SHEAT, THEAT, respectively). Modal dynamic analyses, namely transient dynamics, random vibration, frequency response, and shock spectrum analyses, are performed in a second pass after the eigenvalue analysis. Description of the input data setup for these analyses is given in [Chapter 8](#).

This chapter also explains the rules of the free format input and gives detailed descriptions of the executive commands, which constitutes the first data block in a typical NISA deck, and their applicable analysis types. Detailed descriptions of the other two data blocks in a typical NISA deck, namely the model data block and the analysis data block are given in [Chapter 6](#) and [Chapter 7](#) respectively.

The input data for a typical NISA analysis consists of three data blocks and a data terminator as shown in [Figure 5.1](#). The data deck should be preceded by the appropriate job control cards (JCL) pertinent to the operating system. These control cards are required for batch mode execution of NISA and for ‘background’ runs on some interactive computers. Control cards vary according to the type of computer system and the particulars of the

analysis being performed. System manuals should be consulted for the detailed description of job control cards.

The three data blocks and the data terminator constituting a NISA input deck should appear in the sequence shown in [Figure 5.1](#), namely:

- ❑ Executive commands data block
- ❑ Model data block
- ❑ Analysis data block
- ❑ Data terminator

A brief description of each data block and its general format are as follows:

1. Executive Commands Data Block

This is the first data block in a typical NISA input deck. It consists of command cards which specify general control parameters for the execution of the program, such as: type of analysis, restart option, post-processing files, etc. The syntax for a typical executive command is:

Command name = option or data

where, the left hand side (Command name) is a character string, and the right hand side (option or data) is an applicable option(s) or data for the specific command.

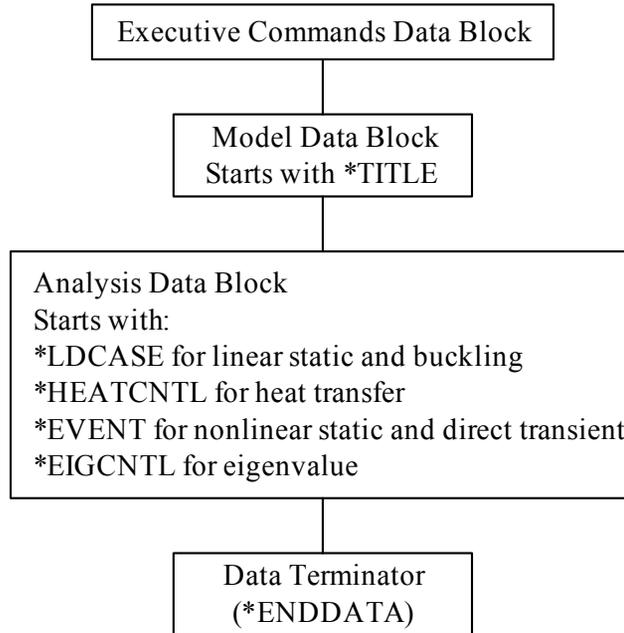


Figure 5.1: NISA Input deck setup for a typical analysis

Detailed descriptions of the executive commands for linear and nonlinear static, direct transient, eigenvalue, buckling, and steady state and transient heat transfer analyses are given in [Section 5.3](#). The executive commands for modal dynamic analyses, namely transient dynamics, random vibration, frequency response, and shock spectrum, are explained in [Chapter 8](#). For quick reference, an alphabetical list of all executive commands available in the program, including executive commands for modal dynamic analyses, are given in [Table 5.3](#).

2. Model Data Block

The model data block describes the model characteristics of the structure, e.g., coordinates, connectivities, material properties, etc. It consists of a *TITLE card(s), followed by distinct data groups that are arranged arbitrarily to form the block. The *TITLE data group should be the first group in the model data in order to enable printing the title of the problem on every page of the printout. Each data group consists of a group identification card which has to be the first card in the group, followed by the pertinent free format data of the group. For example, all nodal point coordinates are defined in one

data group which is preceded by its group identification card, *NODES. Detailed descriptions of all the groups of the model data block is given in [Chapter 6](#).

3. Analysis Data Block

The analysis data block describes data pertinent to various analysis types, e.g., loading, kinematic boundary conditions, load combinations, etc. Print controls and specific output features are also described in the analysis data. It should be noted that coupled displacements, rigid links and multipoint constraint equations are considered as model characteristics and they are specified in the model data block. Specified zero and non-zero displacements are considered as specified kinematic boundary conditions for a particular degree of freedom and they are defined in the analysis data block.

The analysis data block consists, in general, of a delimiter data group followed by distinct data groups that can be arranged arbitrarily to form the block. The delimiter data group is *LDCASE for linear static, and buckling, *EIGCNTL for eigenvalue, *EVENT for nonlinear static and direct transient dynamic, *FREQCNTL for direct frequency response or *HEATCNTL for heat transfer. Each data group, including the delimiter group, consists of a group identification card that has to be the first card in the group, followed by the pertinent free format data of the group. For example, all element pressure data is defined in one data group which is preceded by its group identification card, *PRESSURE. Detailed descriptions of all the groups of the analysis data block are given in [Chapter 7](#). The analysis data groups for modal dynamic analyses are explained in [Chapter 8](#).

4. Data Terminator

The input data terminator signals the end of the data deck. The *ENDDATA group identification card represents the data deck terminator which must be the last card in the input data deck.

5.2 Free Format Input Features

1. Overview and Comment Cards

NISA input is designed in a convenient and user friendly free format with various automatic input generation features. As discussed above, the input data is composed of main blocks, each of which consists of distinct data groups. Data groups are identified by a unique group identification card which must be the first card in the group, followed by the pertinent free format data of the group. The following sub-sections describe the general rules and guidelines for the use of the free format input.

Comment card(s) may be inserted anywhere in a NISA deck. Any card starting with two asterisks(**) is a comment card, e.g.:

** Kinematic boundary conditions

Comment cards are for the user's reference only and they are ignored by the program.

2. Group Identification Cards

A group identification card must precede the free format data of a typical data group. The general format of the group identification card is as follows:

*group name, parameter = data

where the 'group name' is a character string constituting a valid group name, the 'parameter' is a pertinent group feature, and the 'data' is a value assigned to the parameter. For upward compatibility with previous NISA versions, prior to version 87.1, the 'old' group identification format (an asterisk followed by a letter followed by a number) is still acceptable by the program. Users are encouraged to switch to the new format presented here, since the old one will not be supported in future versions. A list of all the new group identification names and their corresponding old ones is given in the 'Data Group Correspondence Tables' section.

Group identification names may be abbreviated to a minimum number of unique characters. If names are not abbreviated, however, they must be spelled out correctly to any number of characters specified between the minimum abbreviation form and the full form. For example, the *ELEMENT group identification name may be abbreviated to *ELEM, *ELEME, or *ELEMEN, but *ELEMZ or *ELEMENV are not acceptable. Embedded blanks within group identification names are not allowed. A list of all available group identification names, minimum abbreviations, description, and applicable analysis types for the model data block and the analysis data block is given in [Table 5.1](#) and [Table 5.2](#) respectively. For quick reference, an alphabetical list of all groups available in the program, including groups for modal dynamic analyses, is given in [Table 5.4](#).

3. Data Cards

The data for each group follows the appropriate group identification card. The precise content of each data group is detailed in [Chapter 6](#) and [Chapter 7](#). In what follows, we provide some general rules and/or restrictions applicable to all data cards:

- ❑ Leading, trailing, and embedded blanks are allowed anywhere in a card. A blank card is totally ignored and does not imply null entries.
- ❑ A comma (,) must be entered between two successive entries. A repeated comma (,,) implies a null entry. A comma after the last entry in a card is optional.
- ❑ A slash (/) serves as a horizontal repetition character, e.g., 2.5// means 2.5, 2.5, 2.5. Slashes need not be separated by commas.
- ❑ Special tab character (\$) is marked on many data cards. Inserting the tab character on a data card has the effect of supplying null entries for all variables up to the variable following the tab character. For example, in the data for the *NODES data group, there is a tab character (\$) between the 4th and 5th entry on the card. Thus the following entries are equivalent:

1 \$ 1.0, 1.0, 0.0 and

1, 0, 0, 0, 1.0, 1.0, 0.0

A comma before or after a tab character is optional.

4. Input File Switching (Input Re-direction, *READ)

The input data can be re-directed at any point in the data deck, from the primary input file to a secondary file by using the “*READ, filename” command. The subsequent data will be read from the secondary file until the end of file is reached, then the input will be automatically re-directed to the primary file (This feature is not available in modal dynamic analyses). Only one level of file switching is allowed. That is, the file re-direction command must be entered in the primary file only. An example is shown below:

<u>Primary File (FILE1)</u>	<u>Secondary File (FILE2)</u>
*TITLE	*RCTABLE
Example for file switching	1, 8
*ELTYPE	0.1////////
1, 40, 1	
*READ, FILE2	

..
..
..

5. Input Data Echo (*ECHO)

The echo of the input data may be suppressed or printed (part or all) by using the *ECHO = ON or *ECHO = OFF commands. The *ECHO command may appear anywhere in the data deck. However, it will be in effect for the data group(s) succeeding it. This command may be used as many times as needed to switch the echo on or off.

Table 5.1: Valid group identification names for the *model* data block

Section No. and Group ID ⁽¹⁾	Description	Applicable Analysis Types ⁽²⁾
6.2 <i>Title</i>		
6.2.1 *TITLE	Problem title	ALL1
6.3 <i>Element Data</i>		
6.3.1 *ELTYPE	Element type selection	ALL1
6.3.2 *ELEMENTS	~	ALL1
6.3.3 *E1	Alternate element definition	ALL1
6.3.4 *RCTABLE	Real constant table	ALL1
6.3.5 *LAMANGLE	Rotation angles for composites	ST, EV, BU, LT, NL, NT, DF
6.3.6 *LAMSQ2	Composite lamination sequence	ST, EV, BU, LT, NL, NT, DF
6.3.7 *BMDATA	miscellaneous data for nonlinear beam element	ST, EV, BU, LT, NL, NT, DF
6.3.8 *BMSECT	Cross section data for nonlinear beam element	ST, EV, BU, LT, NL, NT, DF
6.3.9 *NLSPRING	Force-deflection data for nonlinear springs	NL, NT
6.4 <i>Nodal Data</i>		
6.4.1 *LCSYSTEM	Local coordinate system	ALL1

Input Setup and Executive Commands

Free Format Input Features

Section No. and Group ID ⁽¹⁾	Description	Applicable Analysis Types ⁽²⁾
6.4.2 *NODES	Nodal coordinates	ALL1
6.4.3 *G2	Alternate local system definition	ST, EV, BU, NL, LT, NT, DF
6.5 <i>Material Data</i>		
6.5.1 *MATERIAL	Material property data	ST, EV, BU, NL, LT, NT, DF
6.5.2 *MATDIR1	Ortho. material axes at nodes	ALL1
6.5.3 *MATDIR2	Ortho. material axes at elements	ALL1
6.5.4 *MATHEAT	Heat transfer material properties	SH, TH
6.5.5 *HYPEREL	Hyperelastic material properties	NL, NT
6.5.6 *PLASTIC	Elastoplastic material properties	NL, NT
6.5.7 *APLASTIC	Anisotropic plastic material properties	NL, NT
6.5.8 *CREEP	Creep material properties	NL
6.5.9 *PDAMPING	Viscous damping properties	LT, NT, DF
6.6 <i>Kinematic Constraints</i>		
6.6.1 *RIGLINK	Rigid element data	ST, EV, BU, NL, LT, DF
6.6.2 *MPCEQN	Multi-point constraints	ST, EV, BU, NL, LT, DF
6.6.3 *CPDISP	Coupled displacement data	ST, EV, BU, NL, LT, DF
6.6.4 *CPTEMP	Coupled temperature data	SH, TH
6.7 <i>Miscellaneous Data</i>		
6.7.1 *SETS	Definition of sets	ALL1
6.7.2 *VECTORS	Definition of vectors	ST, EV, BU, LT, NL, NT, DF
6.7.3 *TEMPFN	Temperature curves	SH, TH
6.7.4 *TIMEAMP	Time-amplitude curves	NL, TH, LT, NT
6.7.5 *PCHANGE1	Phase change data	TH

Section No. and Group ID⁽¹⁾	Description	Applicable Analysis Types⁽²⁾
6.7.6 * CRACKNODES	Definition of Crack mouth and Crack front nodes.	ST, NL
6.7.10 * FREQFUNCTION	Frequency function data	DF
6.8 <i>Surface Contact Data</i>		
6.8.1 * CSURFACE	Definition of contact surfaces	NL
6.8.2 * CELEMENTS	Definition of contact elements	NL

(1) **Acceptable minimum abbreviations are in bold face**

- (2) ST: Linear static EV: Eigen value BU: Buckling
 NL: Nonlinear static SH: Steady state heat transfer TH: Transient heat transfer
 LT: Linear direct transient NT: Nonlinear direct transient DF: Direct frequency response
 ALL1: All analyses (ST, EV, BU, NL, LT, NT, SH, TH, DF)

Table 5.2: Valid group identification names for the *analysis* data block

Section No. and Group ID⁽¹⁾	Description	Applicable Analysis Types⁽²⁾
<i>7.1 Analysis Control Data</i>		
7.1.1 *LDCASE	Static load case control	ST, BU
7.1.2 *EIGCNTL	Eigenvalue analysis control	EV, BU
7.1.3 *EVENT	~	NL, LT, NT
7.1.4 *FREQCNTL	Direct frequency analysis control	DF
7.1.5 *HEATCNTL	Heat transfer analysis control	SH, TH
7.1.6 *LDCOMB	Load combination	ST
7.1.7 *LCTITLE	Load case title	ST, BU, NL, LT, NT
7.1.8 *MODEOUT	Mode selection for output	EV, BU
7.1.9 *TIMEINTEG	Time integration parameters	TH
7.1.10 *STEP	User specified time steps	NL, TH, NT
<i>7.2 Fourier Analysis Data</i>		
7.2.1 *FRCNTL	Fourier analysis control	ST
7.2.2 *FRCOEF	Fourier coefficients	ST
7.2.3 *ANGSEC	Angular section for response	ST
<i>7.3 Boundary Conditions and Loading</i>		
7.3.1 *SPDISP	Specified displacements	ST, EV, BU, NL, LT, NT, DF
7.3.2 *CFORCE	Concentrated nodal forces	ST, BU, NL, LT, NT, DF
7.3.3 *CFOLLOWER	Follower concentrated forces	NL, NT
7.3.4 *PRESSURE	Pressure loads	ST, BU, NL, LT, NT, DF
7.3.5 *BEAMLOAD	Beam loads	ST, BU, LT, NT, DF
7.3.6 *BODYFORCE	Body forces	ST, BU, NL
7.3.7 *NDTEMPER	Nodal temperatures	ST, BU, NL

Section No. and Group ID⁽¹⁾	Description	Applicable Analysis Types⁽²⁾
7.3.8 *NDTEMPDIF	Nodal temperature differences	ST, BU, NL
7.3.9 *INITIAL	Initial conditions	LT, NT
7.3.10 *TBODYFORCE	Body forces for transient dynamics	LT, NT
7.3.11 *L1	Alternate form for pressure loads	ST, BU
7.3.12 * MASTER	To specify master degrees of freedom for Guyan reduction	EV, BU
7.3.13 * EDGFORCE	Edge force for shell element data group	ST, BU, NL, LT, NT, DF
<i>7.4 Boundary Conditions and Loading for Heat Transfer</i>		
7.4.1 *INITEMP	Initial temperatures for transient	TH
7.4.2 *SPTEMP	Specified nodal temperatures	SH, TH
7.4.3 *CFLUX	Concentrated nodal fluxes	SH, TH
7.4.4 *DFLUX	Distributed heat fluxes	SH, TH
7.4.5 *ELHEATGEN	Heat generation at elements	SH, TH
7.4.6 *NDHEATGEN	Heat generation at nodes	SH, TH
7.4.7 *CONVBC	Convection boundary conditions	SH, TH
7.4.8 *RADBC	Radiation boundary conditions	SH, TH
7.4.9 *RADSURFACE	Surface radiation data	SH, TH
<i>7.5 Output Control Data</i>		
7.5.1 *EIGOUT	Eigenvalue output control	EV, BU
7.5.2 *NLOUT	Event output control	NL, LT, NT, DF
7.5.3 *PRINTCNTL	Selective printout control	ALL1
7.5.4 *REGIONS	Regions for stress printout	ST, EV, BU, NL, NT
7.5.5 *STRSFILTER	Nodal stress filtering data	ST, EV, BU, NL, NT
7.5.6 *SFDCOMP	Stress filtering for composites	ST, EV, BU

Input Setup and Executive Commands

Free Format Input Features

Section No. and Group ID ⁽¹⁾	Description	Applicable Analysis Types ⁽²⁾
7.5.7 * TEMPHISTORY	Temperature history	TH
7.5.8 * TEMPOUT	Time steps for transient output	TH
7.5.9 * HISTOUT	Time history output	LT, NT, DF
7.5.10 * I5	Alternate form for printout	ST, EV, BU, NL
7.5.11 * N5	Alternate form for stress printout	ST, EV, BU, NL
7.5.12 * POSTCNTL	Selective element post data control	ST, BU
7.6 <i>Data Deck Terminator</i>		
7.6.1 * ENDDATA	Input data terminator	ALL1

(1) **minimum abbreviations are in bold face**

(2) ST: Linear static EV: Eigen value BU: Buckling
NL: Nonlinear static SH: Steady state heat transfer TH: Transient heat transfer
LT: Linear direct transient NT: Nonlinear direct transient DF: Direct frequency response
ALL1: All static and heat transfer analyses (ST, EV, BU, NL, LT, NT, SH, TH, DF)

Table 5.3: Alphabetical list of all executive commands for *all* analyses

Executive Command Name⁽¹⁾	Description	Applicable Analysis Types⁽²⁾
ACUTOFF	Range of model coupling	RA
ANALYSIS	Specify analysis type	ALL1, ALL2
AUTO CONSTRAINT	Spurious normal rotation constraint	ST, EV, BU, NL, LT, NT, DF
BLANK COMMON	Blank common storage unit	ALL1, ALL2
CONMETHOD	Method of elimination in wavefront solution of contact problems	NL
CFREQ	Cut-off Frequency for missing mass correction	SHO
CTAL	Selection of contact algorithm	NL
DAMPING	Type of damping	ALL2
DELT TIME	Time increment	TR
DIRECTION	Directional superposition procedure	SHO
EIGEN EXTRACTION	Eigenvalue extraction method	EV, BU
ELEMENT ECHO	Echo element input	ALL1
EXECUTION	Select execution/checking run	ALL1, ALL2
ENDTIME	Ending time	TR
FILENAME	Specify NISA file prefix	ALL1, ALL2
FLOWER	Lower limit of exciting frequency	RA, FR
FRQRDF	Reduction factor for exciting frequency point generation	RA, FR
FSMALL	Frequency cut-off to identify rigid body modes	ALL2
FSIZ	File size for wavefront solver	ST, EV, BU, NL, LT, NT
FUPPER	Upper limit of exciting frequency	RA, FR

Input Setup and Executive Commands

Free Format Input Features

Executive Command Name⁽¹⁾	Description	Applicable Analysis Types⁽²⁾
GENFREQUENCY	Control generation of exciting frequency points	RA, FR
GEOM PROPERTIES	Compute and print geometric properties for the model	ST, EV, BU, NL, LT, NT, DF
GFACTOR	Acceleration due to gravity	ALL2
IDNUMBER	Selection of large/regular input ID Number format	ALL1
INITIAL TEMPERATURE	Define initial temperature	TH
INPOLATION	Type of interpolation	RA, FR, SHO
INPHASE	Type of interpolation for phase spectra	FR
INTEGRATION	Type of integration procedure	RA
MASS FORMULATION	Mass or heat capacitance formulation	EV, TH, LT, NT, DF
MAXCPU TIME	Limit of CPU seconds	NL, NT
MRESPONSE	Control printout of modal responses	FR
NLTYPE	Specify nonlinear static analysis type	NL, NT
NODE ECHO	Echo node input	ALL1
NONSTATIONARY	Non-stationary data	RA
PATH	Directory for NISA temporary files	ALL1, ALL2
PAT1	Directory for specific NISA files	ALL1, ALL2
PAT2	Directory for specific NISA files	ALL1, ALL2
PAT3	Directory for specific NISA files	ALL1, ALL2
PAT4	Directory for specific NISA files	ALL1, ALL2
ORTHOTROPIC	Orthotropic direction definition	SH, TH
REFCONFIG	Specify original or updated reference configuration	NL, NT
RESEQUENCE	Element resequencing	ALL1

Executive Command Name⁽¹⁾	Description	Applicable Analysis Types⁽²⁾
RESTART	Restart options	ALL1
SAVE FILE	Save specific NISA files	ALL1, ALL2
SOLV	Selection of equation solver	ST, NL, LT, NT, SH, TH, DF
SORT STRESS	Sort highest stresses	ST
SRAD	Surface radiation data	SH, TH
STARTING TIME	Starting time	TR
STEP	Total number of steps for heat transfer analysis	TH
TSEISMIC	Time duration for seismic event	SHO
UFIJ	View factor and/or radiation control	SH, TH
WARNING	Set warning flag	ALL1, ALL2

(1) Acceptable minimum abbreviations are in bold face

- (2) ST: Linear static EV: Eigen value BU: Buckling
 NL: Nonlinear static SH: Steady state heat transfer TH: Transient heat transfer
 TR: Modal transient dynamics FR: Frequency response RA: Random vibration
 SHO: Shock spectrum LT: Linear direct transient NT: Nonlinear direct transient
 DF: Direct frequency response
- ALL1: All static, heat transfer, and linear direct transient analyses (ST, EV, BU, NL, LT, NT, SH, TH, DF)
 ALL2: All modal dynamic analyses (TR, FR, RA, SHO)

Table 5.4: Alphabetical list of *all* data groups (model and analysis data blocks) for *all* analyses

Group ID⁽¹⁾	Section No.	Description	Application Analysis Types⁽²⁾
*ADDFREQUENCY	8.5.4	Additional exciting frequencies	RA, FR
*ANGSEC	7.2.3	Angular section for response	ST
*APLASTIC	6.5.7	Anisotropic plastic material properties	NL, NT
*ARRIVALTIME	8.6.1	Arrival time data	TR
*BEAMLOAD	7.3.5	Beam loads	ST, BU, NL, NT
*BMDATA	6.3.7	General data for 3D general beam element	ST, BU, EV, NL, LT, NT, DF
*BMSECT	6.3.8	Cross section data for 3D general beam element	ST, BU, EV, NL, LT, NT, DF
*BODYFORCE	7.3.6	Body forces	ST, BU, NL, NT
*CELEMENTS	6.8.2	Contact element definition	NL
*CFLUX	7.4.3	Concentrated nodal fluxes	SH, TH
*CFOLLOWER	7.3.3	Follower concentrated forces	NL, NT
*CFORCE	7.3.2	Concentrated nodal forces	ST, BU, NL, LT, NT,DF
*CONVERGENCE	7.1.11	Convergence criteria	NL
*CONVBC	7.4.7	Convection boundary conditions	SH, TH
*CORRELATION	8.6.4	Correlation ensemble data	RA
*CPDISP	6.6.3	Coupled displacement data	ST, EV, BU, NL, LT,DF
*CPTEMP	6.6.4	Coupled temperature data	SH, TH
*CREEP	6.5.8	Creep material properties	NL
*CSURFACES	6.8.1	Contact surface definition	NL
*DAMPING	8.5.2	Damping value	ALL2

Group ID⁽¹⁾	Section No.	Description	Application Analysis Types⁽²⁾
*DCFORCE	8.7.3	Concentrated nodal force data	TR, RA, FR
*DFLUX	7.4.4	Distributed heat fluxes	SH, TH
*DPRESSURE	8.7.4	Pressure loading data	TR, RA, FR
*DRIVER	8.7.5	Driver node data	FR
*EIGCNTL	7.1.2	Eigenvalue analysis control	EV, BU
*EIGOUT	7.5.1	Eigenvalue output control	EV, BU
*ELEMENTS	6.3.2	Element definition	ALL1
*ELHEATGEN	7.4.5	Heat generation at elements	SH, TH
*ELTYPE	6.3.1	Element type selection	ALL1
*ENDDATA	7.6.1, 8.9.1	Input data terminator	ALL1, ALL2
*ENVELOPE	8.6.8	Envelope function data	RA
*EVENT	7.1.3	Event control	NL, LT, NT
*FREQCNTL	7.1.4	Direct frequency analysis control	DF
*FREQFUNCTION	6.7.10	Frequency function data	DF
*FLOORRESPONSE	8.5.7	Floor Response Spectra Data	TR
*FRCNTL	7.2.1	Fourier analysis control	ST
*FRCOEF	7.2.2	Fourier coefficients	ST
*FRSOUT	8.8.8	Floor Response Spectra Output	TR
*GROUND	8.7.2	Ground motion data	ALL2
*HEATCNTL	7.1.5	Heat transfer analysis control	SH, TH
*HISTORY	8.8.2	Response time-history request data	TR
*HISTOUT	7.5.9	Time history output	LT, NT, DF
*HYPEREL	6.5.5	Hyperelastic material properties	NL, NT

Input Setup and Executive Commands

Free Format Input Features

Group ID⁽¹⁾	Section No.	Description	Application Analysis Types⁽²⁾
*INITEMP	7.4.1	Initial temperature for transient heat	TH
*INITIAL	7.3.9, 8.5.5	Initial displacement and velocity	TR, LT, NT
*LAMANGLE	6.3.5	Rotation angles for composites	ST, EV, BU, LT, NT, DF
*LAMSQ2	6.3.6	Composite lamination sequence	ST, EV, BU, LT, NT, DF
*LCSYSTEM	6.4.1	Local coordinate system	ALL1
*LCTITLE	7.1.7	Load case title	ST, BU, NL, LT, NT
*LDCASE	7.1.1	Static load case control	ST, BU
*LDCOMB	7.1.6	Load combination	ST
*MATDIR1	6.5.2	Orthotropic material axes at nodes	ALL1
*MATDIR2	6.5.3	Orthotropic material axes at elements	ALL1
*MATERIAL	6.5.1	Material property data	ST, EV, BU, NL, LT, NT, DF
*MATHEAT	6.5.4	Heat transfer material properties	SH, TH
*MDSPECTRUM	8.6.6	Multiple damping spectrum group data	SHO
*MODEOUT	7.1.8	Mode selection for output	EV, BU
*MODESELECTION	8.5.1	Mode selection data	ALL2
*MPCEQN	6.6.2	Multi-point constraints	ST, EV, BU, NL, LT, DF
*MSEXCITATION	8.7.8	Multiple support excitation data	SHO
*MVELOAD	8.7.6	Moving load definition	TR
*MVEPATH	8.7.7	Moving load path definition	TR
*NDHEATGEN	7.4.6	Heat generation at nodes	SH, TH
*NDTEMPDIF	7.3.8	Nodal temperature difference	ST, BU, NL
*NDTEMPER	7.3.7	Nodal temperatures	ST, BU, NL

Group ID⁽¹⁾	Section No.	Description	Application Analysis Types⁽²⁾
*NLOUT	7.5.2	Nonlinear static output	NL, LT, NT, DF
*NLSPRING	6.3.9	Force-deflection data for nonlinear springs	NL, NT
*NODES	6.4.2	Nodal coordinates	ALL1
*NONSTATIONARY	8.6.7	Non-stationary data	RA
*PCHANGE1	6.7.5	Phase change data	TH
*PDAMPING	6.5.9	Viscous damping properties	LT, NT
*PLASTIC	6.5.6	Elastoplastic material properties	NL, NT
*POSTCNTL	7.5.12	Selective element post data control	ST, BU
*PRESSURE	7.3.4	Pressure load	ST, BU, NL, LT, NT, DF
*PRINTCNTL	7.5.3	Selective printout control	ALL1
*PRSPECTRA	8.6.9	Prestored response spectra	SHO
*PSDFUNCTION	8.6.3	Power spectral density function	RA
*PSDOUT	8.8.4	Response PSD request data	RA
*RADBC	7.4.8	Radiation boundary conditions	SH, TH
*RADSURFACE	7.4.9	Surface radiation data	SH, TH
*RCTABLE	6.3.4	Real constant table	ALL1
*REGIONS	7.5.4	Regions for stress printout	ST, EV, BU, NL, LT, NT
*RESPONSE	8.8.7	Maximum response request data	SHO
*RIGLINK	6.6.1	Rigid element data	ST, EV, BU, LT
*RMSOUT	8.8.5	RMS response request data	RA
*RSET	8.8.1	Response set data	TR, RA, FR
*SETS	6.7.1	Definition of sets	ALL1
*SFDCOMP	7.5.6	Stress filtering for composites	ST, EV, BU, LT, NT

Input Setup and Executive Commands

Free Format Input Features

Group ID⁽¹⁾	Section No.	Description	Application Analysis Types⁽²⁾
*SNAPSHOT	8.8.3	Snap shot response request data	TR
*SPDISP	7.3.1	Specified displacements	ST, EV, BU, NL, LT, NT, DF
*SPECTRUM	8.6.5	Spectrum definition data	FR, SHO
*SPOUT	8.8.6	Output spectra request data	FR
*SPTEMP	7.4.2	Specified nodal temperatures	SH, TH
*STEPsize	7.1.10	User specified time steps	NL, TH, NT
*STRSFILTER	7.5.5	Nodal stress filtering data	ST, EV, BU, NL, LT, NT
*TbodyFORCE	7.3.10	Transient body force data	LT, NT
*TEMPFN	6.7.3	Temperature curves	SH, TH
*TEMPHISTORY	7.5.7	Temperature history	TH
*TEMPOUT	7.5.8	Time steps for transient output	TH
*TIMEAMP	6.7.4	Time-amplitude curves	NL, TH, LT, NT
*TIMEFUNCTION	8.6.2	Time function definition	TR
*TIMEINTEG	7.1.9	Time integration parameters	TH
*TITLE	6.2.1, 8.4.1	Problem title	ALL1, ALL2
*TSTEP	8.5.3	Time step data	TR
*VECTORS	6.7.2	Definition of vectors	ST, EV, BU, LT, NT, DF

(1) Minimum abbreviations are in bold face

- (2) ST: Linear static EV: Eigen value BU: Buckling
 NL: Nonlinear static SH: Steady state heat transfer TH: Transient heat transfer
 TR: Modal transient dynamics FR: Frequency response RA: Random vibration
 SHO: Shock spectrum LT: Linear direct transient NT: Nonlinear direct transient
 DF: Direct frequency response
- ALL1: All static, heat transfer and linear direct transient analyses (ST, EV, BU, NL, LT, NT, SH, TH, DF)
 ALL2: All modal dynamic analyses (TR, RA, FR, SHO)

5.3 Executive Commands

5.3.1 General Description

As described above, the executive commands constitute the first data block in a typical NISA input deck, and they define general control parameters for program execution. The syntax for a typical executive command is:

Command name = option or data

The command name is a character string that may be abbreviated to a minimum of first four characters, though, a well spelled-out form is also acceptable.

The option or data is a character string or a data value that will be assigned to the specific command. [Section 5.3.3](#) gives detailed description of executive commands that are applicable to specific analysis types excluding modal dynamic analysis (see [Chapter 8](#)). The listing of the commands within each section is arranged alphabetically and an overall alphabetical list is given in [Table 5.5](#). The following general notes apply to all executive commands:

- ❑ Embedded blanks in executive commands are ignored.
- ❑ Command names may be abbreviated. Minimum abbreviations are shown in bold face letters.
- ❑ Various command options are shown between square brackets. Default values of options are shown between braces.

5.3.2 Executive Commands Common to all Analyses

ANALYSIS -Specify analysis type

ANALYSIS = $\left[\begin{array}{c} \{\text{STATIC}\} \\ \text{BUCKLING} \\ \text{EIGENVALUE} \\ \text{NLSTATIC} \\ \text{LTRANSIENT} \\ \text{NLTRANSIENT} \\ \text{SHEAT} \\ \text{THEAT} \end{array} \right]$

where,

- STATIC : indicates linear static analysis
- BUCKLING : indicates buckling analysis to obtain buckling load factors and mode shapes
- EIGENVALUE : indicates eigenvalue analysis to extract natural frequencies and mode shapes
- NLSTATIC : indicates nonlinear static analysis including material and/or geometric nonlinearities
- LTRANSIENT : indicates linear direct transient analysis
- NLTRANSIENT : indicates nonlinear direct transient analysis
- SHEAT : indicates steady state heat transfer analysis
- THEAT : indicates transient heat transfer analysis

Table 5.5: Alphabetical list of executive commands for all analyses except modal dynamic analyses

Executive Command Name⁽¹⁾	Description	Applicable Analysis Type⁽²⁾
ANALYSIS	Specify analysis type	ALL1
AUTO CONSTRAINT	Spurious normal rotation constraint	ST, EV, BU, NL, LT, NT
BLANK COMMON	Blank common storage unit	ALL1
CONMETHOD	Method of elimination in wavefront solution of contact problems	NL
CTAL	Selection of contact algorithm	NL
EIGEN EXTRACTION	Eigenvalue extortion method	EV, BU
ELEMENT ECHO	Echo element input	ALL1
EXECUTION	Select execution/checking run Specify	ALL1
FILE NAME	NISA file prefix	ALL1
GEOM PROPERTIES	Compute and print geometric properties of the model	ST, EV, BU, NL, LT, NT
IDNUMBER	Selection of large/regular input ID Number format	ALL1
INITIAL TEMPERATURE	Define initial temperature	TH
MASS FORMULATION	Mass or heat capacitance formulation	EV, LT, TH, NT
MAXCPU TIME	Limit of CPU seconds	NL, NT
NLTYPE	Specify nonlinear static analysis type	NL, NT
NODE ECHO	Echo node input	ALL1
PATH	Directory for NISA temporary files	ALL1
PAT1	Directory for specific NISA files	ALL1
PAT2	Directory for specific NISA files	ALL1
PAT3	Directory for specific NISA files	ALL1

Input Setup and Executive Commands

Executive Commands

Executive Command Name⁽¹⁾	Description	Applicable Analysis Type⁽²⁾
PAT4	Directory for specific NISA files	ALL1
ORTHOTROPIC	Orthotropic direction definition	SH, TH
REFCONFIG	Specify original or updated reference configuration	NL, NT
RESEQUENCE	Element resequencing	ALL1
RESTART	Restart options	ALL1
SAVEFILE	Save specific NISA file	ALL1
SIGMA	Stefan-Boltzmann coefficient	SH, TH
SOLV	Selection of equation solver	ST, NL, NT, SH, TH
SOLVER	Select equation solver	ALL
SORT STRESS	Sort highest stresses	ST
STEP	Total number of steps for heat transfer analysis	TH
THREADS	Select number of OpenMP threads	ALL
UFIJ	View factor and/or radiation control	SH, TH
WARNING	Set warning flag	ALL1

BLANK COMMON - Blank common storage limit

$$\text{BLANK COMMON} = \left[\begin{array}{c} n \\ \{ 50000 \} \end{array} \right]$$

where,

- n: is an integer specifying the blank common storage limit. This limit is *only* used for dynamic memory allocation during reading and partially processing the input data

ELEMENT ECHO - Echo element input
$$\text{ELEMENT ECHO} = \begin{bmatrix} \{\text{ON}\} \\ \{\text{OFF}\} \end{bmatrix}$$

where,

ON : element input echo is on

OFF : element input echo is off

Users are advised to use the more general *ECHO command ([Section 5.2](#)) since this command will not be supported in future versions.

EXECUTION -Select execution / checking run
$$\text{EXECUTION} = \begin{bmatrix} \{\text{GO}\} \\ \text{CHECK} \\ \text{CGO} \end{bmatrix}$$
FILE NAME - Specify NISA file prefix
$$\text{FILE NAME} = \begin{bmatrix} \text{fname} \\ \{\text{no default}\} \end{bmatrix}$$

where,

fname: is an alphanumeric character string that will be used as a file name prefix followed by the NISA file number to be saved. Up to 64 characters may be used (system dependent).

This command must be used in conjunction with the SAVE FILE command which specifies the NISA file numbers to be saved (e.g., 26, 27,...).

Note:

1. For certain computer systems, the NISA files to be saved are actually specified by the Job Control Language (JCL) and this command is not needed. Refer to pertinent system manual for details.

FILE SIZE -Specify the size for wavefront solver

$$\mathbf{FSIZ} = \left[\begin{array}{c} n \\ \{2000\} \end{array} \right]$$

where,

- n: is a integer number to specify the size of each file for wavefront solver. The unit of n is mega bytes (10^6 bytes). File 30 is the first file used for wavefront solver. If more than one file is needed, nine more files (file 61 to 69) can be used for wavefront solver.

GEOM PROPERTIES -Compute and print geometric properties of the model

$$\mathbf{GEOM\ PROPERTIES} = \left[\begin{array}{c} \{ON, LIST\} \\ ON, NOLIST \\ OFF \end{array} \right]$$

where

- ON, LIST : the ON option activates computation and saving geometric properties of the model on file 26. These are the total volume, mass, the center of gravity and the moments of inertia with respect to global and center of gravity axes. The LIST option prints the geometric properties in the output file.
- ON, NOLIST : compute geometric properties of the model and save it on file 26. No geometric properties are printed in the output file.
- OFF : suppress the computation and printing of the geometric properties.

Note:

1. This command is not applicable for heat transfer analysis. For geometric nonlinearity analysis, this command is only applicable to computations for the undeformed geometry.

$$\mathbf{IDNUMBER} = \begin{bmatrix} \{\text{REGULAR}\} \\ \text{LARGE} \end{bmatrix}$$

where,

REGULAR : Regular input format

LARGE : Large input format

NODE ECHO -Echo node input

$$\mathbf{NODE ECHO} = \begin{bmatrix} \{\text{ON}\} \\ \text{OFF} \end{bmatrix}$$

where,

ON : node input echo is on

OFF : node input echo is off

Users are advised to use the more general *ECHO data group ([Section 5.2](#)) since this command will not be supported in future versions.

PATH - Directory for temporary files

$$\mathbf{PATH} = \begin{bmatrix} \text{Pname} \\ \{\text{no default}\} \end{bmatrix}$$

where,

pname: is a character string that will be used as a directory for NISA temporary files. Up to 64 characters may be used (system dependent). The directory name should include the last character just before the file name. For example, a directory \USER\NISA\TMP in DOS operating system should be given as \USER\NISA\TMP\. The last back slash is required.

Note:

1. NISA temporary file will be open in user's current working directory if this command is not used.

PAT1

PAT2

PAT3

PAT4 - Directory for NISA files

PAT1 = $\left[\begin{array}{l} \text{pname, n1, n2, ...} \\ \{ \text{nodefault} \} \end{array} \right]$

PAT2

PAT3

PAT4

where,

pname: is a character string that will be used as a directory for NISA files n1, n2, etc. Up to 64 characters may be used (system dependent). The directory name should include the character just before the file name. For example, a directory \USER\NISA\TMP in DOS operating system should be given as \USER\NISA\TMP\. The last back slash is required.

n1, n2,...: NISA file numbers which should be open in pname directory. These four commands allow the user to open files in four different directories other than the current working directory during a NISA run. If files are saved files, these files will be saved under the pname directory.

REPL – Specify file names for user input element

REPL = skfile, smfile, scfile

where,

Skfile: file name for user input element stiffness matrices

Smfile: file name for user input element mass matrices

Scfile: file name for user input element damping matrices

RESEQUENCE-Element resequencing

$$\mathbf{RESEQUENCE} = \begin{bmatrix} \{\text{ON}\}, \text{NOKC}, \text{MAX}, \text{LIST} \\ \text{GEOM} \\ \text{BOTH} \\ \text{OFF} \end{bmatrix}$$

where,

ON : use topology scheme for wavefront minimization.

GEOM : use geometric sorting scheme for wavefront minimization.

BOTH : use both geometric sorting and topology schemes for wavefront minimization.

NOKC : do not consider kinematic constraints (multi-point constraint equations, rigid links, and coupled displacements) during the resequencing process. This option should be exercised with caution. The actual wavefront parameters may be *higher* than predicted. The *default* option is to *include* the kinematic constraints effect during the resequencing process. The predicted wavefront parameters are always upper bounds to the actual parameters in this case.

MAX : select the trial that gives the least maximum wavefront. The *default* option is to select the trial that gives the least root-mean-square (RMS) wavefront. This option may be used if the available memory is the determining factor. In most cases, the trial that gives the least RMS wavefront also gives the least maximum wavefront.

- LIST : output a table of correspondence between the element IDs and the internal numbering sequence. The default option is not to provide this table.
- OFF : do not resequence the elements

The above options can be in any order. The element resequencing is transparent to the user. The element IDs are not changed. All communications between the user and the program as far as input and output are concerned are in terms of the user's element identification numbers. No table of correspondence for element numbering is provided if LIST option is not specified.

The wavefront minimization schemes select the trial with the least RMS wavefront as a default option. The topology scheme works better in most of the cases and takes into account the effect of kinematic constraints. However, the geometric sorting is a faster minimization scheme.

Sparse matrix solver resequences equations automatically inside the solver. Wavefront minimization is not needed, it is better to set RESE = OFF to cut down unnecessary computation.

Iterative solver will benefit from equation resequencing. RESE = GEOM is recommended for iterative solver.

RESTART -Restart options

$$\text{RESTART} = \left[\begin{array}{l} n, \text{NOLIST}, \text{NEW30}, \text{USE30} \\ \{0\}, \{\text{LIST}\}, \{\text{OLD30}\} \end{array} \right]$$

where,

- n: is an integer number designating the restart type number:
= 0 (default), no restart, the execution proceeds from scratch
> 0, n in this case should refer to a valid restart type number as described in [Section 5.5](#) and briefed in [Table 5.11](#) and [Table 5.12](#).

Note:

The restart type number, n , must be given before any other option (e.g., NOLIST), if any. The second and third entries may appear in any order. Also the entire command may be omitted for fresh (or scratch) run since this is the default option.

NOLIST: suppress the printout which lists the parameters of the previous run. The list provides information about the model and the analysis data, e.g., number of elements and nodes, number of MPCs, number of load cases, wavefront parameters, types of loading, post-data saved, etc. The default option, {LIST}, is to list the restart information.

Note:

This list is not provided in heat transfer restarts.

** The following options are only significant for restart 3 in linear static analysis and restart 5 in linear direct transient dynamics. Refer to [Section 5.5](#) for more details. **

OLD30: (default option).

- For static analysis, the decomposed stiffness matrix in file 30 and files 61 to 69 from a previous static analysis run is provided. For first new load case, reuse the files if displacement boundary conditions are similar to the last new load case in prior run(s), otherwise start a new decomposition and overwrite the supplied file.
- For direct transient dynamic analysis, the decomposed effective stiffness matrix in file 30 and files 61 to 69 from a previous direct transient analysis run is provided. For first new event, reuse the file if, β, γ and Δt are the same as in the last event of the run immediately preceding; otherwise start a fresh decomposition and overwrite the supplied file.

Note:

The supplied files (30, 61 to 69) may be overwritten if a new decomposition is needed in any load case (event for direct transient dynamics).

NEW30: The decomposed stiffness matrix (in files 30, 61 to 69) from a previous run is not provided. Create a new file (scratch or permanent depending on whether the new file will be saved) and perform new decomposition for first load case (first event for direct transient dynamics) in this restart run.

Note:

The new files (30, 61 to 69) are not saved unless explicitly indicated through the command 'SAVEFILE'.

USE30: File 30 from a prior static analysis run is provided. Reuse the file for the first load case in this run regardless of whether the displacement boundary conditions are similar to the last new load case of prior runs or not. For subsequent load cases, the file may be overwritten if a new decomposition is needed.

Note:

The supplied file 30 may be overwritten if a new decomposition is needed for subsequent load cases after the first one. It is the user's responsibility to ensure that the supplied file 30 and the displacement boundary conditions of the first load case are compatible.

For direct transient dynamics, this option means that a File 30 from a previous run is provided and must be used for the first event in this restart run unconditionally.

OLD30, NEW30, and USE30 options are not applicable to NLTRANSIENT analysis.

Note: For sparse matrix solver, unit 30 still being used in NISA input data to represent the decomposed matrix and its related files. For example, users want to save the decomposed matrix for a restart run, SAVE = 30 will have saved all these files. However, the file names are different than the one (FILE30.DAT) used in the frontal solver. Sparse matrix solver uses more than one file to keep the decomposed matrix. All files with names such as FILE.PC* and FILE.casi*.sdc are files needed for a restart run. (FILE is the name given in executive control FILE and * represents a wide card character). In order to have a successful restart, all these files should be present at the time of a restart.

SAVE FILE -Save Specific NISA files

$$\text{SAVE FILE} = \left[\begin{array}{l} n1, n2, \dots \\ \{\text{no default}\} \end{array} \right]$$

where,

n1, n2, ... : NISA file numbers to be saved.

This command must be used in conjunction with the FILE NAME command which specifies a character prefix (fname) for files to be saved. Refer to [Chapter 3](#) for NISA file numbers.

Note:

1. For certain computer systems, the NISA files to be saved are actually specified by the Job Control Language (JCL) and this command is not needed. Refer to pertinent system manual for details.

SOLVER -Select equation solver

$$\text{SOLVER} = \left[\begin{array}{c} \{\text{FRON}\} \\ \text{ITER, TOLE, NITR, MPIVT, CORE} \\ \text{SPAR, MEMSP} \\ \text{PARDISO, } m, n \end{array} \right]$$

where,

- FRON : use wavefront solver. This is default
- ITER : use iterative solver
- TOLE : convergence tolerance for iterative solver default value = 1×10^{-6}
- NITR : max. number of allowable iterations for a iterative solution default value = 2500
- MPIVT : option to modify pivot, a bad element matrix to avoid negative pivot.
The iterative solver can only solve a positive definite global matrix
= 0 no modification (default)
= 1 modify negative pivot to a small positive number
= 2 change the negative pivot to a positive number
- CORE : activate the out-of-core scheme instead of the in-core scheme for the iteration process to speed up the performance of the solver on certain machines such as PC for which virtual memory I/O is slow.
= IN use in-core scheme for iteration

Input Setup and Executive Commands

Executive Commands

- = OUT use out-of-core scheme for iteration
- SPAR : use sparse matrix solver (see note 3 to 5)
- MEMSP : size of RAM (Mbytes) available to sparse matrix solver. The rule of thumb is the available RAM in the system subtract 32M. The default value on Unix system is 64, on Window systems, NISA will automatically set.
- PARDISO : use Parallel Direct Sparse Solver from Intel® MKL.
- m : = 0 in-core computation
= 1 solver will decide whether to go for in-core/out-of-core computation
= 2 out-of-core computation
- n : Peak available physical memory (Mbyte)
- THREADS : Number of threads.
= 1 to Maximum available processor's. Default maximum available processor.

Notes:

1. Iterative solver is available for linear static, nonlinear static, linear direct transient dynamics, nonlinear direct transient dynamics, and heat transfer analysis. However, if contact element is used in nonlinear analysis, frontal solver will be used unconditionally.
2. Users should be aware that iterative solver can only solve problems with positive definite equations. Any problems with non-positive definite equations due to modeling, deformation behavior, or material properties etc. should use frontal solver.
3. Sparse matrix solver is faster direct solver than the frontal solver and is available for linear static, nonlinear static, linear direct transient dynamics, nonlinear direction transient dynamics, eigenvalue analysis and buckling analysis. However, if contact element is used in nonlinear analysis, frontal solver will be used automatically.
4. Sparse matrix solver performs equation resequencing inside the solver. No wavefront minimization is needed.

WARNING -Set warning flag

WARNING = $\left[\begin{array}{l} \{GO\} \\ STOP \end{array} \right]$

where,

GO : directs the program to continue after a warning(s) is produced during input processing

STOP : program will stop if a warning is produced during input processing

5.3.3 Executive Commands Valid for Specific Analyses

AUTO CONSTRAINT -Spurious normal rotation constraint

AUTO CONSTRAINT = $\left[\begin{array}{l} \{ON, VALUE, OPTION\} \\ OFF \end{array} \right]$

Applicable analysis types: STATIC, BUCKLING, EIGENVALUE, NLSTATIC,
LTRANSIENT, NLTRANSIENT

where,

ON : activates automatic spurious normal rotation constraint for shells and composite shells

OPTION : 0 or 1
Option=1 means the value is the actual value of normal rotation stiffness to be added. Option=0 means the value.

VALUE : Value is the ratio of the rotation stiffness to the bending rotation stiffness.

OFF : deactivates automatic spurious normal rotation constraint

CONMethod -Method of elimination in wavefront solution of surface contact problems

$$\text{CONMethod} = \begin{bmatrix} n \\ \{1,2\} \end{bmatrix} \quad \text{Applicable analysis types: NLSTATIC}$$

where,

n = 1 will postpone the eliminations during the wavefront solution of degrees of freedom attached to nodes on contact surfaces until all NISA elements (with NKTP values) are processed. *This is the default option when NLTYPE = GAP is used, and it corresponds to AUTOMATIC SUBSTRUCTURING*

n = 2 Eliminations of degrees of freedom attached to nodes on contact surface depend on the *current* active set in the contact region. If a degree of freedom at a node on a contact surface is not active in contact, it may be eliminated as soon as all elements attached to it are processed.

This is the default option when the nonlinear analysis type indicated by the command NLTYPE is not GAP.

CONTACT ALGORITHM -Selection of contact algorithm

$$\text{CTAL} = \begin{bmatrix} \text{[PENA]} \\ \text{AUGM} \\ \text{LAGR} \end{bmatrix}$$

where,

PENA : Penalty method

AUGM : Augmented Lagrangian multiplier method (see [Section 3.4.1](#))

LAGR : Lagrangian multiplier method

EIGEN EXTRACTION -Eigenvalue extraction method

$$\text{EIGEN EXTRACTION} = \left[\begin{array}{c} \text{SUBSPACE, ACCELERATED} \\ \text{SUBSPACE, CONVENTIONAL} \\ \\ \text{INVERSE} \\ \text{LANCZOS} \\ \text{GUYAN} \\ \\ \text{LANCZOS, SHIFTED} \end{array} \right] \quad \begin{array}{l} \text{Applicable analysis types:} \\ \text{EIGENVALUE,} \\ \text{BUCKLING} \end{array}$$

where,

SUBSPACE, ACCELERATED : indicates accelerated subspace iteration algorithm

SUBSPACE, CONVENTIONAL : indicates conventional subspace iteration algorithm

INVERSE : indicates inverse iteration algorithm

LANCZOS : indicates Lanczos algorithm

GUYAN : Guyan reduction followed by Householder - QR iteration (master degrees of freedom can be specified in *MASTER data group)

LANCZOS, SHIFTED : Indicates shifted block Lanczos algorithm

GMAT - Global assembled stiffness, damping and mass matrices output

$$\text{GMAT} = [\text{STIFFNESS, DAMPING, MASS}] \quad \begin{array}{l} \text{Applicable analysis types: STATIC,} \\ \text{DIRECT FREQUENCY} \end{array}$$

STIFFNESS, DAMPING and MASS can be used for requesting the global assembled stiffness, damping and mass matrices respectively. These entities can be any order.

INITIAL TEMPERATURE -Define initial temperature

$$\text{INITIAL TEMPERATURE} = \left[\begin{array}{c} \text{Temp} \\ \{0.0\} \end{array} \right] \quad \text{Applicable analysis types: THEAT}$$

where,

Temp: is the initial temperature for the whole model. If initial temperature is not uniform, use the *INITEMP data group ([Section 7.4.1](#)).

MASS FORMULATION- Mass (or capacitance) formulation for eigenvalue (or heat transfer) analyses

MASS FORMULATION = $\left[\begin{array}{l} \{ \text{CONSISTENT} \} \\ \text{LUMPED} \end{array} \right]$ Applicable analysis types EIGEN-VALUE, LTRANSIENT, THEAT, NLTRANSIENT

where,

CONSISTENT : activates consistent mass (or capacitance) matrix formulation

LUMPED : activates lumped mass (or capacitance) matrix formulation

Using consistent mass matrix formulation usually gives more accurate results than lumped mass matrix formulation.

MAXCPU TIME - Limit of CPU seconds

MAXCPU TIME = $\left[\begin{array}{l} n \\ \{ 0 \} \end{array} \right]$ Applicable analysis types: NLSTATIC, NLTRANSIENT

where,

n: is an integer indicating the maximum limit of CPU seconds for the analysis, n = 0 implies unlimited CPU time.

NLTYPE -Specify nonlinear static analysis type

$$\text{NLTYPE} = \left[\begin{array}{l} \text{MATGEOM} \\ \text{MATGEOM, LARGE} \\ \text{MATERIAL} \\ \{\text{GEOMETRY}\}, \text{LARGE} \\ \text{GAP} \\ \text{LINEAR} \end{array} \right] \begin{array}{l} \text{Applicable analysis types: NLSTATIC,} \\ \text{NLTRANSIENT} \end{array}$$

where,

- MATGEOM : indicates combined material and geometrical nonlinearities
- MATERIAL : indicates material nonlinearity only
- GAP : indicates that the boundary nonlinearity represented by the node-to-node contact elements, NKTP = 49, 50, or the surface contact elements is the only type of nonlinearity considered. Other elements remain linear elastic throughout the analysis. This option is valid only if node-to-node elements, NKTP = 49, 50, or surface contact (*CSURFACES and *CELEMENTS) exist in the model. Other NLTYPE options may be used with the node-to-node or surface-to-surface contact elements.
- GEOMETRY : indicates geometrical nonlinearity only
- LINEAR : indicates linear analysis
- MATGEOM, LARGE : same as MATGEOM but with large strain capabilities.
- GEOMETRY, LARGE : same as GEOMETRY but with large strain capabilities.

Note:

1. Large strain capability is only available for 3D-solid element (NKTP = 4) in the current release. Also, when large strain analysis is opted, the program will automatically reset **REFCONFIG** to **UPDATED**, regardless of user's input.

ORTHOTROPIC -Orthotropic direction definition

ORTHOTROPIC = $\begin{bmatrix} n \\ \{0\} \end{bmatrix}$ Applicable analysis types: SHEAT, THEAT

where,

- n** : an integer indicating the method of defining orthotropic directions;
 - = 0 material is isotropic
 - = 1 orthotropic directions are defined for each node (using *MATDIR1 data group, Section 6.4.2)
 - = 2 orthotropic directions are defined for each element (using *MATDIR2 data group, Section 6.4.3)
 - = 3 orthotropic directions coincide with global directions

Users are advised to use the *ELEMENTS data group ([Section 6.3.2](#)) for the definition of this variable since this executive command will not be supported in future versions.

REFCONFIG - Specify original or updated reference configuration

REFCONFIG = $\begin{bmatrix} \{TOTAL\} \\ \{UPDATED\} \end{bmatrix}$ Applicable analysis types: NLSTATIC, NLTRANSIENT

where,

- TOTAL** : indicates original undeformed reference configuration or total Lagrangian formulation
- UPDATED** : indicates updated reference configuration or updated Lagrangian formulation

Note:

1. For some elements and/or nonlinear analysis types, one of the above formulations may not be applicable. See Element Library ([Chapter 4](#)) for details.

SIGMA-Stefan-Boltzmann coefficient

$$\text{SIGMA} = \left[\begin{array}{c} \sigma \\ \{\text{no default}\} \end{array} \right] \quad \text{Applicable analysis types: SHEAT, THEAT}$$

where,

σ : is the Stefan-Boltzmann coefficient for problems with radiation boundary conditions, and/or surface radiation.

SOLVER -Select equation solver

$$\text{SOLVER} = \left[\begin{array}{c} \{\text{FRON}\} \\ \text{ITER, TOLE, NITR, MPIVT, CORE} \end{array} \right]$$

where,

{FRON} : use wavefront solver. This is default

ITER : use iterative solver

TOLE : convergence tolerance for iterative solver default value = 1×10^{-6}

NITR : max. number of allowable iterations for a iterative solution default value = 2500

MPIVT : option to modify pivot, a bad element matrix to avoid negative pivot.

The iterative solver can only solve a positive definite global matrix.

= 0 no modification (default)

= 1 modify negative pivot to a small positive number

= 2 change the negative pivot to a positive number

SORT STRESS - Sort highest stresses

$$\text{SORT STRESS} = \left[\begin{array}{c} n \\ \{n= 25\} \end{array} \right] \quad \text{Applicable analysis types: STATIC}$$

where,

n: is the number of highest nodal and element centroidal stresses to be sorted in descending order.

SRAD - Analysis with surface radiation

$$\text{SRAD} = \left[\begin{array}{c} \text{ON, IOBST, NAXI, NDIV, } \alpha \\ \{ \text{OFF} \} \end{array} \right]$$

where,

IOBST : = 1, Third surface obstruction and self-obstruction checks will be made for view factor calculation.

= 0, Third surface obstruction checks will be skipped. However, self-obstruction checks will still be made.

NAXI : Number of divisions along the circumference. This value will be made for view factor calculation.

NDIV : Number of sub-divisions of a surface for view-factor calculation

α : Relaxation parameter for radiative heat flux ($0 < \alpha < 1$)

Notes:

1. Whenever possible, the user should use IOBST = 0. This leads to substantial savings of CPU time for view factors calculation.
2. If NAXI is specified to be less than 8, NISA/HEAT will set this value equal to 8. If NAXI is specified to be a very large value (say 9999), it will be reset to the maximum allowable value based on available memory.

3. For 2-D problems (NKTP = 102) having third surface obstructions, the user can instruct NISA/HEAT to subdivide a radiation surface into “NDIV” number of divisions for view factor computation. This leads to more accurate values of view-factors. However, CPU time will also be higher. Maximum allowable value for NDIV is equal to 10. There is no provision to subdivide a surface for 3D-problems.
4. Relaxation parameter for radiative flux should lie between 0 and 1 ($0 < \alpha < 1$). A small value of α can slow down the rate of convergence and a high value of α may cause solution to diverge. Values between 0.1 and 0.3 are recommended.
5. Unless specified by the user, the following default values will be used; IOBST = 1, NAXI = 32, NDIV = 1 and $\alpha = 0.2$.

STEP - Total number of steps for heat transfer analysis

$$\text{STEP} = \begin{bmatrix} n \\ \{ 1 \} \end{bmatrix} \quad \text{Applicable analysis types: THEAT}$$

where,

- n: is the total number of time steps to be used. This command should be used only when the maximum time of the analysis (variable TMAX in the *TIMEINTEG data group, [Section 7.1.10](#)) is set to zero.

Actual number of time steps may be different from the number defined here if: the problem is nonlinear, a variable time step size is selected, or a user defined time step size is used.

UFIJ - View factor and/or surface radiation control

User defined view factors can also be read by NISA/3D-HEAT. In addition, view factors computed by NISA/3D-HEAT can be printed out in the output file. This is done using an executive command.

Executive card UFIJ can be used to:

$$\text{UFIJ} = \begin{bmatrix} \text{IUSR VF} \\ \text{IPRVF} \end{bmatrix}$$

where,

IUSRVF: 0 = Do not read user defined view factors. Complete view factors internally.
This is a default.

1 = Read user defined view factors, do not compute view factors internally.

IPRVF : 0 = Do not print view factor matrix in the output file. This is default.

1 = Print the view factor matrix in the output file.

Note:

Numbering radiation surfaces (for user defined view factors):

Surfaces are assigned numbers 1 through N in the order they appear in the NISA file. Relevant data such as Stefan-Boltzman constant, surface emissivity and T_{∞} are given in the NISA file along with radiation surfaces.

Reading if user view factors:

If IUSRVF 1, NISA/3D-FLUID will read user defined view factors given in 'vfacnisa.dat' instead of computing view factors on its own. The file 'vfacnisa.dat' must be present in the current working directory of the user. The following FORTRAN subroutine can be used to create view factor input file 'vfacnisa.dat' for NISA/3D-FLUID.

```
SUBROUTINE VFNISA (NSIZE, NSURF, FIJ)
```

```
IMPLICIT NONE
```

```
INTEGER*4 NSIZE, NSURF, I, J
```

```
REAL*8 FIJ
```

```
DIMENSION FIJ (NSIZE, NSIZE)
```

```
OPEN (UNIT = 21, FILE = 'vfacnisa.dat', STATUS = 'UNKNOWN')
```

```
C    NSIZE: SIZE OF THE VIEW FACTOR MATRIX
```

```
C    NSURF: NO. OF SURFACES
```

```
C    FIJ: VIEW FACTOR MATRIX (KNOWN TO THE USER)
```

```
    DO 1000 I = 1, NSURF
```

```
        WRITE (21,2000) (FIJ (I, J), J = 1, NSURF)
```

```
2000 FORMAT (5 (2X, E13.6))
```

```
1000 CONTINUE
```

CLOSE 21
RETURN
END

INERTIA RELIEF – Activate inertia relief calculation

REL = KREL,KAUTO,NODBC0
{OFF}, {ON}, { }

where,

KREL : OFF - no inertia relief calculation
ON - activate inertia relief calculation

KAUTO : ON - program constrains the rigid body motions for a free body (see note 2)
OFF - no rigid body constraint is provided from the program

NODBC0 : A user selected node to be constrained for all translation motions (optional see note 3)

Note:

1. Inertia relief is only available for linear static analysis. Inertia relief is a feature that NISA computes inertia forces and moments to counter balance motion effect of applied loads to a free structure. The displacement constraints applied to the structure should be only enough to remove the rigid body motions to avoid singularity in linear static solution.
2. Program can properly constrain the structure internally to eliminate the rigid body motions for a free body when KAUTO is set to ON. If this feature is activated, it is suggested to avoid additional boundary constraints as they may over-constrain the body.
3. NISA2 will select a active node internally to constrain rigid body motions. User can override the program's choice by providing NODBC0 as the constrained node.

5.4 Input Data Setup for Various Analysis Types

5.4.1 Summary of Applicable Data Groups for Various Analyses

This section provides a quick reference summary of applicable executive commands and data groups for various analysis types (excluding modal dynamic analysis which are given in [Chapter 8](#)). [Table 5.6](#) shows executive commands applicable for various analyses whereas [Table 5.7](#) and [Table 5.8](#) show the model and analysis data groups applicable for each analysis type, respectively.

Table 5.6: Executive commands⁽¹⁾ applicable for various analysis types

Analysis Type	Commands for Specific Analysis Types		Commands Common to All Analyses
Linear Static	AUTO CONSTRAINT SORT STRESS SOLV	GEOM PROPERTIES FSIZ	ANALYSIS BLANK COMMON ELEMENT ECHO EXECUTION FILE NAME
Eigenvalue	AUTO CONSTRAINT GEOM PROPERTIES FSIZ	EIGEN EXTRACTION MASS FORMULATION	NODE ECHO RESEQUENCE RESTART SAVE FILE WARNING

Analysis Type	Commands for Specific Analysis Types		Commands Common to All Analyses
Buckling	AUTO CONSTRAINT GEOM PROPERTIES	EIGEN EXTRACTION FSIZ	ANALYSIS BLANK COMMON ELEMENT ECHO EXECUTION FILE NAME NODE ECHO RESEQUENCE RESTART SAVE FILE WARNING
Nonlinear Static	GEOM PROPERTIES NLTYPE FSIZ SOLV	MAXCPU TIME REFCONFIG CONMETHOD CTAL	
Linear Transient	AUTO CONSTRAINT MASS FORMULATION	GEOM PROPERTIES FSIZ SOLV	
Nonlinear Transient	GEOM PROPERTIES NLTYPE MASS FORMULATION FSIZ	MAXCPU TIME REFCONFIG AUTO CONSTRAINT SOLV	
Heat Transfer	INITIAL TEMPERATURE ORTHOTROPIC STEP SOLV	MASS FORMULATION SIGMA SRAD UFIJ	
Direct Frequency Response	AUTO CONSTRAINT MASS FORMULATION	GEOM PROPERTIES SOLV	

Table 5.7: Model data groups⁽¹⁾ applicable for various analysis types

Analysis Type	Commands for Specific Analysis Types	Commands Common to All Analyses
Linear Static	6.3.5 *LAMANGLE 6.3.6 *LAMSQ2 6.3.7 *BMDATA 6.3.8 *BMSECT 6.4.3 *G2 6.5.1 *MATERIAL 6.6.1 *RIGLINK 6.6.2 *MPCEQN 6.6.3 *CPDISP 6.7.2 *VECTORS 6.9.1 *MLOAD 6.9.2 *MPATH	
Eigenvalue	6.3.5 *LAMANGLE 6.3.6 *LAMSQ2 6.3.7 *BMDATA 6.3.8 *BMSECT 6.4.3 *G2 6.5.1 *MATERIAL 6.6.1 *RIGLINK 6.6.2 *MPCEQN 6.6.3 *CPDISP 6.7.2 *VECTORS	6.2.1 *TITLE 6.3.1 *ELTYPE 6.3.2 *ELEMENTS 6.3.3 *E1 6.3.4 *RCTABLE 6.4.1 *LCSYSTEM 6.4.2 *NODES 6.5.2 *MATDIR1 6.5.3 *MATDIR2 6.7.1 *SETS
Buckling	6.3.5 *LAMANGLE 6.3.6 *LAMSQ2 6.3.7 *BMDATA 6.3.8 *BMSECT 6.4.3 *G2 6.5.1 *MATERIAL 6.6.1 *RIGLINK 6.6.2 *MPCEQN 6.6.3 *CPDISP 6.7.2 *VECTORS	

Analysis Type	Commands for Specific Analysis Types	Commands Common to All Analyses
Nonlinear Static	6.3.5 *LAMANGLE 6.3.6 *LAMSQ2 6.3.7 *BMDATA 6.3.8 *BMSECT 6.3.9 *NLSPRING 6.4.3 *G2 6.5.1 *MATERIAL 6.5.5 *HYPEREL 6.5.6 *PLASTIC 6.5.7 *APLASTIC 6.5.8 *CREEP 6.6.2 *MPCEQN 6.6.3 *CPDISP 6.7.2 *VECTOR 6.7.4 *TIMEAMP 6.8.1 *CSURFACE 6.8.2 *CELEMENTS	6.2.1 *TITLE 6.3.1 *ELTYPE 6.3.2 *ELEMENTS 6.3.3 *E1 6.3.4 *RCTABLE 6.4.1 *LCSYSTEM 6.4.2 *NODES 6.5.2 *MATDIR1 6.5.3 *MATDIR2 6.7.1 *SETS
Linear Transient	6.3.5 *LAMANGLE 6.3.6 *LAMSQ2 6.3.7 *BMDATA 6.3.8 *BMSECT 6.4.3 *G2 6.5.1 *MATERIAL 6.5.9 *PDAMPING 6.6.1 *RIGLINK 6.6.2 *MPCEQN 6.6.3 *CPDISP 6.7.2 *VECTOR 6.7.4 *TIMEAMP	

Input Setup and Executive Commands

Input Data Setup for Various Analysis Types

Analysis Type	Commands for Specific Analysis Types		Commands Common to All Analyses
Nonlinear Transient	6.3.5 *LAMANGLE 6.3.7 *BMDATA 6.3.9 *NLSPRING 6.5.1 *MATERIAL 6.5.6 *PLASTIC 6.5.9 *PDAMPING 6.7.4 *TIMEAMP	6.3.6 *LAMSQ2 6.3.8 *BMSECT 6.4.3 *G2 6.5.5 *HYPEREL 6.5.7 *APLASTIC 6.7.2 *VECTOR	6.2.1 *TITLE 6.3.1 *ELTYPE 6.3.2 *ELEMENTS 6.3.3 *E1 6.3.4 *RCTABLE 6.4.1 *LCSYSTEM 6.4.2 *NODES 6.5.2 *MATDIR1 6.5.3 *MATDIR2 6.7.1 *SETS
Heat Transfer	6.5.4 *MATHEAT 6.7.3 *TEMPFN 6.7.5 *PCHANGE1	6.6.4 *CPTEMP 6.7.4 *TIMEAMP	
Direct Frequency Response	6.3.7 *BMDATA 6.4.3 *G2 6.5.9 *PDAMPING 6.7.2 *VECTOR	6.3.8 *BMSECT 6.5.1 *MATERIAL 6.6.3 *CPDISP 6.7.10 *FREQFUNCTION	

(1) Acceptable minimum abbreviations are in bold face

Table 5.8: Analysis data groups⁽¹⁾ applicable for various analysis types

Analysis Type	Commands for Specific Analysis Types					
Linear Static	7.1.1	*LDCASE	7.1.6	*LDCOMB	7.1.7	*LCTITLE
	7.2.1	*FRCNTL	7.2.2	*FRCOEF	7.2.3	*ANGSEC
	7.3.1	*SPDISP	7.3.2	*CFORCE	7.3.4	*PRESSURE
	7.3.5	*BEAMLOAD	7.3.6	*BODYFORCE	7.3.7	*NDTEMPER
	7.3.8	*NDTEMPDIF	7.3.11	*L1	7.5.3	*PRINCNTL
	7.5.4	*REGIONS	7.5.5	*STRSFILTER	7.5.6	*SFDCOMP
	7.5.10	*I5	7.5.11	*N5	7.5.12	*POSTCNTL
7.6.1	*ENDDATA					
Eigen Value	7.1.2	*EIGCNTL	7.1.8	*MODEOUT	7.3.1	*SPDISP
	7.5.1	*EIGOUT	7.5.3	*PRINCNTL	7.5.4	*REGIONS
	7.5.5	*STRSFILTER	7.5.6	*SFDCOMP	7.5.10	*I5
	7.5.11	*N5	7.6.1	*ENDDATA		
Buckling	7.1.1	*LDCASE	7.1.2	*EIGCNTL	7.1.7	*LCTITLE
	7.1.8	*MODEOUT	7.3.1	*SPDISP	7.3.2	*CFORCE
	7.3.4	*PRESSURE	7.3.5	*BEAMLOAD	7.3.6	*BODYFORCE
	7.3.7	*NDTEMPER	7.3.8	*NDTEMPDIF	7.3.11	*L1
	7.5.1	*EIGOUT	7.5.3	*PRINCNTL	7.5.4	*REGIONS
	7.5.5	*STRSFILTER	7.5.6	*SFDCOMP	7.5.10	*I5
	7.5.11	*N5	7.5.12	*POSTCNTL	7.6.1	*ENDDATA

Input Setup and Executive Commands

Input Data Setup for Various Analysis Types

Analysis Type	Commands for Specific Analysis Types					
Nonlinear Static	7.1.3	*EVENT	7.1.7	*LCTITLE	7.1.10	*STEPSIZE
	7.1.11	*CONVERGENCE	7.3.1	*SPDISP	7.3.2	*CFORCE
	7.3.3	*CFOLLOWER	7.3.4	*PRESSURE	7.3.5	*BEAMLOAD
	7.3.6	*BODYFORCE	7.3.7	*NDTEMPER	7.3.8	*NDTEMPDIF
	7.5.2	*NLOUT	7.5.3	*PRINCNTL	7.5.4	*REGIONS
	7.5.5	*STRSFILTER	7.5.10	*I5	7.5.11	*N5
	7.6.1	*ENDDATA				
Linear Transient	7.1.3	*EVENT	7.1.7	*LCTITLE	7.3.1	*SPDISP
	7.3.2	*CFORCE	7.3.4	*PRESSURE	7.3.9	*INITIAL
	7.3.10	*TBODYFORCE	7.5.2	*NLOUT	7.5.3	*PRINCNTL
	7.5.4	*REGIONS	7.5.9	*HISTOUT	7.6.1	*ENDDATA
Nonlinear Transient	7.1.3	*EVENT	7.1.7	*LCTITLE	7.3.1	*SPDISP
	7.3.2	*CFORCE	7.3.3	*CFOLLOWER	7.3.4	*PRESSURE
	7.3.5	*BEAMLOAD	7.1.10	*STEPSIZE	7.3.9	*INITIAL
	7.3.10	*TBODYFORCE	7.5.2	*NLOUT	5.5.3	*PRINCNTL
	7.5.4	*REGIONS	7.5.9	*HISTOUT	7.6.1	*ENDDATA
Nonlinear Transient	7.1.5	*HEATCNTL	7.1.9	*TIMEINTEG	7.1.10	*STEPSIZE
	7.4.1	*INITEMP	7.4.2	*SPTEMP	7.4.3	*CFLUX
	7.4.4	*DFLUX	7.4.5	*ELHEATGEN	7.4.6	*NDHEATGEN
	7.4.7	*CONVBC	7.4.8	*RADBC	7.5.3	*PRINCNTL
	7.4.9	*RADSURFACE	7.5.7	*TEMPHISTORY	7.5.8	*TEMPOUT
	7.6.1	*ENDDATA				

Analysis Type	Commands for Specific Analysis Types					
Direct Frequency Response	7.1.4	*FREQNTL	7.3.1	*SPDISP	7.3.2	*CFORCE
	7.3.4	*PRESSURE	7.5.2	*NLOUT	7.5.3	*PRINTCNTL
	7.5.9	*HISTOUT	7.6.1	*ENDDATA		

5.4.2 Input Data Setup for Static Analysis

Figure 5.2 shows a global description of the input data setup for a typical static analysis run. The first data block (executive commands) should flag the analysis type as a static analysis, and should include appropriate or applicable commands for file saving, element resequencing, etc. Note that for static analysis, the entire executive commands block may be skipped if the user elects to use all the default options of these commands.

The second data block (the model data) should start with the ***TITLE** data group in order to print the problem title as a heading on all pages of the output file. The model data block then describes model characteristics, for example: element definition (***ELEMENT**), nodal coordinates (***NODES**), material properties (***MATERIAL**), etc. Some kinematic constraints are considered to be model characteristics and are defined in this block, namely: the rigid element data (***RIGLINK**), the multi-point constraint equations (***MPCEQN**), and the coupled displacement data (***CPDISP**). Other general groups such as definition of vectors and sets (***VECTORS** and ***SETS**, respectively) are also defined in this block.

The third data block (the analysis data) must start with the ***LDCASE** data group which defines control parameters for the first load case. A load case title is available on the ***LCTITLE** data group. The block then describes various loadings and kinematic boundary conditions of the structure. Loadings may include for example pressure (***PRESSURE**), concentrated nodal forces (***CFORCE**), body forces (***BODYFORCE**), thermal loads (***NDTEMPER** or ***NDTEMPDIF**), etc. Kinematic boundary conditions only contains the specified displacements (***SPDISP**). The block also defines various output and print control options. Data for other load cases should follow similar to the first one. Load combination of various load cases may then start with the ***LDCOMB** data group. Each load combination may have its own title (***LCTITLE**) and/or output options.

The last card of the input deck must always be the input data terminator, the *ENDDATA data group.

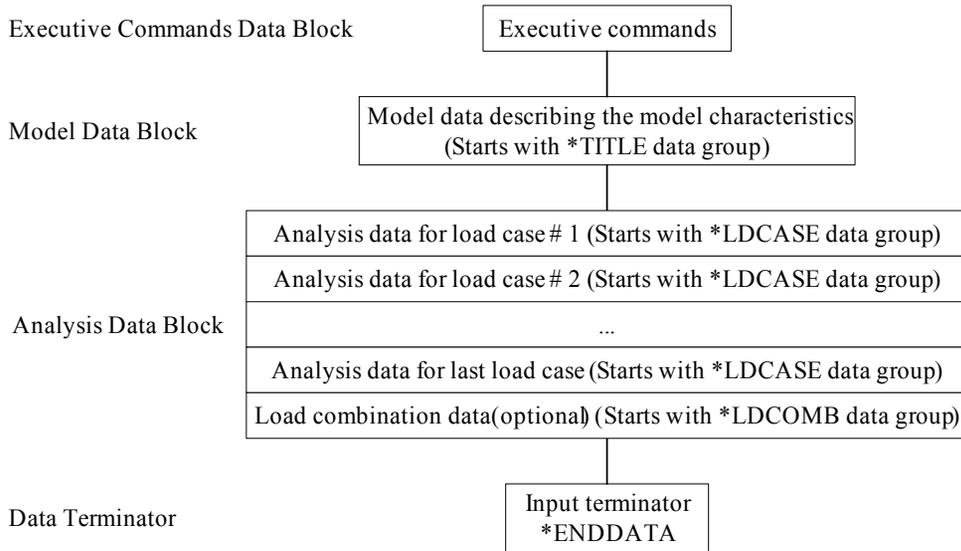


Figure 5.2: Input data setup for static analysis

5.4.3 Input Data Setup for Eigenvalue Analysis

Figure 5.3 shows a global description of the input data setup for a typical eigenvalue analysis run. The first data block (executive commands) must flag the analysis type as an eigenvalue analysis, and should include appropriate or applicable commands for file saving, element resequencing, eigenvalue extraction algorithm, mass formulation, etc.

The second data block (the model data) should start with the *TITLE data group in order to allow printing of the problem title as a heading on all pages of the output file. The model data then describes model characteristics for example: element definition (*ELEMENT), nodal coordinates (*NODES), material properties (*MATERIAL), etc. Some kinematic constraints are considered to be model characteristics and are defined in this block namely: the rigid element data (*RIGLINK), the multi-point constraint equations (*MPCEQN), and the coupled displacement data (*CPDISP). Other general groups such as definition of vectors and sets (*VECTORS, and *SETS, respectively) are also defined in this block.

The third data block (the analysis data) should start with the eigenvalue analysis control, (*EIGCNTL). No loading is defined for eigenvalue analysis, and the specified displacement boundary condition may be given, but only zero values are allowed. The block then describes output control options, using *MODEOUT, *EIGOUT, and other pertinent data groups.

The last card of the input deck must always be the input data terminator, the *ENDDATA data group.

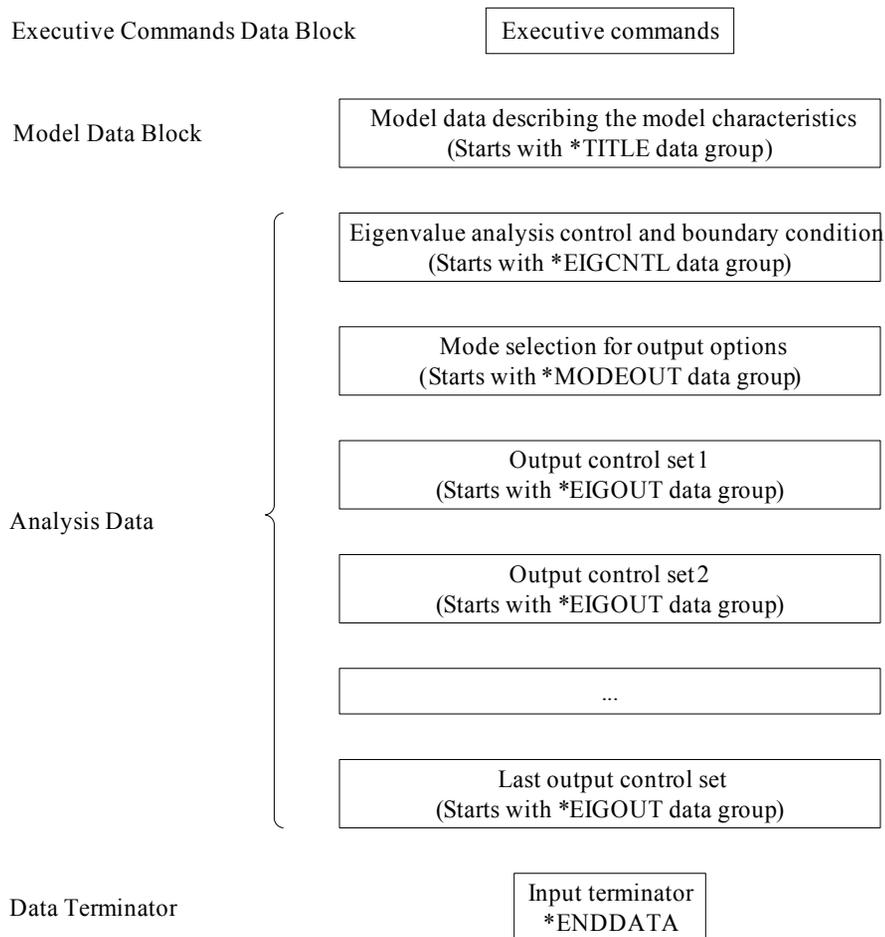


Figure 5.3: Input data setup for eigenvalue analysis

5.4.4 Input Data Setup for Nonlinear Static Analysis

Figure 5.4 shows a global description for the input data setup for a typical nonlinear static analysis run. The first data block (executive commands) must flag the analysis type as a nonlinear static analysis, and should include appropriate or applicable commands for file saving, element resequencing, specific type(s) of nonlinearities, reference configuration, etc.

The second data block (the model data) should start with the *TITLE data group in order to allow printing of the problem title as a heading on all pages of the output file. The model data then describes model characteristics, for example: element definition (*ELEMENT), nodal coordinates (*NODES), material properties (*MATERIAL, *HYPEREL, *PLASTIC, *CREEP), etc. Other groups such as the definition of sets and time-amplitude curves (*SETS, and *TIMEAMP, respectively) are also defined in this block.

The third data block (the analysis data) should start with the *EVENT data group. An event title is available on the *LCTITLE data group. The block then describes various loadings and kinematic boundary conditions of the structure. Loadings include for example follower or non-follower pressure data (*PRESSURE), concentrated forces (*CFORCE), follower concentrated forces (*CFOLLOWER), body forces (*BODYFORCE), thermal loads (*NDTEMPER or *NDTEMPDIF), etc. Kinematic boundary conditions only contain the specified displacements data (*SPDISP). The block also defines various output and print control options. Data for other events should follow similar to the first one. Generally an event should be defined when new load types are introduced or when an existing load changes its sign. Each event can have one or more load increments or steps. It should be noted that, currently, the specified displacement data; *SPDISP, cannot be changed from one event to another. When creep analysis is used, the first event should be without creep. During creep, the loads should not change.

The last card of the input deck must always be the input data terminator, the *ENDDATA data group.

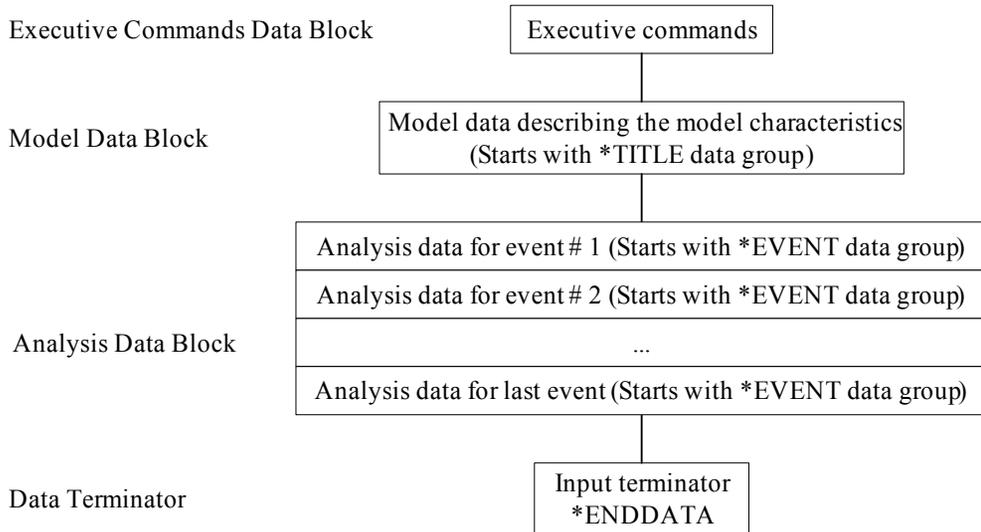


Figure 5.4: Input data setup for nonlinear static analysis

5.4.5 Input Data Setup for Buckling Analysis

Buckling analysis is a one-run, two-pass procedure. The first pass is a static analysis which solves for the stresses under the prescribed loading conditions. The second pass is an eigenvalue analysis which determines the buckling load factors and associated mode shapes. As such, the input data setup for buckling consists of static analysis data followed by eigenvalue analysis data.

Figure 5.5 show global description of the input data setup for a typical buckling analysis run. The first data block (executive commands) should flag the analysis type as a buckling analysis and should include appropriate or applicable commands for file saving, element resequencing, eigenvalue extraction algorithm, etc.

The second and third data blocks are the model data and the analysis data for the static analysis pass. Only one static load case may be used.

The eigenvalue analysis data block then follows. The last card of the input deck must always be the input data terminator, the *ENDDATA data group.

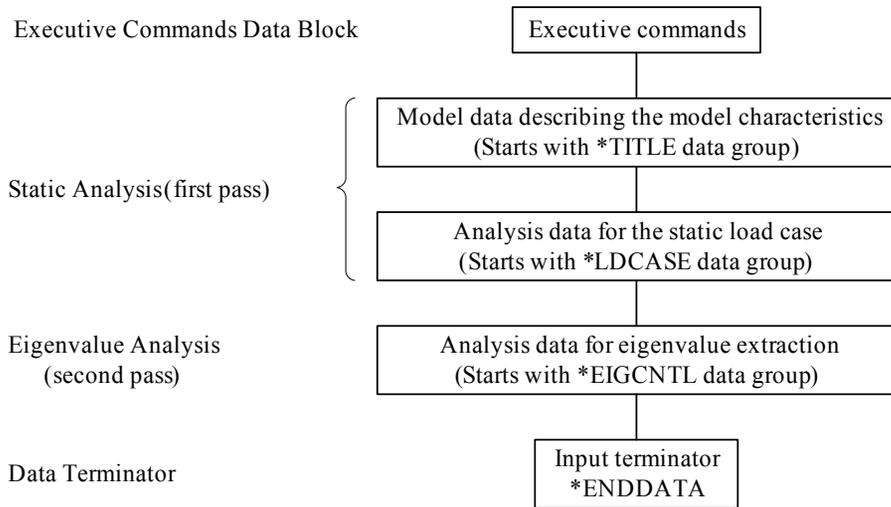


Figure 5.5: Input data setup for buckling analysis

5.4.6 Input Data Setup for Heat Transfer Analysis

Figure 5.6 shows a global description for the input data setup for a typical heat transfer run. The first data block (executive commands), must flag the analysis type as a steady state or transient heat transfer (SHEAT or THEAT, respectively). It should include appropriate or applicable commands for file saving, element re-sequencing, etc.

The second data block (the model data) should start with the *TITLE data group in order to allow printing of the input title as a heading on all pages of the output file. The model data then describes model characteristics as for example: element definition (*ELEMENT), nodal coordinates (*NODES), material properties (*MATHEAT), etc. Other general groups such as temperature function table (*TEMPFN), definition of sets (*SETS), and time-amplitude curves (*TIMEAMP) are also defined in this block.

The third data block (the analysis data) should start with the *HEATCNTL data group. The block then describes various heat fluxes, heat generation, and other boundary conditions. Heat fluxes may include, for example, distributed heat flux data (*DFLUX), and concentrated nodal heat fluxes (*CFLUX). Internal heat generation is specified as nodal (*NDHEATGEN) or elemental (*ELHEATGEN) values. Other boundary conditions include, for example: specified nodal temperatures (*SPTEMP), radiation and convection

boundary conditions (*RADBC and *CONVBC, respectively), etc. The block also specifies time integration and step size control parameters, as well as various output and print out options. Only one load case is allowed for each heat transfer run.

The last card of the input data must always be the input data terminator, the *ENDDATA data group

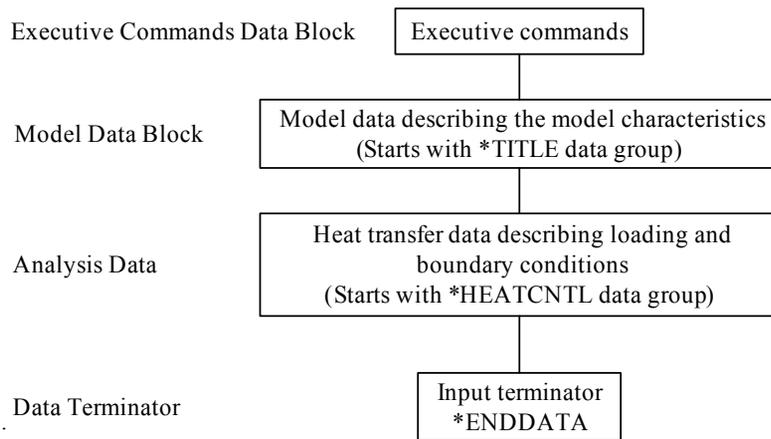


Figure 5.6: Input data setup for heat transfer analysis

5.4.7 Procedure for Thermal Stress Analysis Following Heat Transfer Analysis

Thermal stress analysis may be performed by executing a heat transfer analysis run followed by a static analysis run. The heat transfer analysis run solves for the nodal temperature distribution which will then be input into the structural analysis run to calculate displacements and stresses due to the thermal loading. The procedure may be summarized in the following two main steps:

1. Heat Transfer Analysis Run:

In this run, the data setup given in [Section 5.4.6](#) is followed. In the executive commands, user has to save file 39 which will be later accessed by the structural analysis run. File 39 is an ASCII file written in the format of the *NDTEMPER and

*NDTEMPDIF data groups for each time step. For steady state analysis, file 39 will contain only one-step data.

2. Structural Analysis Run:

The first step in this run is to edit the model data used in the heat transfer run to be applicable for structural analysis. In particular, element types have to be changed, material properties have to be defined, and boundary condition and/or kinematic constraints should be added. As a rule-of-thumb, subtracting 100 from the heat transfer NKTP element number should give a compatible structural NKTP element number, e.g., NKTP = 104 is compatible with NKTP = 4. In some cases, particularly the shell (NKTP = 120) and the bar (NKTP = 112) elements, the user has to choose from various structural shell and beam elements available in the program.

The second step in this analysis is to access the data saved on file 39 of the heat transfer run and use it as a loading condition for the structural analysis run. This may be done either by using the file read commands of *NDTEMPER or *NDTEMPDIF data groups *or* by amending file 39 into structural analysis data deck. The file read commands of *NDTEMPER and *NDTEMPDIF data groups can be used to access the required time step data from file 39 (see [Section 7.3.7](#) and [Section 7.3.8](#) for more information).

5.4.8 Procedure for Modal Dynamic Analysis

(Transient Dynamic, Random Vibration, Frequency Response and Shock Spectrum Analyses)

Modal dynamic analyses, namely transient dynamics, random vibration, frequency response, and shock spectrum analyses are performed following eigenvalue analysis. The procedure may be summarized in the following two steps:

1. Eigenvalue Analysis Run

The data setup given in [Section 5.4.4](#) should be followed for this run. If a particular response quantity such as stress is to be evaluated in the modal dynamic analysis, the corresponding modal response quantity, that is modal stresses, must be requested in the eigenvalue analysis run. NISA files 26 and 27 should be saved using the executive command 'SAVE FILE'. If a transient dynamic analysis with nonzero initial conditions needs to be performed, then files 30, 32 and 35 also need to be saved.

2. Modal Dynamic Analysis Run:

The input data setup for this run is described in detail in [Chapter 8](#). The executive commands for this run is summarized in [Table 5.9](#) classified according to the modal dynamic analysis type.

The valid data groups for this run are given in [Table 5.10](#). NISA files 26 and 27 saved from the eigenvalue analysis should be made available to this run using the executive command ‘FILE NAME’ (see [Section 8.3](#)). These two files will be updated and automatically saved.

Table 5.9: Executive commands for modal dynamic analyses

Analysis Type	Commands for Specific Analysis Types	Commands common to All Analyses
TRANSIENT	ENDTIME DELTATIME STARTINGTIME	
RANDOM	ACUTOFF FLOWER FRQRDF FUPPER GENFREQUENCY INPOLATION INTEGRATION TSEISMIC	
FREQUENCY	FLOWER FRQRDF FUPPER GENFREQUENCY INPHASE INPOLATION MRESPONSE	
SHOCK	DIRECTION INPOLATION TSEISMIC CFREQ	

Table 5.10: Data groups for modal dynamic analyses

Analysis Type	Commands for Specific Analysis Types		Data Groups Common to All Analyses
TRANSIENT	8.6.1 *ARRIVALTIME 8.6.2 *TIME FUNCTION 8.8.2 *HISTORY	8.5.5 *INITIAL 8.5.3 *TSTEP 8.8.3 *SNAPSHOT	8.4.1 *TITLE 8.7.2 *GROUND 8.5.1*MODESELECTION 8.7.3 *DCFORCE 8.5.2 *DAMPING 8.7.4*DPRESSURE 8.8.1 *RSET 8.9.1 *ENDDATA 8.7.8 *MSEXCITATION
RANDOM	8.6.4 *CORRELATION 8.5.4 *ADDFREQUENCY 8.8.5 *RMSOUT	8.6.3 *PSDFUNCTION 8.8.4 *PSDOUT	
FREQUENCY	8.6.5 *SPECTRUM 8.7.5 *DRIVER	8.5.4 *ADDFREQUENCY 8.8.6 *SPOUT	
SHOCK	8.6.5 *SPECTRUM 8.8.7 *RESPONSE	8.6.6 *MDSPECTRUM 8.6.9 *PRSPECTRA	

Note:

The shock spectrum analysis does not allow for *DCFORCE or *DPRESSURE load specification, and does not employ the *RSET data group.

5.4.9 Input Data Setup for Linear and Nonlinear Direct Transient Dynamic Analysis

Figure 5.7. shows a global description for the input data setup for a typical linear direct transient dynamic analysis run. The first data block (executive commands) must flag the analysis type as linear direct transient analysis, and should include appropriate or applicable commands for file saving, element resequencing, etc.

The second data block (the model data) should start with the *TITLE data group in order to allow printing of the problem as a heading on all pages of the output file. The model data then describes model characteristics, as for example: element definition (*ELEMENT), nodal coordinates (*NODES), material properties (*MATERIAL), etc. Some kinematic constraints are considered to be model characteristics and are defined in this block, namely: the rigid element data (*RIGLINK), the multi-point constraint equations (*MPCEQN), and the coupled displacement data (*CPDISP). Other general groups such as definition of vectors and sets (*VECTORS and *SETS, respectively) are also defined in this block.

The third data block (the analysis data) should start with *EVENT data group. An event title is available on the *LCTITLE data group. The block then describes various loadings, kinematic boundary conditions and initial conditions of the structure. The initial condition data (*INITIAL) can appear before the *EVENT data group or in the first event. Loadings include, for example, concentrated force (*CFORCE) and body force (*TBODYFORCE). Kinematic boundary conditions only contain the specified displacement data (*SPDISP). The block also defines various output and print control options. It should be noted that the specified displacement data, *SPDISP, cannot be changed from one event to another.

The last card of the input deck must always be the input data terminator, the *ENDDATA data group.

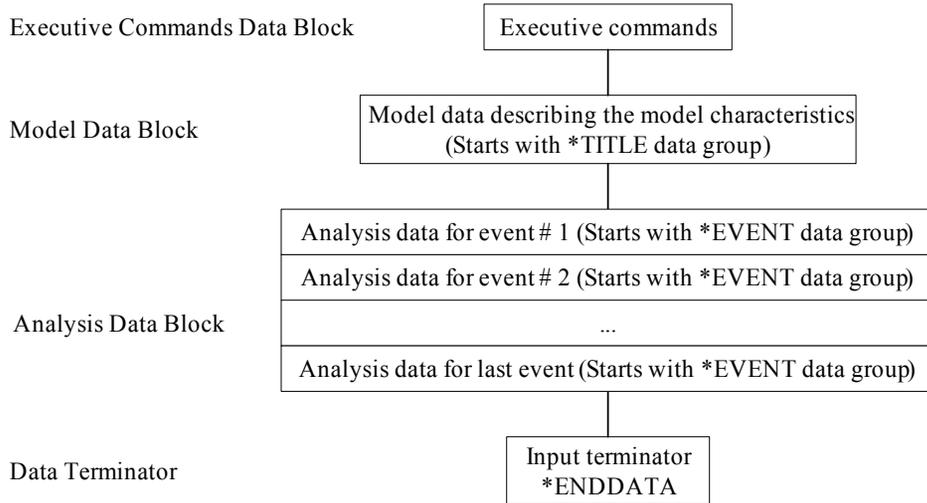


Figure 5.7: Input data setup for linear transient analysis

5.4.10 Input Data Setup for Direct Frequency Response Analysis

Figure 5.8 shows a global description for the input data setup for a typical direct frequency response analysis run. The first data block (executive commands) must flag the analysis type as direct frequency response analysis, and should include appropriate or applicable commands for file saving, element resequencing, etc.

The second data block (the model data) should start with the *TITLE data group in order to allow printing of the problem as a heading on all pages of the output file. The model data then describes model characteristics, as for example: element definition (*ELEMENT), nodal coordinates (*NODES), material properties (*MATERIAL), etc. Some kinematic constraints are considered to be model characteristics and are defined in this block, namely: the rigid element data (*RIGLINK), the multi-point constraint equations (*MPCEQN), and the coupled displacement data (*CPDISP). Other general groups such as definition of vectors and sets (*VECTORS and *SETS, respectively) are also defined in this block.

The third data block (the analysis data) should start with frequency control *FREQCNTL data group. The block then describes various loadings, kinematic boundary conditions and initial conditions of the structure. Loadings include, for example, concentrated force (*CFORCE) and pressure (*PRESSURE). Kinematic boundary conditions only contain the

specified displacement data (*SPDISP). The block also defines various output and print control options. It should be noted that the specified displacement data, *SPDISP, cannot be changed from one frequency control to another.

The last card of the input deck must always be the input data terminator, the *ENDDATA data group.

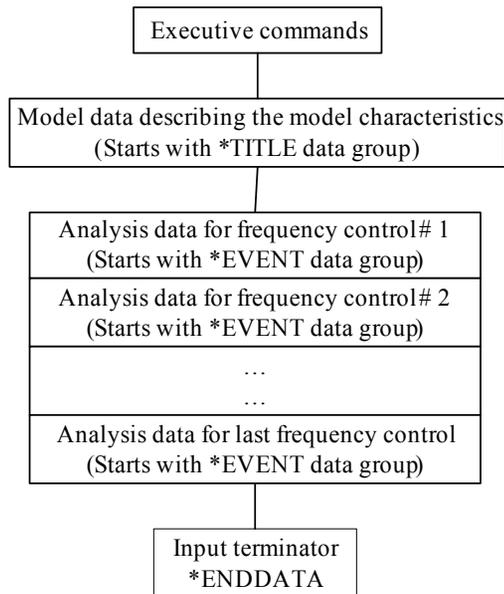


Figure 5.8: Input data setup for direct frequency response analysis

5.5 Restart Procedures

5.5.1 Introduction

The restart capabilities for structural and heat transfer analyses are summarized in [Section 3.16](#). This section explains the data deck setup and file requirements for restarts, gives some guidelines for effective use of the capabilities, and describes some special features in detail. The procedures for modal dynamic analyses, which may also be viewed as restart runs, are given in [Section 3.5](#), [Section 5.4.8](#) and [Chapter 8](#).

As mentioned in [Section 3.16](#), the restart capabilities provide added flexibility and may be used to utilize some data or results that have been processed or computed in the preceding run, usually a fresh (or scratch) run, provided that certain files have been saved and some requirements are met.

The available restart types are summarized in [Table 5.11](#) and [Table 5.12](#) for structural and heat transfer analyses, respectively. Also shown in the tables are the applicable analysis types for the previous and current runs together with file requirements and brief description of the capabilities. A restart from a restart run is allowed subject to the requirement of previous and present analysis types as summarized in the tables. [Figure 3.22](#) gives an example for possible multiple restart-from-restart runs for structural analysis.

5.5.2 Data Deck Setup for Restart Runs

The input data setup for restarts is the same as for fresh runs. That is, it consists of an executive command block, a model data block and an analysis data block followed by the data deck terminator as explained in [Section 5.4](#). However, since the restart run is dependent on the preceding run, some executive commands and data groups may not be applicable in the analysis being performed. [Table 5.5.3](#) through [Table 5.5.5](#) give the applicable executive commands and data groups for each restart type.

The program has a built-in capability to only consider the applicable executive commands and the data groups for the analysis and restart type being performed. Data groups which are not applicable are automatically skipped. (Note that the given executive commands are echoed in the printout whether they are applicable or not.)

In general, the model data block may be omitted since the processed model data from the preceding run is used. However, certain model data groups may be given in the restart run as indicated in [Table 5.5.4](#) and [Table 5.5.5](#). For structural restarts, these data groups are regarded as miscellaneous data rather than truly model data. In particular, the *TITLE and *SETS data group may be given, otherwise the previous definitions are used.

The executive commands 'FILENAME', 'SAVEFILE' and 'RESTART' (see [Section 5.3](#)) should be examined carefully during the preparation of the input data for fresh and restart runs as indicated below.

The command 'FILENAME' specifies the file name prefix to be used for files to be saved in the current run, which are in turn specified by the command 'SAVEFILE'. For a restart run, the 'FILENAME' command must be given and should correspond to the file name prefix specified in the preceding run (unless the files are copied or renamed with a different prefix by the user).

The command 'SAVEFILE' may be given in the restart run to request saving of some 'new' files. Old files from a preceding run which are used in a restart run are automatically saved. For a typical restart type, the files required from the preceding run(s) are internally identified, and a checking on their existence is performed by the program as a restart validation check.

The restart run is indicated in the input data deck through the executive command 'RESTART = n' ([Section 5.3](#)), where, n is a valid restart type number as indicated in [Table 5.11](#) and [Table 5.12](#). A fresh (or scratch) run is designated by a restart type number of zero (n = 0) and is the default option.

For users who are utilizing the restart capabilities for the first time, the following procedure is suggested:

- ❑ Before submitting a normal NISA fresh run, refer to [Table 5.11](#) and [Table 5.12](#) for information about restarts that can be performed subsequently.
- ❑ Refer to the following subsections for detailed information about restart capabilities and requirements.
- ❑ For large models, it may be worthwhile to exercise the intended restart options on smaller models first.

- ❑ Prepare the input data deck for the fresh run, include the executive commands ‘FILENAME’ and ‘SAVEFILE’ to indicate the file name prefix for permanent files and NISA file numbers to be saved.
- ❑ After a successful run, the input file for the restart run may be prepared from scratch or the input file from the previous run may be edited for very few modifications. The executive commands ‘RESTART’ and ‘FILENAME’ should specify the restart type and the permanent file name prefix, respectively. For structural restarts, the intended restart options may be verified through a restart check run as indicated in the [Section 5.5.3](#).
- ❑ Before the restart run is submitted, it is recommended that the files saved in the original run be copied for backup purposes, for possible reuse in a different restart run (that references the original run rather than the preceding run), and to guard against accidental computer failure or file overwriting during a run.

5.5.3 General Requirements and Guidelines for Restart Runs

The following are important rules and guidelines that should be observed for successful use of the restart capabilities:

1. The executive commands ‘RESTART’ and ‘FILENAME’ must be given in the input file for restarts. If the ‘RESTART’ command, which specifies the restart type number, is accidentally omitted, the program assumes that a fresh run (restart = 0) is being performed. Thus the files saved from an earlier run may be accidentally overwritten if this command is not provided. In addition, the command ‘FILENAME’ should specify the file name prefix to identify the permanent files required in the restart run.

Caution

NISA uses two sets of files; permanent files which are to be saved with the prefix specified in ‘FILENAME’ command and scratch files which are deleted at the end of the run. If a permanent file with the same prefix name and file number exists, it will be overwritten in the current run if it corresponds to a “new” file requested to be saved. “Old” files are automatically saved in restarts.

2. The preceding run from which a restart is done should be a valid analysis type from which the restart can be attempted as indicated in [Table 5.11](#) and [Table 5.12](#). For multiple restart-from-restart runs, a typical restart run is always based on the preceding run. If the analysis type is changed in an intermediate restart run (and the same file 26 is used), a subsequent restart run cannot access the results of the previous analysis type.

This is depicted in [Figure 3.22](#) as an example; once the analysis type is changed from linear static analysis to eigenvalue analysis, the previous static analysis results are not accessible. Therefore, a subsequent static restart type 2 or 3 is not possible (unless the necessary files were copied before the change in the analysis type).

3. Some of the executive commands may not be applicable in restart runs, although they may be applicable for the analysis type being performed. For example, the executive command ‘RESEQUENCE’ is ignored in restart runs since the element resequencing process for wavefront reduction may only be invoked in a fresh run. Similarly, the commands which in one way or another affect the model definition are ignored. [Table 5.13](#) gives the applicable executive commands for each restart type.
4. While new loading conditions or a change of analysis type may be possible in restarts, the model definition cannot be changed in general. [Table 5.14](#) and [Table 5.15](#) list the applicable model and analysis data groups for structural and heat transfer restarts, respectively.
5. Since material properties cannot be changed in structural restarts, the original definition should specify all material properties which may be needed in subsequent restarts. A common pitfall is that the material density (or an equivalent lumped mass representation) is not specified in a fresh run for an analysis type that does not require such a definition, e.g., static analysis without body forces. A subsequent eigenvalue restart 4 will not, in this case, be able to find frequencies because of the absence of the mass representation in the model data.
6. The element types to be used in a restart run must be valid element types for the analysis type to be performed. For example, a nonlinear static run using the cable element (NKTP = 45, NORDR = 1) cannot be followed by an eigenvalue restart 4 run, since this element is not available in eigenvalue analysis. [Table 4.1](#) and [Table 4.2](#) give the applicable analysis types for all the NISA elements. With few exceptions, most of the element types are available in all analysis types. The applicable element types are verified internally for the analysis type being performed as part of data checking.
7. There is no direct restart available between structural and heat transfer analyses. However, a procedure for thermal stress analysis following heat transfer analysis is given in [Section 5.4.7](#).
8. For structural restarts, the information about the preceding run contained in the model-and-analysis data base file (file 26), and the post-data file (file27), are printed in the output file for restart runs by default (see command ‘RESTART’, [Section 5.3](#)). The information includes a list of the files saved in the preceding run in addition to the model parameters, e.g., number of elements and nodes, element types used, largest node and element IDs, number of MPCs and rigid links, etc. Summary of the analysis data is also included, e.g., number of load cases, maximum wavefront, number of

degrees of freedom, loading types used, etc. The available output quantities on the post-data file are also indicated.

9. For structural restarts, the meaning of a check run is different from that for a fresh run ([Section 3.12.1](#)). A restart check run is activated by including the executive command 'EXECUTION = CHECK' in the input file for restart. A restart check run provides the applicable information about the preceding run as mentioned earlier, and performs some restart validation checks. No further data checking is performed (data checking is performed in a normal restart run). The run terminates with a message that the check run is completed and no files are updated. The user can then correct the input file for errors, if any are indicated by diagnostic messages, and resubmit the run as a normal restart run (i.e., delete the command 'EXECUTION = CHECK').
10. The permanent files saved from previous runs and used in the current run are either appended with new data, overwritten, or used as read-only files as shown in [Section 5.5.6](#). In case of an abnormal termination of a restart run, the status of these files should be evaluated before being reused. Normally, no change in the files occur during the input reading and processing stage.
11. The model-and-analysis data base file; file 26, has a dictionary that contains all the problem parameters and allows access to different data on the same file as well as on other files used. The dictionary on file 26 is updated (overwritten) in restart runs in the following cases:

For structural restarts:

- The restart run is completed successfully.
- Some progress has been achieved in the run before completion. This includes: a successful completion of a typical load case for static restarts, a successful computation of eigenvalue extraction in eigenvalue or buckling restart 4, a converged load step scheduled to be saved in nonlinear static restarts, and a completed event in direct transient dynamics.

For heat transfer restarts:

- The restart run is completed successfully.

Once the dictionary on file 26 is updated, it will reflect the current completed stages in the present run. Further restarts, if desired, should be based on the completed assignments of this run. (For static restarts 2 and 3, the dictionary will reflect the entire history of all old load cases of prior runs and all 'completed' load cases in the present run).

12. For structural restart type 2, 3, 4 and 5, the previous run must not be a check run. This is because these restarts access results of the previous run. As indicated in [Section 3.12.1](#),

no actual computations are performed during a check run. For structural restart type 1, however, the previous run may be a check run.

13. For printout control purposes, the *PRINTCNTL data group ([Section 7.5.3](#)) should not refer to a *PRINTCNTL group specified in a previous run.
14. Even though it is possible, structural restart type 2 (post-solution for static and eigenvalue analysis) is not recommended for printout control purposes only. It should be noted in this regard that the DISPLAY-POST program has a feature for selective tabulation of output quantities. Refer to the 'EDT' command in the DISPLAY User's Manual
15. The results of all previous restart runs (made using the same files 26 and 27) may be postprocessed or plotted using the DISPLAY-POST program, refer to command 'RSR' ('CLD' command for nonlinear static analysis restart) in the DISPLAY User's Manual. For static restarts, this feature may not be of considerable use since upon a successful completion of a typical static analysis run, the dictionary in file 26 reflects all the information about old and new load cases. This could be viewed as if all the static load cases (defined in fresh or restart 2 and 3 runs) were combined and submitted in a single run. However, for other restart types, this facility is significant, especially when the analysis type is changed in a restart run; for example, an eigenvalue analysis following a static or nonlinear static analysis.

5.5.4 Special Features and Requirements in Static Analysis Restarts

It should be noted that static restarts 2 and 3 are not available for axisymmetric elements with non-axisymmetric loading (NKTP = 34, 37). With the exception of static restart 1, multiple static restarts 2 and 3 using the same files 26 and 27 will preserve all the accumulated information of all processed load cases. That is, following a typical static restart 2 or 3, the information on files 26 and 27 will be similar to that which would have been obtained if all the processed load cases throughout the restart history were submitted in a single static fresh run. Thus, a typical static restart 2 or 3 run can directly reference or access any static load case completed in all previous runs, not necessarily the last one.

It is recommended that, for restart type 2 and 3, new load cases and new load combinations be given with specified identification numbers since conflicts may arise if the default identification numbers are used (see *LDCASE and *LDCOMB in [Section 7.1.1](#) and [Section 7.1.5](#) respectively).

Static analysis restart type 2

This restart may be used for post-solution and/or load combination of previous load cases. A typical load case to be used in this restart must be identified with a *LDCASE data group which should specify the post-solution options in addition to the load case-ID. The given load case identification number must not be a new load case-ID, instead it should correspond to the referenced load case of prior runs. The analysis data specifying loading conditions cannot be changed and if they are given (e.g. *CFORCE, *SPDISP) they are ignored.

For load combination, the desired response quantities of load cases to be combined should have been (or are currently being) calculated for the specified load cases. Load combinations of prior runs, if any, cannot be referenced in this restart.

It should be noted that the printout options given for the referenced load cases should correspond to output quantities which are requested in the given *LDCASE data group in this run. For example, if the stress computation key (KSTR) is set to zero, no printout of stresses is possible regardless of what the same key was set to for the same load case in the previous run.

Static analysis restart type 3

This restart may be used to analyze “new” load cases with possibly different displacement boundary conditions from those of the last load case of the previous run. At least one load case must be given. This restart may be used for load combination of the “new” and/or “old” load cases. However, the response quantities to be combined should have been computed in previous run(s) for the referenced old load case (otherwise, use restart 2 first).

The load case identification numbers specified for the new load cases in this run must be unique and must not have been used before for old load cases (or load combinations) in previous runs. The default load case ID number is the same as in a fresh run (see *LDCASE in [Section 7.1.1](#)), that is, the “previous” load case ID plus 1. For the first new load case, however, the “previous” ID is the maximum identification number of all prior load cases and load combinations.

As mentioned before this restart should be thought of as a continuation of the previous runs as if everything were submitted in a single run. Therefore, the boundary conditions of the first “new” load case in this restart run, if not explicitly given via a *SPDISP data group, are defaulted to the last “new” load case processed in prior runs (not necessarily in the

preceding run which may have been a restart 2 run, which does not introduce new load cases).

It is noted that the “new” load cases throughout the restart history should be ordered in such a way that the number of new decompositions of the stiffness matrix are minimized and that repeated decompositions for the same set of displacement boundary conditions are avoided ([Section 3.11.2](#)). An option that requires user’s action regarding the decomposed-stiffness file, file 30, is described below. This option is specified via the executive command ‘RESTART’, and may be used to keep multiple copies of file 30, each for a different set of displacement boundary conditions. However, this option should be used with care and it is the user’s responsibility to copy and rename the file.

The parameter concerning file 30 determines whether a new decomposition will be initiated for the “first” load case in this run. It has the form: “OLD30”, “NEW30”, or “USE30”.

When the option “OLD30” is used (this is the default option), file 30 from a previous run is supplied and will be reused for the first load case if its displacement boundary conditions are similar to those of the last new load case in prior runs. Otherwise a new decomposition is initiated and file 30 is overwritten. (Refer to [Section 3.11.2](#) for classification of similar and different boundary conditions).

When the option “NEW30” is used, a new file 30 is created and a new decomposition is initiated for the first load case in this run. The new file 30 may be saved using the command ‘SAVEFILE’, otherwise it is considered as a scratch file. It is noted that this option may be used if:

- ❑ File 30 from a prior run was not saved, or
- ❑ The user knows that the first load case of the current restart run has a different set of displacement boundary conditions from the last new load of prior runs, so that a new decomposition is necessary anyway and there is no need to overwrite the old file 30; perhaps the existing copy of file 30 may be reused in a subsequent restart run (using the option “USE30”).

When the option “USE30” is used, file 30 from a prior run is supplied and will be reused for the first load case in this run regardless of whether the displacement boundary conditions are similar to those of the last new load case in prior runs or not. When using this option, it is the user’s responsibility to ensure that the supplied file corresponds to the set of boundary conditions being used.

It should be noted that all of the above options apply to the first load case only. For subsequent load cases, file 30 may be reused as is, or overwritten depending on whether a new (and different) set of displacement boundary conditions is specified.

5.5.5 Special Features and Requirements in Eigenvalue Analysis Restarts

It should be noted that if an eigenvalue restart is performed on the same set of files 26 and 27 following a modal dynamic analysis run, the results of the modal dynamic analysis are overwritten. Thus, if the user wants to access the modal dynamic analysis results, e.g. for post-plotting, the necessary files should be copied before the eigenvalue restart run.

Eigenvalue Analysis Restart 2

This restart feature is useful in modal dynamics run. With this option, user can

- (i) run eigenvalue analysis again with *MODEOUT and *EIGOUT cards activated, in order to obtain the modal stresses, reactions, internal forces and/or pseudo static solution of missing mass correction.
- (ii) run eigenvalue analysis again with *MODEOUT and *EIGOUT cards reactivated, in order to obtain the modal stresses, reactions, internal forces and/or pseudo static solution of missing mass correction with different definitions of *EIGCNTL , *MODEOUT and *EIGOUT cards.

If pseudo static solution is activated, a new pseudo static solution is computed and overwrites the one from the previous run. Since the pseudo static computation requires stiffness and mass matrix, both file 23 and 24 should be saved from the previous run. If frontal solver is used in the previous run, restart 2 will replace the frontal solver by the sparse matrix solver internally.

Eigenvalue Analysis Restart 4

In eigenvalue restart 4, the element stiffness matrices from a linear of nonlinear static run are utilized for the eigenvalue extraction. The element mass matrices are calculated in this run. Thus the material density (or alternatively a system of equivalent point mass elements) should be defined in the original run. The displacement boundary conditions should be defined in this restart, otherwise a free-free structure is assumed.

This restart following a nonlinear static analysis may be used to find the natural frequencies of a prestressed structure (e.g., spinning structure) as described in [Section 3.16](#).

5.5.6 Special Features and Requirements in Buckling Analysis Restarts

It is mentioned in [Section 3.6](#) that buckling analysis is a two-pass analysis. The first pass is a static analysis which determines the stresses for a given reference set of loads. The second pass is an eigenvalue analysis which computes the buckling load factors (eigenvalues) and the corresponding mode shapes (eigenvectors). For a fresh run (restart=0) and for restart = 1 run, the procedure is automated such that the two passes are made in one run.

Buckling Analysis Restart 4

For buckling restart type 4, only the second pass is performed since the first pass is done in the previous static analysis run, as shown in [Table 5.11](#). Since the reference static run may have several load cases, the user should specify in the input data the referenced load case for which buckling analysis is required. This is achieved by specifying the *LDCASE card with a load case-id that corresponds to the referenced load case of prior runs. If no *LDCASE card is given, the last new load case of the previous static analysis run(s) is used as the reference load case. The displacement boundary conditions of the referenced load case are used; new definition is not allowed.

Buckling restart 4 may be utilized to perform buckling analysis when one set of loads is held constant (e.g., dead weight of a structure) while another set of loads is varied from zero to the point of structural instability (e.g., live loads). The procedure is as follows:

- ❑ A geometric nonlinear static analysis is performed under the first set of loads to be held constant. File 24, containing the element stiffness matrices (which include the stiffening effect of these loads), is saved from this run.
- ❑ A linear static analysis is carried out for the second set of loads. Gauss point stresses, which are needed in buckling analysis, are saved using the *POSTCNTL data group. Files 26 and 27 are saved from this run.
- ❑ A buckling analysis restart 4 is then performed to obtain buckling load factors. Files saved from the first two runs are accessed in this run by providing the same file name prefix. The buckling load factors thus computed, when applied to the second set of loads with the first set of loads held constant, will cause elastic instability.

5.5.7 Special Features and Requirements in Nonlinear Static Analysis Restarts

As shown in [Table 5.11](#), nonlinear static restarts can only be performed from a previous nonlinear static run.

The *TIMEAMP data group describes time-amplitude curves for subsequent reference in the analysis data. If this group is not provided in the restart run, the curves defined in previous runs are used. Otherwise, the union of the new specified curves and the old ones are stored. A new time-amplitude curve overrides an old one if it has the same curve identification number.

Nonlinear Static Analysis Restart 5

The analysis restarts from the last converged and saved load step of the previous run (see *NLOUT group, [Section 7.5.2](#) for frequency of saving results). Some executive commands, e.g., NLTYPE, cannot be changed in this restart, refer to [Table 5.13](#). This restart should be considered as a continuation of the last saved solution. As such, a *EVENT group must be given; event parameters may be changed or added. New load types may be activated but the displacement boundary conditions cannot be changed. The beginning time for the first event in this run is internally set to the end time of the last saved step in the previous run.

An analysis which runs for a long time, if divided into several events can run one event at a time as a restart from the previous event. This helps the user in several ways. The results can be checked and evaluated after each event. Also, the event and output parameters can be modified and copies of the restart files ([Table 5.11](#)) can be made. These copies of files are useful in case of computer system failure during a run, or when the user wants to restart from the last step of an earlier event.

5.5.8 Special Features and Requirements in Direct Transient Dynamic Restarts

The restart run for direct transient dynamic analysis is primarily a continuation of the previous run. Similar to nonlinear static analysis restart 5, a *EVENT group must be given and the event parameters may be changed or added. New load types may be activated but the boundary conditions cannot be changed. If the time step size, β and γ in the restart run is the same as that in the previous run, the option “OLD30” (see executive command ‘RESTART’,

[Section 5.5.3](#)) may be used to reuse the decomposed matrix on file 30. On the other hand, if one of the parameters is changed, a new decomposition may be done using the option “NEW30”. The option USE30 implies that the program should unconditionally use the file 30 supplied by the user for the first event in the Restart 5 run. Options OLD30, NEW30, and USE30 are not applicable to nonlinear direct transient analysis. A fresh decomposition is unconditionally performed for a NLTRANSIENT restart 5 run.

The initial displacements/velocities/accelerations for the first event of the restart 5 run are assumed to be the same as the final displacement/velocities/accelerations, respectively, of the last event of the run immediately preceding. Any entries in the *INITIAL data group if input by the user in the restart file are ignored.

5.5.9 Special Features and Requirements in Heat Transfer Restarts

The heat transfer analysis restart types 1 and 2 are used for transient and steady state analysis respectively. Table 5.15 gives the applicable model and analysis data groups for the heat transfer restarts.

Transient Heat Transfer Analysis Restart 1

Restart type 1 is used for continuation of a transient heat transfer analysis run. The beginning time for this restart is the last saved time step of the previous run. The previous run may be a scratch run (restart = 0) or a restart run (restart = 1). In nonlinear analysis, the converged solution from the last saved time step of the previous run is used. In this restart, new loading conditions (e.g. distributed heat flux) may be used (specified temperature boundary conditions cannot be changed).

Static Heat Transfer Analysis Restart 2

Restart type 2 is primarily used for steady state nonlinear analysis requiring several iterations for convergence. For restart type 2, the conductivity matrices saved in the previous run are accessed and the solution proceeds further. The model and analysis data may not change excepting the control parameters governing the convergence.

Table 5.11: Description of structural restarts^(a)

Restart Type No	Present Analysis Type	Previous Analysis Type	Files Saved in Previous Run	Description
0	All	—	—	The executive command 'RESTART = 0' designates a fresh (scratch) run. This is the default option
1	ST,EV or BU	ST,EV or BU	26	Restart from a processed model data of a previous run, wavefront reduction is bypassed. The new analysis data given replaces the old one. Normally used to restart from a scratch check run.
	NL	NL	26	
	LT	LT	26	
	NT	NT	26	

Restart Type No	Present Analysis Type	Previous Analysis Type	Files Saved in Previous Run	Description
2	ST	ST	24, 26, 27, 28	Post-solution computation (e.g., stresses and internal forces) of previous load case(s) and/or load combination of old cases (not available for axisymmetric elements with non-axisymmetric loading, NKTP = 34, 37).
	EV	EV	22, 23, 24, 26, 27	Post-solution computation (e.g., stresses, reactions and internal forces) for any mode computed in the previous run, useful for computation of modal quantities of modes to be selected for subsequent modal dynamics run. However, if the required 22, 24, 26, 27 files are available from the NISA Eigen analysis, modal dynamics execution will automatically call the Eigen restart execution and continues with the modal analysis.
3	ST	ST	24, 26, 27, 28, (30), 7(61 to 69)	Processing of new loading conditions, possibly with new B.C. and load combination of new and previous cases. The options "OLD30", "NEW30", or "USE30" determine whether a new decomposition is initiated for 1st new load case (see Section 5.5.4 and the command 'RESTART' in Section 5.3). Displacement constraints of 1st new load case default to last new load case in prior runs (not available for axisymmetric elements with non-axisymmetric loading, NKTP = 34, 37).

Input Setup and Executive Commands

Restart Procedures

Restart Type No	Present Analysis Type	Previous Analysis Type	Files Saved in Previous Run	Description
4	BU	ST	24, 26, 27	Buckling restart from a linear static analysis run to compute buckling load factors and mode shapes. Previous run may have included multiple load cases, buckling analysis is performed for only one load case specified by user. Displacement constraints of the referenced static load case may not be changed.
	EV	ST or NL	24, 26, 27	Vibration eigenvalue restart from a linear or nonlinear static analysis run. Only mass matrix is computed, stiffness matrix from previous run is used (Mass density should be defined in the fresh run). Useful for vibration analysis of initially stressed structures, e.g., rotating structures. Displacement constraints must be given for the restart run, otherwise a free-free structure is assumed.
5	NL	NL	26, 27, 37	Nonlinear static restart from the last converged and saved load step of a previous nonlinear static run. Beginning time of 1st event in the current run defaults to end time of last converged and saved time step of previous run. New time-amplitude data may be specified and are appended to old time-amplitude curves. Event parameters can be changed or added. New load types may be introduced. Displacement B.C. may not be changed from that of previous run.
	LT	LT	26, 27, (30), 37, (61 to 69)	Direct transient dynamics restart from the last event of a previous direct transient run. New time-amplitude data may be specified and are appended to old time-amplitude curves. New load types may be introduced. Displacement B.C. and history request data may not be changed. In LT, the option 'OLD30' can be used to bypass new decomposition if the time step size is not changed. Otherwise, the option 'NEW30' may be used (see Section 5.5.8 and the command 'RESTART' in Section 5.3)

Restart Type No	Present Analysis Type	Previous Analysis Type	Files Saved in Previous Run	Description
	NT	NT	26, 27, 37	Non linear direct transient restart from last event of a previous nonlinear transient run. New time-amplitude data may be specified and are appended to old time-amplitude curves. New load types may be introduced. Displacement BCs and history request data (*HIS-TOUT) cannot be changed. Options OLD30, NEW30, and USE30 are not applicable. A fresh decomposition is always carried out.

- (a) ST: Linear static EV: Eigenvalue BU: Buckling
 NL: Nonlinear static LT: Linear direct transient NT: Nonlinear direct transient

Table 5.12: Description of heat transfer restarts^(a)

Restart type no	Present analysis type	Previous analysis type	Files saved in previous run	Description
0	All	-	-	The executive command 'RESTART = 0' designates a fresh (scratch) run. This is the default option.
1	TH	TH	26	Transient heat transfer restart from a previous transient heat run. Starting time of the restart run proceeds from end time of previous run. If *STEPSIZE data was used in previous run, new *STEPSIZE data must be defined for the restart run.
2	SH	SH	23, 26	Steady state heat transfer restart from a previous steady state run. Conductivity matrices saved in previous run are reused.

- (a) SH: Steady state heat transfer TH: Transient heat transfer

Table 5.13: Applicable executive commands in restarts^(a)

Analysis Type	ST				EV				BU			NL			LT/NT			SH		TH	
Command \ Restart	0	1	2	3	0	1	2	4	0	1	4	0	1	5	0	1	5	0	2	0	1
ANALYSIS	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x
AUTOCONSTRAINT	x	-	-	-	x	-	-	-	x	-	-	x	-	-	x	-	-	-	-	-	-
BLANKCOMMON	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x
CONMETHOD	-	-	-	-	-	-	-	-	-	-	-	x	-	-	-	-	-	-	-	-	-
CTAL	-	-	-	-	-	-	-	-	-	-	-	x	-	-	-	-	-	-	-	-	-
EIGENEXTRACTION	-	-	-	-	x	x	-	x	x	x	x	-	-	-	-	-	-	-	-	-	-
ELEMENTECHO	x	-	-	-	x	-	-	-	x	-	-	x	-	-	x	-	-	x	-	x	-
EXECUTION	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	-	x	x
FILENAME	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x
FSIZ	x	x	-	x	x	x	-	x	x	x	x	x	x	x	x	x	x	-	-	-	-
GEOMPROPERTIES	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	-	-	-	-
INITIALTEMPERATURE	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	x	-
MASSFORMULATION	-	-	-	-	x	x	-	x	-	-	-	-	-	-	x	x	-	-	-	x	-
MAXCPU TIME	-	-	-	-	-	-	-	-	-	-	-	x	x	x	x	x*	x*	-	-	-	-
NLTYPE	-	-	-	-	-	-	-	-	-	-	-	x	-	-	x*	-	-	-	-	-	-
NODEECHO	x	-	-	-	x	-	-	-	x	-	-	x	-	-	x*	-	-	x	-	x	-
PATH	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x
PAT1	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x
PAT2	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x
PAT3	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x
PAT4	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x
ORTHOTROPIC	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	x	-	x	-

Analysis Type	ST				EV				BU			NL			LT/NT			SH		TH	
Command \ Restart	0	1	2	3	0	1	2	4	0	1	4	0	1	5	0	1	5	0	2	0	1
REFCONFIG	-	-	-	-	-	-	-	-	-	-	-	x	-	-	x*	-	-	-	-	-	-
RESEQUENCE	x	-	-	-	x	-	-	-	x	-	-	x	-	-	x	-	-	x	-	x	-
RESTART	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x
SAVEFILE	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x
SIGMA	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	x	-	x	-
SORTSTRESS	x	x	x	x	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-
SOLV	x	x	x	-	-	-	-	-	-	-	-	x	x	x	x	x	x	x	x	x	x
SRAD	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	x	x	x	x
STEP	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	x	x
UFIJ	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	x	x	x	x
WARNING	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x

- (+) ST: Linear static EV: Eigen value BU: Buckling
 NL: Nonlinear static LT: Linear direct transient NT: Nonlinear direct transient
 SH: Steady state heat transfer TH: Transient heat transfer
 x: Applicable command -: Executive command not applicable
 *: Applicable to nonlinear direct transient (NT) only

Table 5.14: Applicable data groups in structural restarts^(a)

MODEL DATA^(a)																	
ST....			EV....			BU..	NL..	LT....				
restart =	0	1	2	3	0	1	2	4	0	1	4	0	1	5	0	1	5
*HISTOUT	-	-	-	-	-	-	-	-	-	-	-	-	-	-	x	x	-
*TIMEAMPL	-	-	-	-	-	-	-	-	-	-	-	x	x	x	x	x	x
*TITLE	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x
*SETS	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x

ANALYSIS DATA																	
<u>ANALYSISCONTROL DATA</u>																	
ST....			EV....			BU....	NL....	LT....				
restart =	0	1	2	3	0	1	2	4	0	1	4	0	1	5	0	1	5
*CONVERGENCE	-	-	-	-	-	-	-	-	-	-	-	x	x	x	-	-	-
*EIGCNTL	-	-	-	-	x	x	-	x	x	x	x	-	-	-	-	-	-
*EVENT	-	-	-	-	-	-	-	-	-	-	-	x	x	x	x	x	x
*INITIAL	-	-	-	-	-	-	-	-	-	-	-	-	-	-	x	x	-
*LCTITLE	x	x	x	x	-	-	-	-	x	x	-	x	x	x	x	x	x
*LDCASE	x	x	x	x	-	-	-	-	x	x	x	-	-	-	-	-	-
*LDCOMB	x	x	x	x	-	-	-	-	-	-	-	-	-	-	-	-	-
*MODEOUT	-	-	-	-	x	x	x	x	x	x	x	-	-	-	-	-	-
*STEPsize	-	-	-	-	-	-	-	-	-	-	-	x	x	x	x	x	x

<u>FOURIER ANALYSIS DATA</u>										
ST....EV....BU....NL....LT....					
restart =	0 1 2 3	0 1 2 4	0 1 4	0 1 5	0 1 5					
*ANGSEC	x x - -	- - - -	- - - -	- - - -	- - - -					
*FRCNTL	x x - -	- - - -	- - - -	- - - -	- - - -					
*FRCOEF	x x - -	- - - -	- - - -	- - - -	- - - -					
<u>BOUNDARY CONDITIONS AND LOADING</u>										
ST....EV....BU....NL....LT....					
restart =	0 1 2 3	0 1 2 4	0 1 4	0 1 5	0 1 5					
*BEAMLOAD	x x - x	- - - -	x x -	x x x	x x x					
ST....EV....BU....NL....LT....					
*BODYFORCE	x x - x	- - - -	x x -	x x x	- - -					
CFOLLOWER	- - - -	- - - -	- - - -	x x x	x x* x*					
*CFORCE	x x - x	- - - -	x x -	x x x	x x x					
*L1	x x - x	- - - -	x x -	x x x	x x x					
*NDTEMPDIF	x x - x	- - - -	x x -	x x x	- - -					
*NDTEMPER	x x - x	- - - -	x x -	x x x	- - -					
*PRESSURE	x x - x	- - - -	x x -	x x x	x x x					
*SPDISP	x x - x	x x - x	x x -	x x -	x x -					
*TBODYFORCE	- - - -	- - - -	- - - -	- - - -	x x x					
<u>OUTPUT CONTROL DATA</u>										
ST....EV....BU....NL....LT....					
restart =	0 1 2 3	0 1 2 4	0 1 4	0 1 5	0 1 5					
*EIGOUT	- - - -	x x x x	x x x	- - -	- - -					
*I5	x x x x	x x x x	x x x	x x x	- - -					

Input Setup and Executive Commands

Restart Procedures

*N5	X X X X		X X X	X X X	- - -
*NLOUT	- - - -	- - - -	- - -	X X X	X X X
*POSTCNTL	X X X X	- - - -	X X -	- - -	- - -
*PRINTCNTL	X X X X	X X X X	X X X	X X X	X X X
*REGIONS	X X X X	X X X X	X X X	X X X	X X X
*SFDCOMP	X X X X	X X X X	X X X	X X X	X X X
*STRSFILTER	X X X X	X X X X	X X X	X X X	X X X
DATA DECK TERMINATOR					
ST....EV....BU....NL....LT....
restart =	0 1 2 3	0 1 2 4	0 1 4	0 1 5	0 1 5
*ENDDATA	X X X X	X X X X	X X X	X X X	X X X

(a) Data groups are listed in alphabetical order for each block.

Proper sequence must be followed, however, when preparing data.

ST: Linear static

EV: Eigen value

BU: Buckling

NL: Nonlinear static

LT: Linear direct transient

NT: Nonlinear direct transient

x: valid data group

-: Data group not applicable

(b) All other model data are not applicable in restarts.

(*) Applicable to nonlinear direct transient (NT) only

Table 5.15: Applicable data groups in structural restarts⁽⁺⁾

MODEL DATA⁽⁺⁺⁾				
	-STEADY STATE-		-TRANSIENT-	
	restarts = 0	2	0	1
*MATHEAT	x	-	0	-(1)
*PCHANGE1	-	-	x	x
*SETS	x	-	x	x
*TEMPFN	x	-	x	x
*TIMEAMPL	-	-	x	x
*TITLE	x	x	x	x
ANALYSIS DATA				
<u>ANALYSIS CONTROL DATA</u>				
	-STEADY STATE-		-TRANSIENT-	
	restarts = 0	2	0	1
*HEATCNTL	x	x	x	x
*STEPsize	-	-	x	x
*TIMEINTEG	-	-	x	x

<u>BOUNDARY CONDITIONS AND LOADING</u>				
	-STEADY STATE-		-TRANSIENT-	
	restarts = 0	2	0	1
*INITEMP	-	-	x	-
*SPTEMP	x	-	x	x (2)
*CFLUX	x	-	x	x

Input Setup and Executive Commands

Restart Procedures

*DFLUX	x	-	x	x
*ELHEATGEN	x	-	x	x
*NDHEATGEN	x	-	x	x
*CONVBC	x	-	x	x
*RADBC	x	-	x	x
*RADSURFACE	x	x	x	x
<u>OUTPUT CONTROL DATA</u>				
		-STEADY STATE-		-TRANSIENT-
*PRINCNTL	x	-	x	x
*TEMPHISTORY	-	-	x	x
*TEMPOUT	-	-	x	x
<u>DATA DECK TERMINATOR</u>				
		-STEADY STATE-		-TRANSIENT-
	restarts = 0	2	0	1
*ENDDATA	x	x	x	x

- (+) Data groups are listed in alphabetical order for each data block.
Proper sequence must be followed, however, when preparing data.
x: valid data group -: Data groups not applicable
- (++) All other model data are not applicable in restarts
- (1) If material data is temperature dependent, only the curve-id can be changed.
- (2) The constrained degrees of freedom cannot be changed. However, the specified temperature values and the time-amplitude curve-id may be changed.

Table 5.16: File status during and after restart run⁽⁺⁺⁾

File code	Applicable restart type	Applicable analysis type	Status
22	2	EV	Read-only, no change
23	2	SH	Overwritten for every iteration with current element conductivity matrices (depending on the parameter IFORM in the *HEATCNTL group)
24	2, 3	ST, EV	Read-only, no change
26	1, 2, 3, 4, 5	ALL1	Appended with new data, dictionary is updated if any progress is achieved (e.g., 1st static load case is successfully completed, or a step is converged in nonlinear statics)
	1, 2	ALL2	Appended with new data. Dictionary is updated after a successful completion of run
27	2, 3, 4, 5	ALL1	Appended with new results, old results are intact (for restart 1, a new file 27 is activated)
28	2	ST	Read-only, no change
	3	ST	Appended with new element load vectors, if any. Element load vectors for previous runs are intact.
30	3	ST	Overwritten or no change depending on options used (NEW30 in RESTART command) and BC.s. for new cases. No change if B.C.s. for all new cases are same as BC.s of last load case of previous run.
	5	LT	Overwritten or no change depending on options used for second parameter in RESTART command. If OLD30 option is used, then no change provided Δt , β and γ are same as in previous run; overwritten if not. No change if USE30 option is used. Overwritten if NEW30 option is used.
		NT	Overwritten unconditionally

Input Setup and Executive Commands

Restart Procedures

File code	Applicable restart type	Applicable analysis type	Status
37	5	NL	Overwritten whenever a load step is converged.
	5	LT/NT	Overwritten whenever a time step is completed.
61 to 69	3	ST	Same as file 30 (extension of file 30)
	5	LT	Same as file 30 (extension of file 30)
		NT	Same as file 30 (extension of file 30)

(++) ST: Linear static EV: Eigen value BU: Buckling
NL: Nonlinear static LT: Linear direct transient NT: Nonlinear direct transient
SH: Steady state heat transfer TH: Transient heat transfer
ALL1: ST, EV, NL, BU, LT, NT ALL2: SH, TH

Model Data

6.1 Introduction

The NISA data deck consists of distinct data blocks as has been mentioned in [Chapter 5](#). The data blocks must be ordered in the sequence shown in [Figure 5.1](#). The model data block represents the majority of the input data and is detailed in this chapter. It describes the physical characteristics of the finite element model, but without any loading conditions. The latter is described in the next chapter. The description of the model primarily consists of:

1. Element Data
 - (a) Element types selection: A table of the element types used in the model.
 - (b) Element definition: Specification of element connectivities (node list), and identification numbers (or pointers) for element types, material properties, and real constant (geometric properties) table.
 - (c) Element real constants: For example, thickness for a shell element, moment of inertia for a beam element, and spring constant for a spring element. Refer to NISA element library for the required list of real constants for each element type.
 - (d) Composite lamination data: This is only required for laminated composite elements and includes specification of number of layers and lamination angles.

2. Nodal Data

- (a) Definition of local coordinate systems
- (b) Nodal coordinates
- (c) Specification of local displacement coordinate system at nodes: This is used to refer the components of displacements and forces at any node to any desired coordinate system.

3. Material Data

This includes specification of material properties, and information about principal material axes for orthotropic material. Several linear and nonlinear material models are available. Refer to NISA element library for available material models and required properties for each element type.

4. Kinematic Constraints Data

This includes specification of kinematic relationships between two or more degrees of freedom (displacements or temperatures), for example, multi-point constraints, rigid links, and coupled temperatures. This data is *independent* of load cases. Boundary conditions and specified displacements are part of the analysis data ([Chapter 7](#)), and can be changed from one load case to another.

5. Miscellaneous Data

This includes the definition of sets of node or element numbers, which may be referenced in the analysis data for selective output requests. It also includes the definition of temperature dependency and time-amplitude curves, which may be referenced in some analysis data groups to specify temperature or time variation of a particular quantity.

The above input items of the model data are arranged in distinct data groups. Each data group consists of a group ID card followed by its free format data, which may consist of one or more card sets, each card set may consist of one or more cards. The free format rules are given in [Section 5.2](#). Each group has a descriptive identification name indicating the function of the data group. For example, the element definition is given in the *ELEMENTS data group, the nodal coordinates in *NODES data group.

Two format types, Regular and Large, can be selected to define the element and node IDs. In the regular format the IDs can have a maximum of 6 digits, where as for the large format the maximum is 8 digits. As default NISA reads in the regular format. Large format can be selected by specifying the executive card IDNU (see [Chapter 5](#))

The detailed description of the data groups is presented in this chapter in the sequence shown in [Table 6.1](#). This sequence is adopted according to the function of each data group as shown above. For easy reference, an alphabetical list of *all* data groups available in NISA is given in [Table 5.4](#). It should be noted that the data groups may be arranged in the model data quite arbitrarily, but with the following exceptions:

1. The *ELTYPE data group must precede the *ELEMENTS data group.
2. It is recommended that the *TITLE data group be the first data group in the model data set. This will allow the title of the problem being analyzed to be printed on every page of the output file.

It should also be noted that *not* all of the data groups given in this chapter need to be present in the model data for a particular problem. Some data groups are always required and some data groups are optional. For example, the *ELTYPE, *ELEMENTS and *NODES data groups are always required, whereas the *LAMSQ2 data group is only required if there are composite elements in the model. Whether a data group is always required or not is explicitly stated in the description of each data group.

Table 6.1: List of model data groups

Section No. and Group ID ⁽¹⁾	Description	Applicable Analysis Types ⁽²⁾
<i>6.2 Title</i>		
6.2.1 *TITLE	Problem title	ALL1
<i>6.3 Element Data</i>		
6.3.1 *ELTYPE	Element type selection	ALL1
6.3.2 *ELEMENTS	Element definition	ALL1
6.3.3 *E1	Alternate element definition	ALL1
6.3.4 *RCTABLE	Real constant table	ALL1
6.3.5 *LAMANGLE	Rotation angles for composites	ST, EV, BU, LT, NL, NT
6.3.6 *LAMSQ2	Composite lamination sequence	ST, EV, BU, LT, NL, NT
6.3.7 *BMDATA	Miscellaneous data for 3D general beam	ST, EV, BU, LT, NL, NT
6.3.8 *BMSECT	Cross section data for 3D general beam	ST, EV, BU, LT, NL, NT

Model Data

Introduction

Section No. and Group ID⁽¹⁾	Description	Applicable Analysis Types⁽²⁾
6.3.9 *NLSPRING	Force-deflection data for nonlinear springs	NL, NT
6.3.10 *WELEMENTS	Weld elements definition	ST
<i>6.4 Nodal Data</i>		
6.4.1 *LCSYSTEM	Local coordinate system	ALL1
6.4.2 *NODES	Nodal coordinates	ALL1
6.4.3 *G2	Alternate local system definition	ST, EV, BU, NL, LT, NT
<i>6.5 Material Data</i>		
6.5.1 *MATERIAL	Material property data	ST, EV, BU, NL, LT, NT
6.5.2 *MATDIR1	Orthotropic material axes at nodes	ALL1
6.5.3 *MATDIR 2	Orthotropic material axes at elements	ALL1
6.5.4 *MATHEAT	Heat transfer material properties	SH, TH
6.5.5 *HYPEREL	Hyperelastic material properties	NL, NT
6.5.6 *PLASTIC	Elastoplastic material properties	NL, NT
6.5.7 *APLASTIC	Anisotropic plastic material properties	NL, NT
6.5.8 *CREEP	Creep material properties	NL
6.5.9 *PDAMPING	Viscous damping properties	LT, NT
<i>6.6 Kinematic Constraints</i>		
6.6.1 *RIGLINK	Rigid element data	ST, EV, BU, LT, NL
6.6.2 *MPCEQN	Multi-point constraints	ST, EV, BU, LT, NL
6.6.3 *CPDISP	Coupled displacement data	ST, EV, BU, NL, LT
6.6.4 *CPTEMP	Coupled temperature data	SH, TH
<i>6.7 Miscellaneous Data</i>		
6.7.1 *SETS	Definition of sets	ALL1

Section No. and Group ID⁽¹⁾	Description	Applicable Analysis Types⁽²⁾
6.7.2 * VECTORS	Definition of vectors	ST, EV, BU, LT, NL, NT
6.7.3 * TEMPFN	Temperature curves	SH, TH
6.7.4 * TIMEAMP	Time-amplitude curves	NL, TH, LT, NT
6.7.5 * PCHANGE1	Enthalpy vs. temperature curve for phase change analysis	TH
6.7.6* CRACKNODES	Crack Nodes Definition Data	ST, NL
6.8 <i>Surface Contact Data</i>		
6.8.1 * CSURFACES	Definition of contact surfaces	NL
6.8.2 * CELEMENTS	Definition of contact elements	NL
6.9 <i>Moving Load Data</i>		
6.9.1 * MLOAD	Definition of moving load	ST
6.9.2 * MPATH	Definition of moving load path	ST

(1) Acceptable minimum abbreviations are in bold face

NL: Nonlinear static SH: Steady state heat transfer TH: Transient heat transfer
 LT: Linear direct transient NT: Nonlinear direct transient ST: Linear Static
 EV: Eigenvalue BU: Buckling
 ALL1: All analyses (ST, EV, BU, NL, LT, NT, SH, TH)

6.2 TITLE

6.2.1 *TITLE Data Group - Problem Title

This data group is always required and is used to print a title for the analysis. It is recommended that this data group be the first data group in the model data set so that the title of the problem may be printed on every page of the output file.

Group ID card:

*TITLE

Problem title card set:

Entry No: 1
Variable:

Problem title

Max char: 80

Entry Variable Description

1 - - alphanumeric title (up to 80 characters). This title will appear on the output file.

Note:

1. Up to 6 cards (80 characters per card) may be used for the title. The content of all cards will be printed when this data group is read in. The contents of the first card will be printed as a heading on all subsequent pages of the output file.

6.3 Element Data

6.3.1 *ELTYPE Data Group - Element Type Selection Table

This data group is always required, and must precede the *ELEMENTS (or *E1) data group. This group selects all types of elements used in the model, for subsequent reference in the *ELEMENTS data group.

Group ID card:

*ELTYPE

Element type selection card set:

Entry No:	1	2	3	4	5	6
Variable:	NSRL	NKTP	NORDR	NG1	NG2	NG3
Max Char:	2	3	2	1	1	1

Entry Variable Description

- | | | |
|-------|-------|---|
| 1 | NSRL | serial number to be used for element identification index (in ascending order starting with 1, maximum limit of 50). NISA elements are identified by two variables, NKTP and NORDR. NKTP indicates the element type number. NORDR selects the element shape functions (e.g., linear, parabolic), and hence, the number and location of its nodes. Element definitions in *ELEMENTS data group refer to this NSRL value as a shorthand notation for the NKTP and NORDR of the element. |
| 2 | NKTP | element type number - see NISA element library |
| 3 | NORDR | element order - see NISA element library |
| 4 - 6 | NGi | These parameters NG1, NG2 and NG3 may be specified in nonlinear static and nonlinear direct transient analyses only. They are the number of Gauss points in the ξ , η , and ζ directions, respectively. The default integration lattices for different element types are given in Chapter 2 . The default values for the number of Gauss points are used if NG1 is entered as zero or blank. Admissible values may be used otherwise. |

Model Data

Element Data

Note:

1. The same element NKTP and NORDR values may be specified more than once, with a different NSRL in this data group. The user may wish to do this so that, for example, changes to a model are easily recognized. It may also be useful in plotting, since an option is available in the DISPLAY program to sort elements by the NSRL values before plotting.

6.3.2 *ELEMENTS Data Group - Element Definition

This data group is always required, provided that the alternate form of the element definition, the *E1 data group, has not been used. (Note that both *ELEMENTS and *E1 data groups *must not* coexist in the same data deck).

Group ID card:

*ELEMENTS

Elements may be defined using *any* combination of three methods: individual element definition, first level generation and second level generation. Each element is defined by *two* sets of cards: the first set consists of one card which defines element parameters, and also generation parameters for first level generation; the second set defines the element node list and may consist of more than one card if the element has more than ten nodes.

For individual element definition, the first card set (one card) will include only element parameters. For first level element generation, the first card set will include both element and generation parameters. For second level element generation, the first card set will include only generation parameters and will be followed by the appropriate sets of cards defining the base element(s) either individually or via first level generation. Examples for various methods of element definition is given below.

	\$									
Entry No:	1	2	3	4	5	6	7	8	9	10
Variable:	NELID	MATID	NSRL	IDRC	KISO	NSET	NODINC	NELSET	NELINC	NRCS
Max Char:	6(8)	6	2	6	1	6	6	6	6	6

Card set 1: one card

<u>Entry Variable</u>	<u>Description</u>
—	Entries 1-5 are not needed for second level generation, —
—	enter zeros or start the card with a tab (\$) —
1 NELID	element ID number
2 MATID	material ID number. This number refers to a material with an ID number of MATID, defined in the *MATERIAL data group.
	<u>Exception:</u>

Model Data

Element Data

- for NKTP = 7, 32, 33 (composite elements), MATID is a lamination sequence index, and refers to *LAMSQ2 ID number (see note 9).
- 3 NSRL serial number of element identification index (see *ELTYPE data group).
- 4 IDRC real constant ID number. This number refers to a real constant table with an ID number of IDRC, defined in the *RCTABLE data group.

Exceptions:

- for NKTP = 7, 32, 33 (composite elements), enter zero, since the pointers to *RCTABLE data group will be set in *LAMSQ2 data group.
 - for NKTP = 39 (nonlinear beam), this number refers to a beam data identification number (IDBM) defined in the *BMDATA data group.
- 5 KISO element anisotropic index, see note 2
- = 0 - isotropic
 - = 1 - orthotropic
 - = 2 - laminated anisotropic (this value is automatically set for all composite elements).

— Remaining entries on this card are required only for —
— first or second level element generation —

— tab (\$) —

- 6 NSET the absolute value of NSET indicates the number of sets to be generated from each element in the base set, including the base set. A negative value for NSET activates second level generation.
- 7 NODINC node number increment
- 8 NELSET number of elements in the base set, see notes 3-5.
- 9 NELINC element number increment
- 10 NRCS Reference vector to define local system for element stress resultants
- NRCS > 0 Reference vector ID defined in *VECTOR data group.
 - = 0 Reference vector tangent to element ξ axis at element center (except for triangular element where reference vector tangent to local ξ axis at node 1)
 - = -1 Reference vector parallel to global X

- = -2 Reference vector parallel to global Y
- = -3 Reference vector parallel to global Z
- = -4 Reference vector parallel to global -X
- = -5 Reference vector parallel to global -Y
- = -6 Reference vector parallel to global -Z

Based on the given reference vector, the local system for stress resultants are defined as

$$V_2 = V_n \times V_{ref}$$

$$V_1 = V_2 \times V_n$$

where

V_{ref} = Reference vector

V_n = Vector normal to midsurface of the shell element

Card set 2: more than one card if the element has more than ten nodes

Entry No:	1	2	3	4 - 9	10
Variable:	NODE1	NODE2	NODE3	...	NODE10
Max Char:	6(8)	6(8)	6(8)	...	6(8)

Entry Variable

Description

- 1 NODE1 first node of the element connectivity list
- 2 NODE2 second node of the element connectivity list
- 3 NODE3 third node of the element connectivity list (if required)

Exceptions:

- for NKTP = 11, 12 (3-D beam elements), NODE3 is an end release code for NODE1, see note 5.
- for NKTP = 38 (3-D spring element), NKTP = 46 (3-D straight pipe element), and NKTP = 47 (3-D elbow element), NODE3 is an optional third node to define element local coordinate system, see notes 6, 8.

- 4 NODE4 fourth node of the element connectivity list (if required)

Exceptions:

- for NKTP = 11, 12 (3-D beam elements), NODE4 is an end release code for NODE2, see note 5.

Model Data

Element Data

- for NKTP = 38 (3-D spring element), NKTP = 46 (3-D straight pipe element), and NKTP = 47 (3-D elbow element), NODE4 is an optional vector ID number to define element local coordinate system, see notes 7, 8.
- 5 NODE5 fifth node of the element connectivity list (if required)
Exception:
 - for NKTP = 11, 12 (3-D beam elements), NODE5 is an optional third node to define element local coordinate system, see notes 6, 8.
 - for NKTP = 46, 47 (3D pipe and elbow elements), NODE5 is the end release code for first node of the element
- 6 NODE6 sixth node of the element connectivity list (if required)
Exception:
 - for NKTP = 11, 12 (3-D beam elements), NODE6 is an optional vector ID number to define element local coordinate system, see notes 7, 8.
 - for NKTP = 46,47 (3D pipe and elbow elements), NODE6 is the end release code for second node of the element
- 7 NODE7 seventh node of the element connectivity list (if required)
Exception:
 - for NKTP = 11 (3-D tapered beam), NODE7 is an identification number for a real constant table (*RCTABLE) for beam vertex offset from the mesh nodal points, if any.
- 8 NODE8 eighth node of the element connectivity list (if required)
Exception:
 - For NKTP= 11 (3-D tapered beam), NKTP=12 (3D general beam),
 - NKTP=46, 47 (3-D straight pipe and elbow element), NODE8 is the RCTABLE ID for partial fixity coefficients for NODE1. (see note 10)
- 9 NODE9 ninth node of the element connectivity list (if required)
Exception:
 - For NKTP= 11 (3-D tapered beam), NKTP= 12 (3-D general beam),
 - NKTP=46, 47 (3-D straight pipe and elbow element), NODE9 is the RCTABLE ID for partial fixity coefficients for NODE2.
- 10 NODE10 tenth node (if required), see note 1

Notes:

1. If the element has more than 10 nodes (e.g., the 20 node solid element), continue on additional cards in card set 2 to complete the element node list.
2. Refer to the element library for the available material models for each element type. Note that, regardless of the properties entered in *MATERIAL data group, the elements will be isotropic unless KISO = 1.
3. For first level element generation: (see example below)

If there is only one element in the base set (NELSET = 1), then only two sets of cards are required to generate NSET number of elements.

If there is more than one element in the base set (NELSET > 1), then the *first* base element along with the generation parameters will be defined by the first two card sets. Definition for additional elements in the base set should immediately follow. The total number of elements generated will be equal to NELSET*NSET.
4. For second level element generation, card set 1 is used to define the generation parameters, and should include only entries 6-9, i.e., start the card with a tab (\$). Immediately after the definition of the second level generation parameters, the NELSET number of base elements are defined on subsequent cards using individual definition or first level generation. See examples below.
5. For 3-D beam elements (NKTP = 11, 12), the end release codes are used to release *up to five* of the six degrees of freedom, defined in the *local element's* coordinate system, at either end of the element, NODE1 and NODE2. A released degree of freedom means that the connection between this DOF and the corresponding nodal point is removed. Any released degree of freedom *must* be associated with a non-zero stiffness. The release code is a six-digit number made up of 0's and 1's; the digit 0 represents a retained degree of freedom, and the digit 1 represents a released degree of freedom. The six digits in the release code correspond to respectively, UX, UY, UZ, ROTX, ROTY, and ROTZ in the element's coordinate system. A node with ROTX and ROTZ released will have the release code 000101, or 101. The release code 111111 is not allowed.
6. This third node should be defined in the *NODES data group and is used to define the local xy plane of the line element (NKTP = 11, 12, 38, 46 or 47), such that the element local x-axis is the line joining NODE1 and NODE2, and the third node lies in the local xy plane.
7. This number refers to a vector ID number whose global cartesian components are defined in *VECTORS data group. This vector is used to define the local xy plane of the line element (NKTP = 11, 12, 38, 46 or 47), such that the element local x-axis is the line joining NODE1 and NODE2 and the referenced vector lies in the local xy plane.

8. The local xy plane of the 3-D beam elements (NKTP = 11, 12), the 3-D spring element (NKTP = 38) or the 3-D pipe elements (NKTP = 46, 47) may be defined by one method *only*, either the third node specification (note 6), or the vector specification (note 7). Both definitions remain unchanged for first and second level generations (same as defined for the first base element). For NKTP = 12, these definitions will override the orientation angle or plane definition given in the real constant table, *RCTABLE data group.
9. For composite elements (NKTP = 7, 32, 33), the entry for MATID refers to identification number (LSID) in the *LAMSQ2 data group.
10. Partial fixity is a feature to transfer partial beam end forces instead of full beam forces to the joints (structure node) to simulate a plastic hinge. To activate this feature, the degree of freedom of the beam has to be released using end release code. A RCTABLE has to be specified for the six coefficients of partial fixity for each DOF of each beam node. If no RCTABLE is given for that node, the release DOFs are treated as an end release. For those DOFs without end release, the specified coefficients will have no effect. The coefficient of partial fixity is the ratio of the beam force transfers to the joint. It can vary from 0 to 1. 0 means end release and 1 means normal connection without partial fixity. To avoid numerical problem, any coefficient is greater than .999 will be treated as normal connection without partial fixity.

Example 6.2.1: Individual element generation

The following cards define *one* 15-node wedge element whose element ID is 10, material ID is 1, identification index is 1, anisotropic index is 1, and node list is 1 to 15. Comment cards are included for explanation.

```
**  
**Card set 1: defines element parameters, no generation parameters needed  
**  
10, 1, 1, 0, 1  
**  
**Card set 2: element node list; note that two cards are required since the element has more than 10  
**nodes  
1, 2, 3, 4, 5, 6, 7, 8, 9, 10,  
11, 12, 13, 14, 15  
**
```

Example 6.2.2: First level element generation

The 15 element mesh shown in [Figure 6.1](#) is generated using first level generation. The base set has 3 elements, which are defined individually. Five sets including the base set are required.

```
**
**Card set 1: defines element parameters, for the first element in the base set, along with first level
**generation parameters.
**Five sets are generated including the base set which contains 3 elements
**
  1, 1, 1, 1, 0, 5, 1, 3, 1
**
**Card set 2: defines node list for the first base element
**
  1, 2, 8, 7
**
**Second element in the base set. Two sets of cards are needed. No generation parameters are
**specified.
**
  6, 1, 1, 1
  7, 8, 14, 13
**
**Third element in the base set.
**
  11, 1, 1, 1
  13, 14, 20, 19
**
```

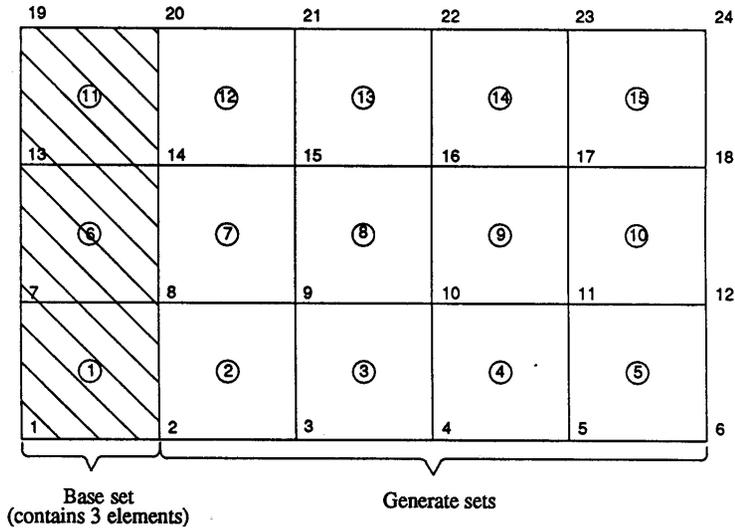


Figure 6.1: First level element generation

Example 6.2.3: Second level element generation

The 15 element mesh shown in [Figure 6.2](#) is generated using second level generation, 3 sets including the base set are required. The base set has 5 elements, which are defined using first level generation.

```

**
**Card set 1: defines second level generation parameters only 3 sets are to be generated
from a base set
**which contains 5 elements
**
  $-3, 6, 5, 5
**
**Card set 1: defines the base element parameters and generation parameters for first
level generation
**
  1, 1, 1, 1, 0, 5, 1, 1, 1
**
**Card set 2: defines node list for the base element of the first level generation
**
  1, 2, 8, 7
  
```

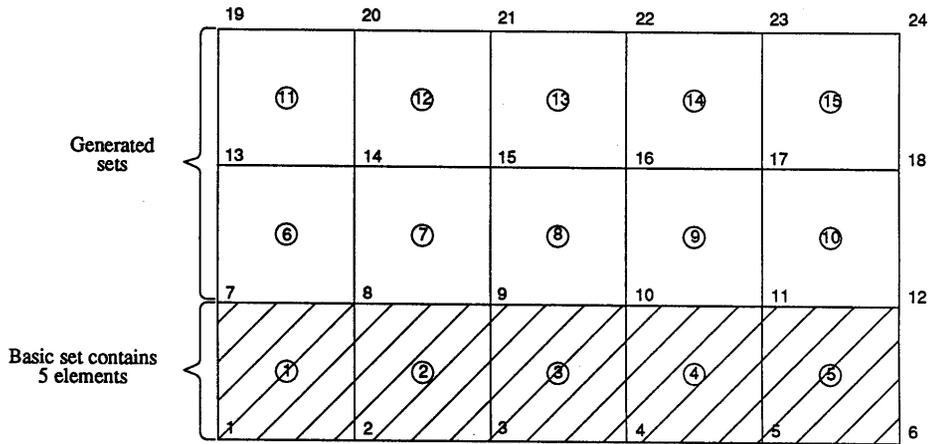


Figure 6.2: Second level element generation

6.3.3 *E1 Data Group - Alternate Form for Element Definition

This data group is required if the standard form for element definition, the *ELEMENTS data group, is not being used. (Note that both the *E1 and *ELEMENTS data groups must not coexist in the same data deck).

Users are advised to use the standard form of element definition, the *ELEMENTS data group, since this data group (*E1) will *not* be supported in future versions.

Group ID card:

*E1

Elements may be defined using *any* combination of three methods: individual definition, first level generation and second level generation. Each element is defined by *one* set of cards which consists of one card except if the element has more than eight nodes. Element definition includes specifying the element node list, element parameters and generation parameters, if any.

For first level element generation, the first card set will define the first base element node list and element parameters as well as the generation parameters. The remaining elements in the base set, if any, should immediately follow with no generation parameters. For second level generation, the first card set will define only generation parameters. All elements in the base set should immediately follow.

Card set 1: one card (except if the element has more than 8 nodes)

Entry No:	1	2	3	4-8	9	10	11	12	13
Variable:	NELID	NODE1	NODE2	...	NODE8	MATID	NSRL	IDRC	NSET
Max Char:	6	6	6	...	6	4	2	4	4

Entry No:	14	15	16
Variable:	NODINC	NELSET	KISO
Max Char:	6	3	1

Exceptions:

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
	——	Entries 1-12 are not needed for second level generation, ——
	——	enter zeros or start card with two tabs (\$\$) ——
1	NELID	element ID number
2	NODE1	first node of the element connectivity list
3	NODE2	second node of the element connectivity list
4	NODE3	third node of the element connectivity list (if required)
		<u>Exception:</u>
		– for NKTP = 11, 12 (3-D beam elements), NODE3 is an end release code for NODE1, see note 5.
		– for NKTP = 38 (3-D spring element), NODE3 is an optional third node to define element local coordinate system, see notes 6, 8.
5	NODE4	fourth node of the element connectivity list (if required)
		<u>Exception:</u>
		– for NKTP = 11, 12 (3-D beam elements), NODE4 is an end release code for NODE2, see note 5.
		– for NKTP = 38 (3-D spring element), NODE4 is an optional vector ID to define element local coordinate system, see notes 7, 8.
6	NODE5	fifth node of the element connectivity list (if required)
		<u>Exception:</u>
		– for NKTP = 11, 12 (3-D beam elements), NODE5 is an optional third node to define element local coordinate system, see notes 6, 8.
7	NODE6	sixth node of the element connectivity list (if required)
		<u>Exception:</u>
		– for NKTP = 11, 12 (3-D beam elements), NODE6 is an optional vector ID to define element local coordinate system, see notes 7, 8.
8	NODE7	seventh node of the element connectivity list (if required)
		<u>Exception:</u>
		– for NKTP = 11 (3-D tapered beam), NODE7 is an identification number for a real constant table (*RCTABLE) for beam vertex offset from the mesh nodal points, if any.

Model Data
 Element Data

9	NODE8	eighth node of the element connectivity list (if required), see note 1.
— tab (\$) —		
10	MATID	material ID number. This number refers to a material with an ID number of MATID, defined in the *MATERIAL data group.
<u>Exception:</u>		
		– for NKTP = 7, 32, 33 (composite elements), MATID is a lamination sequence index, and refers to the entry on *LAMSQ2 data group (see note 9).
11	NSRL	serial number of element identification index (see *ELTYPE data group).
12	IDRC	real constant ID number. This number refers to a real constant table with an ID number of IDRC, defined in the *RCTABLE data group.
<u>Exception:</u>		
		– for NKTP = 7, 32, 33 (composite elements), enter zero, since the pointers to *RCTABLE data group will be set in *LAMSQ2 data group.
		– for NKTP = 39 (nonlinear beam), this number refers to a beam data identification number (IDBM) defined in the *BMDATA data group.
— tab (\$) —		
	——	Entries 13-14 are required only for first ——
	——	or second level element generation ——
13	NSET	the absolute value of NSET indicates the number of sets to be generated from the base set, including the base set. A negative value of NSET activates second level generation.
14	NODINC	node number increment
15	NELSET	number of elements in the base set, see notes 3, 4.
16	KISO	element anisotropic index, see note 2
		= 0 - isotropic
		= 1 - orthotropic
		= 2 - laminated anisotropic (this value is automatically set for all composite elements)

Notes:

1. If the element has more than 8 nodes, (e.g., the 20 node solid element), complete the node list on additional cards (8 nodes per card).
2. Refer to the element library for the available material models for each element type. Note that regardless of the properties entered in *MATERIAL data group, the elements will be isotropic unless KISO = 1.
3. For first level element generation:
 - If there is only one element in the base set (NELSET = 1), then one set of cards (one card except if the element has more than 8 nodes) will be required to generate the number of elements equal to NSET.
 - If there is more than one element in the base set (NELSET > 1), then only the first element definition will contain the generation parameters, i.e., entries 13-15. Definition for additional elements in the base set, if any, should follow immediately. The total number of elements generated will be equal to NELSET * NSET.
4. For second level element generation, card set 1 is used to define the generation parameters and should include only entries 13-15, i.e. start the card with two tabs (\$\$). Immediately after the definition of the second level generation parameters, start the definition of NELSET base elements using individual or first level element generation.
5. For 3-D beam elements (NKTP = 11, 12), the end release codes are used to release *up to five* of the six degrees of freedom, defined in the local element's coordinate system, at either end of the element, NODE1 and NODE2. A released degree of freedom means that the connection between this DOF and the corresponding nodal point is removed. Any released degree of freedom *must* be associated with a non-zero stiffness. The release code is a six-digit number made up of 0's and 1's; the digit 0 represents a retained degree of freedom, and the digit 1 represents a released degree of freedom. The six digits in the release code correspond to UX, UY, UZ, ROTX, ROTY, and ROTZ, respectively, in the element's coordinate system. A node with ROTX and ROTZ released will have the release code 000101, or 101. The release code 111111 is not allowed.
6. This third node should be defined in the *NODES data group and is used to define the local xy plane of the line element (NKTP = 11, 12 or 38), such that the element local x-axis is the line joining NODE1 and NODE2, and the third node lies in the local xy plane.
7. This number refers to a vector ID number whose global cartesian components are defined in *VECTORS data group. This vector is used to define the local xy plane of the line element (NKTP = 11, 12 or 38), such that the element local x-axis is the line joining NODE1 and NODE2 and the referenced vector lies in the local xy plane.

Model Data

Element Data

8. The local xy plane of the 3-D beam elements (NKTP = 11, 12) or the 3-D spring element (NKTP = 38) may be defined by one method *only*, either the third node specification (note 6), or the vector specification (note 7). Both definitions remain unchanged for first and second level generations (same as defined for the first base element). For NKTP = 12, these definitions will override the orientation angle or plane definition given in the real constant table, *RCTABLE data group.
9. For composite elements (NKTP = 7, 32, 33), the entry for MATID refers to a table identification number in the *LAMSQ2 data group. These identification numbers have to be in ascending order beginning with one (see the *LAMSQ2 data group).

6.3.4 *RCTABLE Data Group - Real Constant Table

This data group is required if any element type used in the model requires real constants (e.g., shell thickness, cross-sectional properties, spring constants, etc.). Refer to the element library for the required real constants corresponding to the selected elements in *ELTYPE data group.

Group ID card:

*RCTABLE

Card set 1: Real constant index card

Entry No:	1	2	3	4
Variable:	IDRC	NUMRC	IFORM	IDCRV
Max Char:	6	2	1	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
--------------	-----------------	--------------------

- | | | |
|---|-------|--|
| 1 | IDRC | real constant table identification number. It is the number referred to in the *ELEMENTS data groups to define the element real constants. |
| 2 | NUMRC | number of real constants in this table to be input in the next card(s). |
| 3 | IFORM | key for number of entries per card in card set 2
= 0 (or blank) -8 entries per card
> 0 -4 entries per card |
| 4 | IDCRV | temperature dependency curve identification number <i>for heat transfer analysis only</i> . The temperature dependency curves are defined in *TEMPFN data group. It is used to identify the temperature dependency curve for (a) film coefficient in convection link element (NKTP = 149) or (b) emissivity in radiation link element (NKTP = 150). Enter zero if film coefficient or emissivity in these elements is not temperature dependent. |

Model Data
 Element Data

Card set 2: Real constant table card(s), entered according to IFORM

IFORM = 0

Entry No:	1	2	3	4	5	6	7	8
Variable:	RC1	RC2	RC3	RC4	RC5	RC6	RC7	RC8
Max Char:	10	10	10	10	10	10	10	10

OR

IFORM > 0

Entry No:	1	2	3	4
Variable:	RC1	RC2	RC3	RC4
Max Char:	20	20	20	20

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	RC1	first real constant
2	RC2	second real constant
...		
...		... as required (number of entries per card must be consistent with IFORM)
...		

Notes:

1. Use as many cards as needed in card set 2 to enter NUMRC number of real constants in this table. Data should be entered according to the IFORM parameter (8 entries per card if IFORM = 0 OR 4 entries per card if IFORM > 0).
2. If there is more than one real constant table in this data group repeat the same card set sequence (card set 1 and 2) to define the additional tables.

6.3.5 *LAMANGLE Data Group - Rotation Angles for Laminated Composite Elements

This data group is required if the model includes any composite element (NKTP = 7, 32, 33). Two sets of cards are required for a typical rotation angles table.

Group ID card:

*LAMANGLE

Card set 1: Rotation angle index card

Entry No:	1	2
Variable:	IDRT	NUMRT
Max Char:	4	2

<u>Entry Variable</u>	<u>Description</u>
-----------------------	--------------------

- | | | |
|---|-------|--|
| 1 | IDRT | rotation angle table ID (up to 4 digits). This is the number referred to in *LAMSQ2 data group to define the rotation angles at each node of an element. |
| 2 | NUMRT | number of rotation angles in this table to be input in the next card(s). |

Card set 2: Rotation angles table card

Entry No:	1	2	3	4	5	6	7	8
Variable:	RA1	RA2	RA3	RA4	RA5	RA6	RA7	RA8
Max Char:	10	10	10	10	10	10	10	10

<u>Entry Variable</u>	<u>Description</u>
-----------------------	--------------------

- | | | |
|-----|-----|--|
| 1 | RA1 | rotation angle (in degrees) at node 1 |
| 2 | RA2 | rotation angle (in degrees) at node 2 |
| ... | | ... |
| ... | | ... as required... |
| ... | | ... |
| 8 | RA8 | rotation angle (in degrees) at node 8 (if necessary) |

Model Data

Element Data

Notes:

1. If the element has more than eight nodes, use additional cards as required.
2. Rotation angles are measured with respect to the coordinate system defined with variable IDRA in *LAMSQ2 data group.
3. If there is more than one rotation angles table in this group repeat the same card set sequence (card set 1 and 2) to define additional tables.

6.3.6 *LAMSQ2 Data Group - Composite Lamination Sequence Data

This group is required if the model includes composite elements (NKTP = 7, 32, 33). It replaced the previously used data group *LAMSEQ. NISA II will continue to support the older data group, but users advised to use the *LAMSQ2 data group for all newer models.

This data group is used to set up lamination tables for laminated composite elements, which include layer thickness, fiber orientation, and material property ID for each layer of the laminate. Each table in *LAMSQ2 data group starts with a lamination sequence table ID. This ID will be referenced by the composite elements as MATID in *ELEMENT data group.

The data group consists of two card sets. The first card set, which contains one data card, defines lamination sequence table ID, number of layers, and choice of coordinate system for defining fiber orientation angle.

The second card set lists data for layer thickness, fiber orientation angle, and material ID for the laminated composite. Each card can have a maximum of 10 entries. Number of cards in the second set depends on how many cards are needed to specify 3*NLAY (no. of layers) of data. The first NLAY entries in this set are layer thickness table IDs (IDRC in *RCTABLE data group) from the top layer to the bottom layer, the next NLAY entries are the fiber orientation angle table IDs (IDRT in *LAMANGLE data group), and the last NLAY entries are the material table IDs (MATID in *MATERIAL data group). The layer numbering sequence for the laminated composite always starts from the top layer to the bottom layer of an element.

Group ID card:

*LAMSQ2

Card set 1: Lamination sequence table ID card

Entry No:	1	2	3	4
Variable:	LSID	NLAY	IDRA	IESHR
Max Char:	6	6	6	2

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
--------------	-----------------	--------------------

1	LSID	identification number of the lamination sequence.
---	------	---

- 2 NLAY number of layers
- 3 IDRA coordinate system ID for defining rotation angles, see examples below
- = 0 - right-hand-rule rotation about the normal to the shell surface, measured from the line connecting the first two nodes of the element (or from the tangent, at node 1, to the line connecting nodes 1 and 2, if the element has curved sides).
 - for NKTP = 7 (3-D composite solid element), right-hand-rule rotation about the normal to the bottom face (its mean plane if warped) of the element, measured from the first edge of the face (its projection on mean plane, if warped). See [Figure 4.7](#) and note 4 in [Section 4.8](#).
 - = 1 - right-hand-rule rotation about the global Z-axis, measured from the global X-axis. This option is not available for NKTP = 7.
 - = 2 - right-hand-rule rotation about the global X-axis, measured from the global Y-axis. This option is not available for NKTP = 7.
 - = 3 - right-hand-rule rotation about the global Y-axis, measured from the global Z-axis. This option is not available for NKTP = 7.
 - = 4 - right-hand-rule rotation about the normal to the shell surface, measured from the projection of the global X-axis onto the shell surface.
 - = 5 - right-hand-rule rotation about the normal to the shell surface, measured from the projection of the global Y-axis onto the shell surface.
 - = 6 - right-hand-rule rotation about the normal to the shell surface, measured from the projection of the global Z-axis onto the shell surface.
 - = 7- for NKTP = 7 (3-D composite solid element), right-hand-rule rotation about the normal to the top face (its mean plane if warped), measured from the first edge of the face (its projection on mean plane, if warped).
 - = 8 - for NKTP = 7 (3-D composite solid element), right-hand-rule rotation about the normal to the midsurface (its mean plane if warped), measured from the first edge of the midsurface (its projection on mean plane, if warped).

= -i - the number i is the local coordinate system ID (IDSYS) defined in *LCSYSTEM data group, such that the rotation angle is a right-hand-rule rotation about the third axis of system i (local z), measured from the first axis of system i (local x). Note that the system i is flagged with a negative sign here. This option is not available for NKTP = 7.

4 IESHR Key to activate equivalent transverse shear modulus computation (see note 3). This option is only available for element type NKTP = 32 and 33 in the linear static and linear transient dynamic analysis. It has no effect for other type of element and analysis type.

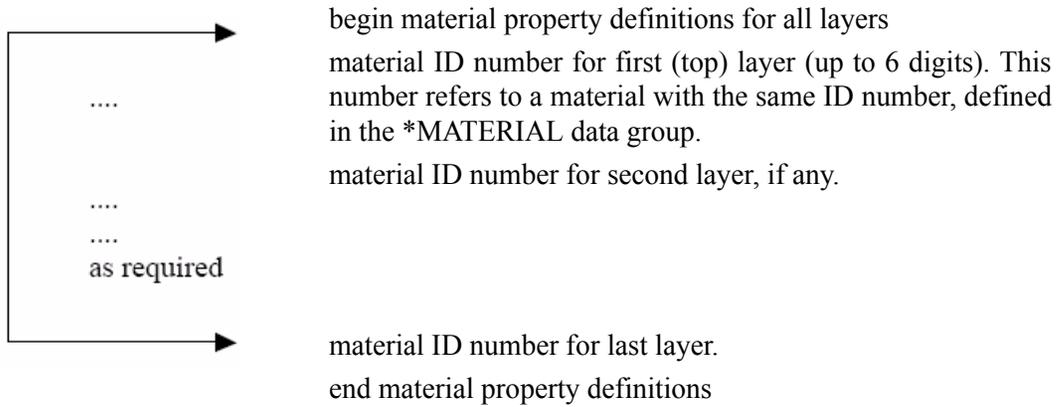
= 0 use theoretical stress-strain relationship (default)

= 1 use equivalent shear modulus for transverse shear properties.

Card Set 2: Lamination sequence property data card

Entry No:	1	2	3	4	5	6	7	8	9	10
Variable:	LS1	LS2	LS3	LS4	LS5	LS6	LS7	LS8	LS9	LS10
Max Char:	6	6	6	6	6	6	6	6	6	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
		begin thickness definitions for all layers
1	LS1	thickness table ID number for first (top) layer (up to 6 digits). This number refers to a table with an ID number of LS1, defined in the *RCTABLE data group (see note 1).
2	LS2	thickness table ID number for second layer, if any.....
	
	as required	
		thickness table ID number for last layer.
		end thickness definitions
		begin rotation angle definitions for all layers
	rotation angle table ID number for first (top) layer (up to 6 digits). This number refers to a table with the same ID number, defined in the *LAMANGLE data group.
	rotation angle table ID for second layer, if any.
	
	as required	
		rotation angle table ID number for last layer.
		end rotation angle definitions



Notes:

1. Three sets of ID are given in the second card set. From 1 to NLAY are for layer thickness ID defined in *RCTABLE data group. From NLAY+1 to 2*NLAY are for fiber orientation angle ID defined in *LAMANG data group. From 2*NLAY+1 to 3*NLAY are for material table ID defined in *MATERIAL data group. The total number of data required in the second card set is NLAY*3
2. The layer numbering sequence always starts at the top surface of an element.
3. The theoretical stress-strain relationship in transverse directions are $SYZ = GYZ \cdot ryz$ and $SXZ = GXZ \cdot rxz$ for each layer. If just one transverse component YZ is considered as an example, the resultant force $NYZ = \sum (GYZ \cdot ti) \cdot ryz$, where ryz is constant through the shell thickness based on linear variation of displacements and ti is the thickness of the layer. It is impossible to use a linear displacement field of the element to accurately represent multi-linear displacement field of the layers. In cases when difference in the shear modulus between the layers is large (e.g. one or more order of magnitude), then the theoretical formula significantly underestimates shear deformation ryz due to larger value of GYZ of one of the layers which dominates the sum of $(GYZ \cdot ti)$. In fact, the physical observation is that the material with lower value of GYZ should have the major contributions to the shear deformation. To alleviate this problem, an effective shear modulus is computed based on $G_{eff} = T_{tot} / \sum (ti / GYZ_i)$, where T_{tot} is the total thickness of the laminate. When all GYZ_i have the same value, $G_{eff} = GYZ_i$. The Effective G_{eff} may be a better representation for problems where shear deformation is large. In cases where theoretical formula is needed, a negative NLAY should be used to activate calculation of G based on the theoretical formulation.

Example 6.2.4:

Consider a flat plate lying in the global XY plane and modeled using 8 node laminated composite shell elements (NKTP = 32, NORDR = 2), as shown in Fig. 6.2.11. The plate has 4 layers. The lamination angles, thicknesses and material specification are shown in the figure. The lamination angles (45/0/90/-45) are about the global Z-axis and are measured from the global X-axis (i.e., IDRA = 1). The lamination data for this problem are as follows:

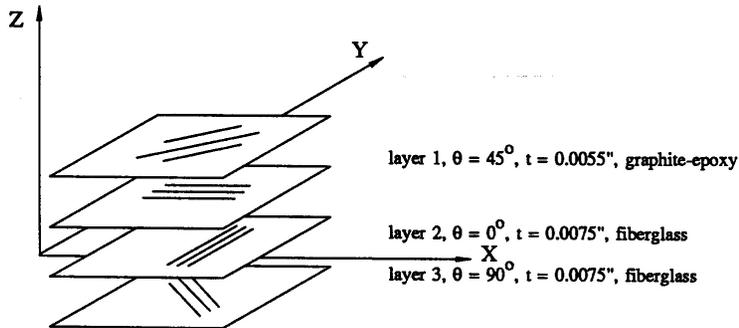


Figure 6.3: Composite flat plate

```

**
*LAMSQ2
** LSID, NLAY, IDRA, IESHR
   10,  4,  1,
** thickness ID angles ID material ID
   11,12,12,11, 21,22,23,24, 101, 102,
** material ID
   102, 101
**
*RCTABLE
**thickness table ID no.11
**
11, 8
0.0055, // // // // //
**thickness table ID no.12
**
12, 8
0.0075, // // // // //

```

```
**
*LAMANGLE
**lamination angle table ID no.21
**
21, 8
45.0, //
**lamination angle table ID no.22
**
22, 8
0.0, //
**lamination angle table ID no. 23
**
23, 8
90.0, //
**lamination angle table ID no.24
**
24, 8
-45.0, //
**
*MATERIAL
**material ID no.101, graphite-epoxy
**
EX, 1, 0, 25.0E6
**other material properties for graphite-epoxy
**
**material ID no. 102, fiberglass
**
EX, 2, 9.0E6
**other material properties for fiberglass
**

**
*LAMSQ2
** LSID, NLAY, IDRA, IESHR
    10, 4, 1,
** thickness ID angles ID material ID
    11,12,12,11, 21,22,23,24, 101, 102,
** material ID
```

Model Data

Element Data

```
102, 101
**
*RCTABLE
**thickness table ID no.11
**
11, 8
0.0055, /////  
**thickness table ID no.12
**
12, 8
0.0075, /////  
**
*LAMANGLE
**lamination angle table ID no.21
**
21, 8
45.0, /////  
**lamination angle table ID no.22
**
22, 8
0.0, /////  
**lamination angle table ID no. 23
**
23, 8
90.0, /////  
**lamination angle table ID no.24
**
24, 8
-45.0, /////  
**
*MATERIAL
**material ID no.101, graphite-epoxy
**
EX, 1, 0, 25.0E6
**other material properties for graphite-epoxy
**
**material ID no. 102, fiberglass
**
```

EX, 2, 9.0E6

**other material properties for fiberglass

**

Example 6.2.5:

To understand the need for more general ways to define fiber angles, consider the complication that arises in the problem of the previous example when the flat plate is changed to a cylindrical shell, as shown in Figure 6.4. The rotation angle of the top layer is still 45° . However, it is no longer a rotation of 45° about the global Z-axis measured from the global X-axis. It is a rotation of 45° about the normal to the shell surface measured from the projection of the global X-axis onto the shell surface (i.e., the projection of the global X-axis on the plane tangent to the shell surface at each node). Thus, IDRA may be set to 4 in the previous example to define the lamination angles for this problem.

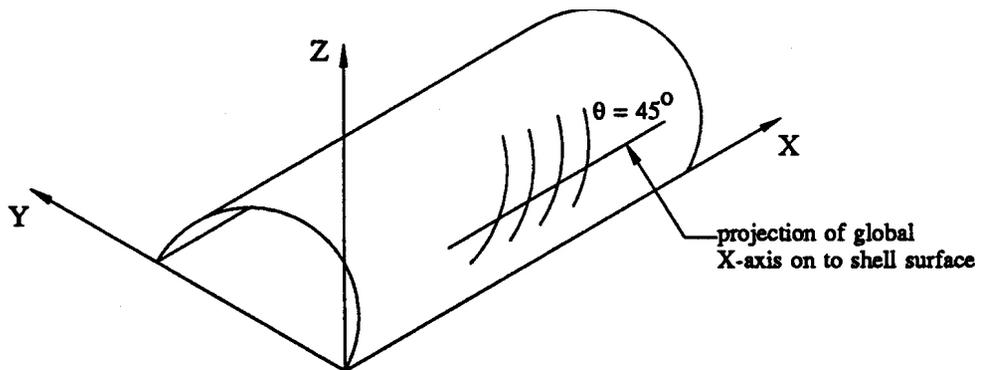


Figure 6.4: Composite cylindrical shell

It should be noted that IDRA = 0 is similar to IDRA = 4, 5 and 6, in as much as it describes a rotation about the normal to the shell; however, the reference line from which angles are measured is associated with each element instead of being defined globally. If the element has straight sides then the angle is defined as in Figure 6.5(b), i.e., the reference line connects the first two nodes of the element. If the element has curved sides (Figure 6.5(a)) then the reference line is the tangent, at node 1, to the line connecting nodes 1 and 2. These examples of using IDRA = 0 in Figure 6.5(a) and Figure 6.5(b) are for flat plates; the generalization to shells consists in realizing that the rotation angle is measured from the projection of the reference line defined above onto the tangent plane to the shell surface at each node.

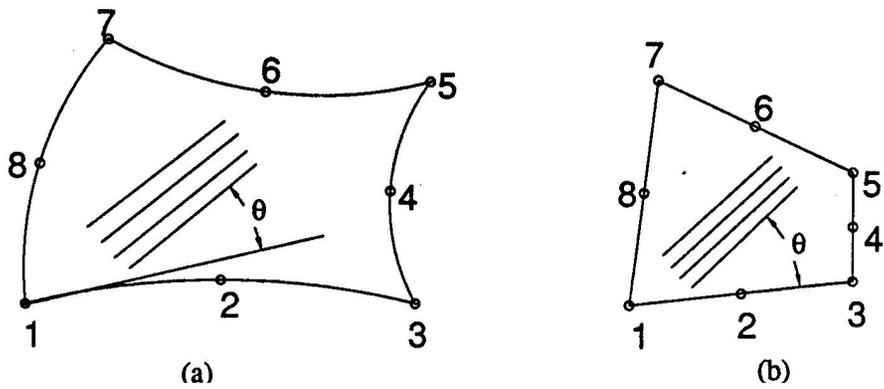


Figure 6.5: Rotation angles for IDRA = 0,

6.3.7 *BMDATA Data Group - Specification of Miscellaneous Data Required for General Beam Element (NKTP = 39)

This data group is required to describe the miscellaneous data used for the general prismatic beam element (NKTP = 39).

Group ID card:

*BMDATA

The miscellaneous beam data is described in three sets of cards, each is composed of one card. The first set defines beam local axes, end release, etc. The second set defines section identification and orientation. The third and last set defines the offset parameters of beam vertices.

Card set 1: One card, always required.

Entry No:	1	2	3	4	5	6	7
Variable:	IDBM	NODE3	NVECT	NCOD1	NCOD2	INTPR	LOFST
Max char:	6	6(8)	6	6	6(8)	1	1

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IDBM	beam data identification number (must be an integer between 1 and 999999). This is the element data identification number IDRC referenced on the element definition group (see *ELEMENTS data group, Section 6.3.2).
2	NODE3	optional third node to define element local coordinate system, see notes 1, 3, 8.
3	NVECT	optional vector ID to define element local coordinate system, see notes 2, 3, 8.
4	NCOD1	end release code for NODE1, see note 4.
5	NCOD2	end release code for NODE2, see note 4.
6	INTPR	integration points along the beam axis, see note 5.
7	LOFST	key to set beam offsets in preferred coordinate system = 0 beam offsets given in global coordinate system = 1 beam offset given in local coordinate system.(see note 8).

Model Data

Element Data

Card set 2: One card, always required to define section orientation.

Entry No:	1	2
Variable:	ISECT	THETA1
Max char:	6	10

Entry Variable Description

- | | | |
|---|--------|--|
| 1 | ISECT | beam cross-section identification number as specified in *BMSECT group (Section 6.3.8) |
| 2 | THETA1 | counterclockwise rotation angle (in degrees) of the section axes with respect to beam local yz axes (see note 7) |

Card set 3: One card to define offset vectors at NODE1 and NODE2 (see note 6)

Entry No:	1	2	3	4	5	6
Variable:	AX	AY	AZ	BX	BY	BZ
Max char:	10	10	10	10	10	10

Entry Variable Description

- | | | |
|---|----|---|
| 1 | AX | X-component of offset vector at first node |
| 2 | AY | Y-component of offset vector at first node |
| 3 | AZ | Z-component of offset vector at first node |
| 4 | BX | X-component of offset vector at second node |
| 5 | BY | Y-component of offset vector at second node |
| 6 | BZ | Z-component of offset vector at second node |

Notes:

1. The third node should be defined in the *NODES data group ([Section 6.4.2](#)) and is used to define the local xy plane for beam element (NKTP = 39), such that local x-axis is the line joining NODE1 and NODE2 and the third node lies in the beam xy plane. For elements with offsets defined in global coordinate system, beam xy plane is defined by the line joining beam vertices (beam ends after offset) as beam x axis and the third point. For beam offsets defined in local coordinate see note 8.

2. This number refers to a vector identification number IDVEC whose global Cartesian components are defined in *VECTORS data group (Section 6.7.2). This vector is used to define the local xy plane of beam element (NKTP = 39), such that the beam local x-axis is the line joining NODE1 and NODE2 or beam vertices (beam ends after offset for offset defined in global coordinate system) and the referenced vector lies in the local xy plane. For offset defined in local system see note 8.
3. The local xy plane of the beam element may be defined by one method only, either the third node specification (note 1), or the vector specification (note 2).
4. The end release codes are used to release up to five of the six degrees of freedom, defined in the local element's coordinate system, at either end of the element, NODE1 or NODE2. A released degree of freedom means that the connection between this DOF and corresponding the nodal point is removed (e.g., to simulate hinge connection). Any released degree of freedom must be associated with a non-zero stiffness. The release code is a six-digit number made up of 0's and 1's. The digit 0 represents a retained degree of freedom, and the digit 1 represents a released degree of freedom. The six digits in the release code correspond to, respectively, UX, UY, UZ, ROTX, ROTY, ROTZ in the element's coordinate system. A node with ROTX and ROTZ released will have the release code 000101, or 101. The release code 111111 is not allowed.

Singularity problems may arise if the assembled stiffness associated with a released DOF is not properly compensated for from neighboring elements. Therefore, the end release option should be used with caution. For example, if the same DOF is released at a common node of two colinear elements, the released DOF should be constrained to zero.

5. If the beam cross-section specified in the *BMSECT group is not a general cross section, a numerical integration using Gauss-Quadrature method will be performed along the beam axial direction. A default value of three points will be assigned if not specified here.
6. The offset vectors (v1 and v2, see Figure 4.34) allow the beam centroid to be offset from the mesh node points at either end. It is defined by the global X, Y and Z components of a vector pointing from the mesh node point to the beam centroid at corresponding beam vertex. In geometric nonlinear analysis, the relative position between the mesh node point to the beam centroid at corresponding beam vertex is maintained.
7. This angle (THETA1) represents the rotation or the twist of the section around the element x-axis as shown in Figure 6.6.

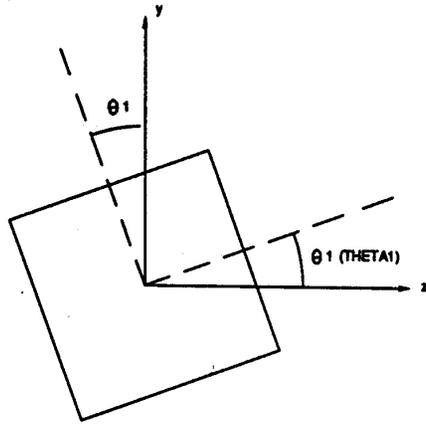


Figure 6.6: Rotation or twist angle for beam sections

- When beam offsets are defined in local system, the orientation of the beam local axes may be different from the way as described in notes 1 and 2. A local coordinate system has to be defined for beam offsets before the beam orientation can be defined. The third node or a vector is not used to define the beam orientation directly but is used to define the local system. First, a local xy plane is formed by using a third node (or a vector) and the axis from node 1 to node 2. Then the local system is formed by the local xy plane and the local z axis normal to the local xy plane. The values of offsets are specified in local y and z. The beam x axis is along the line joining the beam vertices (beam ends after offset). The beam xy plane is formed by a vector parallel to local y and the beam x axis. If the beam offsets are uniform, the beam orientation is parallel to local axis.

6.3.8 *BMSECT Data Group - Specification of Cross Section Data for General Beam Element (NKTP = 39)

This data group is required to describe the section parameters for various types of cross sections used for prismatic beam element (NKTP = 39). Numerical integration points for standard cross sections will also be specified in this data group.

Group ID card:

*BMSECT

The data for a typical beam cross-section consists of two sets of cards. The first set consists of only one card, which specifies the cross-section type and the required numerical integration points if necessary. The second set specifies the required parameters to describe the section geometry. The second set may consist of more than one card depending on the type of cross section selected in the first set.

Card set 1: One card, always required

Entry No:	1	2	3	4	5	6
Variable:	IDBLE	IDSEC	IFORM	INTP1	INTP2	INTP3
Max Char:	6	6	1	1	1	1

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
--------------	-----------------	--------------------

1	IDBLE	label for beam section type, see notes 3 and 4. Allowable labels are: <div style="margin-left: 20px;"> <p>GENERAL - Arbitrary cross-section (used only in linear static or geometric nonlinear analyses). See notes 1 and 3.</p> <p>RECT - Rectangular solid section</p> <p>CIR - Circular solid section</p> <p>PIPE - Circular thin-walled closed pipe section</p> <p>BOX - Rectangular thin-walled closed box section</p> <p>ISECT - Symmetric I-shaped open section</p> <p>LSECT - shaped open section</p> <p>TRAPZ - Trapezoidal solid section</p> </div>
---	-------	--

Model Data

Element Data

- 2 IDSEC section identification number (must be an integer between 1 and 999999). This is the cross-section identification number ISECT referenced on the beam data group, *BMDATA, see [Section 6.3.7](#).
- 3 IFORM format index for input of cards *in the second set*:
= 1 eight entries per card [Default]
= 2 four entries per card
- 4 INTP1 number of integration points in the first direction or branch, see note 2, and [Table 6.3](#).
- 5 INTP2 number of integration points in the second direction or branch, see note 2, and [Table 6.3](#).
- 6 INTP3 number of integration points in the third direction or branch, see note 2, and [Table 6.3](#).

Card set 2: Data to describe the section geometry. Use as many cards as needed (depending on the format index IFORM specified on card set 1) to enter all required parameters.

Entry No:	1	2	3	4	5	6	7	8
Variable:	VAL1	VAL2	VAL3	VAL4	VAL5	VAL6	VAL7	VAL8
Max Char:	10	10	10	10	10	10	10	10

Entry Variable Description

- 1-8 VAL1 to data required to describe the section geometry.
If IFORM = 1 on the previous card set, maximum of 8 entries per card with 10 characters each.
- VAL8 If IFORM = 2, maximum of 4 entries per card with 20 characters each. Continue on additional cards if more data is required to describe the section (details for data input are given in [Table 6.2](#) and [Table 6.3](#)).

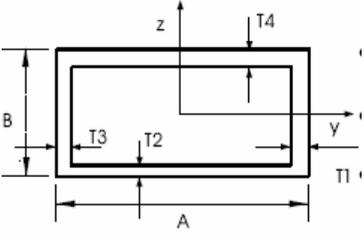
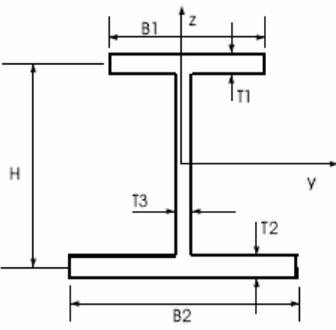
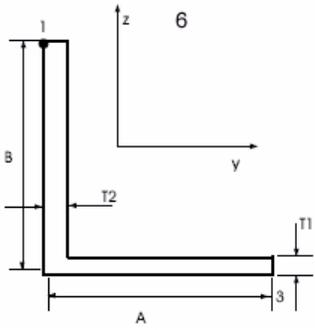
Table 6.2: Data input for different beam cross section

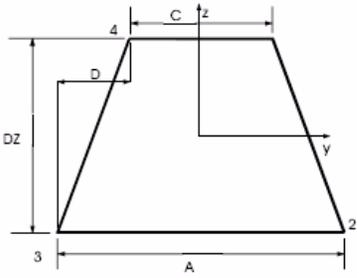
Type	Input Data Required
GENERAL RECT CIR	A, IY, IZ, IYZ, J, EY, EZ, DY, DZ. see note 1 A, B. see note 2 D
PIPE BOX ISECT	DI, DO A, B, T1, T2, T3, T4 B1, T1, B2, T2, H, T3
LSECT TRAPZ	A, B, T1, T2 A, C, DZ, D

Table 6.3: Geometrical Properties for Different Beam Cross Sections

General Cross Section	
A	= cross-sectional area
IY	= moment of inertia about Y-axis
IZ	= moment of inertia about Z-axis
IYZ	= product of inertia ($\int yz \, dA$)
J	= torsional constant
EY	= eccentricity in y-direction
EZ	= eccentricity in z-direction
DY	= depth in the y-direction (used for temperature gradient loading)
DZ	= depth in the z-direction (used for temperature gradient loading)

Standard Cross Section		
Section	Stress Output	Default Integration
Rectangular section		INTP1 = 5 in y direction INTP2 = 5 in z direction
Circular section		INTP1 = 3 in radial direction INTP2 = 8 in circumferential direction
Pipe section		INTP1 = 1 through pipe thickness INTP2 = 8 in circumferential direction

Standard Cross Section		
Section	Stress Output	Default Integration
Box section		<p>INTP1 = 5 in y direction INTP2 = 5 in z direction 1 through thickness</p>
Symmetric I-section		<p>INTP1 = 5 in top flange INTP2 = 5 along web INTP3 = 5 in bottom flange 1 through thickness</p>
L-section		<p>INTP1 = 5 in horizontal flange INTP2 = 5 in vertical flange 1 through thickness</p>

Standard Cross Section		
Section	Stress Output	Default Integration
Trapezoidal section		INTP1 = 5 in y direction INTP1 = 5 in z direction

ISECT	B1, T1, B2, T2, H, T3
LSECT	A, B, T1, T2
TRAPZ	A, C, DZ, D

Notes:

1. Only linear and geometric nonlinear analyses are available for the general or arbitrary section. For geometric nonlinear analysis, some approximation is activated in the formulation to achieve maximum efficiency. In the numerical tests, the exact and the approximate formulations show very good agreement. The user may still activate the exact formulation by using a standard section. Whenever applicable, i.e., in linear and geometrically nonlinear analyses, it is advised, however, to use the general section.
2. Section integration is performed for standard sections. INTP1, INTP2, INTP3 are the number of integration points along different directions or branches. Default integration points are used if not specified. See [Table 6.3](#) for default integration points and locations.
3. For general section, no stress output is available. The user may calculate the stresses by using the end forces.
4. The stress for standard section will be output at default points, the shear stress will be calculated in local coordinate with the combined effect of both torsional and bending shear.

6.3.9 *NLSPRING Data Group - Specification of Force-Deflection Data for Nonlinear Springs

This data group is required for springs in nonlinear analysis (NKTP = 17) to describe the force-deflection curve.

Group ID card:

*NLSPRING

Data for the force-deflection curves is defined in two sets of cards. The first set consists of one card that describes overall control parameters. The second set may consist of more than one card depending on the control parameters chosen in the first set.

Card set 1: One card, always required.

Entry No:	1	2	3	4	5	6
Variable:	IDSP	NCURV	IIFIT	IRFLCT	ICONS	IPOLTY
Max Char:	6	2	1	1	1	2

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IDSP	spring identification number (must be an integer between 1 and 999999). This is the identification number IDSP referenced in the real constant table for springs (see Table 4.39 , Section 4.17)
2	NCURV	= 1 if the force-deflection curve is temperature-independent = n if the force-deflection curves are given at n different temperatures ($1 \leq n \leq 10$)
3	IFIT	type of curve fit requested = 0 piece-wise linear curve fit [Default] = 1 cubic spline fit = 2 arbitrary polynomial fit (see note 3)
4	IRFLCT	flag to reflect positive force and deflection data points into negative ones (see note): = 0 no reflection, full range of data will be given [Default] = 1 reflect the data points into the negative displacement and force domain

Model Data

Element Data

- 5 ICONS flag to define type of unloading (see note 2):
 = 0 conservative unloading [Default]
 = 1 nonconservative unloading (see note 2)
- 6 IPOLTY flag to define spring loading type in polynomial fit; IFIT = 2, see note 5:
 = 0 tension and compression spring [default]
 = 1 tension only spring
 = -1 compression only spring

Card set 2: For each different temperature, a maximum of six cards may be required

Force-deflection data set

Entry No:	1	2	3	4	5	6	7	8
Variable:	TEMP	NPAIR	X1	Y1	X2	Y2	X3	Y3
Max Char:	10	5	10	10	10	10	10	10

Entry Variable Description

- | | | |
|-----|-------|---|
| 1 | TEMP | temperature at which the force-deflection data is given |
| 2 | NPAIR | number of force-deflection data pairs (should be ≤ 20) |
| 3 | X1 | deflection of the first pair, see note 4 |
| 4 | Y1 | force of the first pair, see note 4 |
| 5-6 | --- | |
| 7 | X3 | deflection of the third pair |
| 8 | Y3 | force of the third pair (additional pairs up to a maximum of 20 are given in subsequent cards with 4 pairs in one card and a maximum of 10 characters for each value). See note 3 for the definition (Xi, Yi) in the case of polynomial fit, i.e., IFIT = 2 |

Notes:

1. If the force-deflection data points are the same for negative deflections of the spring as those for positive deflections, then the user can input the data for only the positive deflections (first quadrant data) and obtain the negative data points by setting the reflection key, IRFLCT = 1 (see [Figure 6.7](#)).

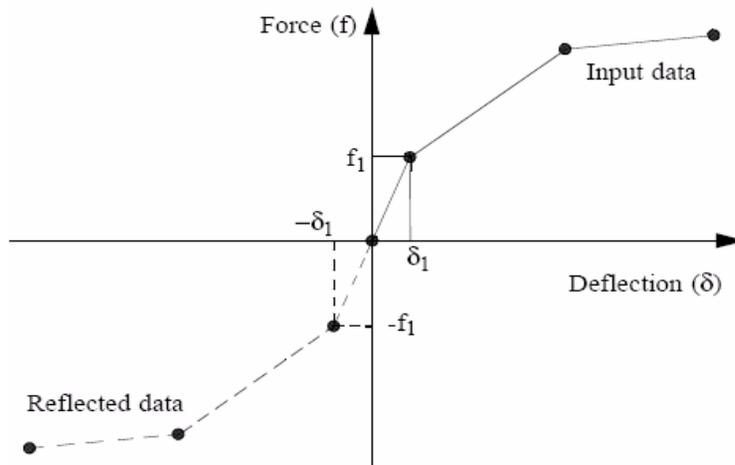


Figure 6.7: Reflected data points

2. For conservative unloading (see [Section 2.8.2](#) for details), the unloading path follows the same loading path as indicated in [Figure 6.8](#).

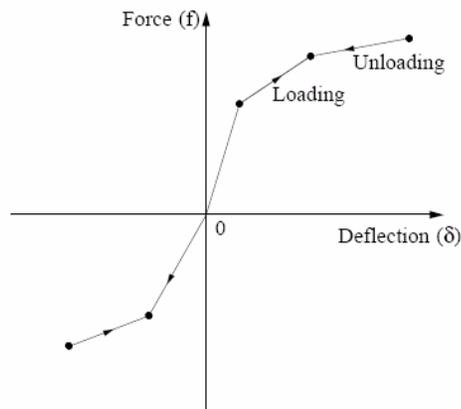


Figure 6.8: Conservative unloading

For non conservative loading, the unloading takes place along a line parallel to the tangent of the original slope at origin. Also, if during unloading, the load reversal crosses the force = 0 line, then an origin shift occurs and the force-deflection curve is shifted to this origin. See [Figure 6.9](#).

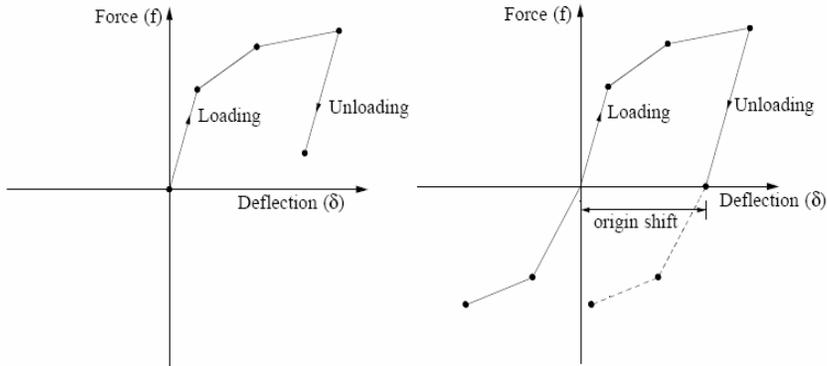


Figure 6.9: Nonconservative unloading

3. If IFIT = 2, a polynomial fit of the form:

$$F = X_1|\delta|^{Y_1} + X_2|\delta|^{Y_2} + \dots + X_n\delta^{Y_n}$$

is assumed, where $(X_1, Y_1), (X_2, Y_2), \dots, (X_n, Y_n)$ are the input data pairs (coefficients and exponents) and the origin should be at the $(0,0)$ point. Care must be taken to avoid infinite slope for any deflection. This means the value of Y_i in the input data should be equal to or greater than 1.0, since slope is computed by:

$$\frac{dF}{d\delta} = \sum_{i=1}^n X_i Y_i |\delta|^{Y_i-1}$$

4. The (X_i, Y_i) data points are to be input in an ascending order of X_i , including the negative values. The curve must pass through the origin $(0,0)$. The program always assumes the $(0,0)$ point if it is omitted by the user.
5. For polynomial fit, the spring may act as a tension-compression spring (IPOLTY = 0, default), or a tension only spring (IPOLTY = 1), or a compression only spring (IPOLTY = -1), as indicated in [Figure 6.10](#).

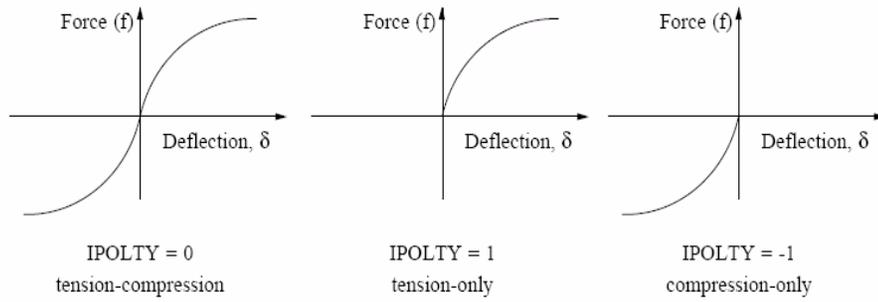


Figure 6.10: Various types of polynomial fit

6.3.10 *WELEMENTS Data Group - Weld Elements Definition

This data group is optional, required in linear static analysis whenever the model contains spot elements connecting the two surfaces. The spot weld element is one dimensional element and is represented by a beam element (NKTP12). The associatively of weld element with beam element is referred in this card.

Group ID card:

*WELEMENTS

Welements elements card set:

Entry No:	1	2		
Variable:	<table border="1"><tr><td>IDWELD</td><td>IDBEAM</td></tr></table>	IDWELD	IDBEAM	
IDWELD	IDBEAM			
Max Char:	8	8		

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IDWELD	Identification number for the weld element
2	IDBEAM	Identification number of a beam element (NKTP 12) that acts as weld element

Note:

For a spot weld, the associated beam element is referred in *WELEMENTS. This beam element ID is defined in *ELEMENTS data card has its RCTABLE ID and MATERIAL ID. The material ID so referred has GXY, GXZ, and GYZ data, the values of which are treated as TMAX (maximum tensile), CMAX (maximum compressive) and SMAX (maximum shear) stresses respectively in the solver. Similarly the RC table ID has circular cross sectional properties of beam element.

6.3.11 RCELEMENT

Defines the connectivity between Reinforcement bars and 3D concrete element

Applicable analysis type: NLSTATIC

It gives the connectivity between reinforcement bar defined in *REINFORCEMENT data group and 3D concrete element (NKTP=31).

Group ID card:

RCELEMENT

Entry no: 1 2 3 4 5 6 7 8 9 10

Variable:	NELI	IDREIN	IDFACE	X1	Y1	Z1	IDFACE	X2	Y2	Z2
	D	F	1				2			

Max Char: 8 8 1 12 12 12 1 12 12 12

<u>Entry</u> <u>No</u>	<u>Variable</u>	<u>Description</u>
1	NELID	-Element identification number
2	REINID	-Reinforcement bar identification number
3	IDFACE1	-Element face ID on which the reinforcement bar enters, see note 1.
4-6	X1, Y1, Z1	-Start point of the part of the reinforcement bar which lies inside NKTP=31, see note 2.
7	IDFACE2	-Element face ID through which the reinforcement bar exits, see note 1.
8-10	X2, Y2, Z2	-End point of the part of the reinforcement bar which lies inside NKTP=31, see note 2.

Notes:

1. A reinforcement bar can pass through any number of 3D concrete elements (NKTP=31). Reinforcement bar can be specified by up to five points, refer *REINFORCEMENT data card. The start and end points of the reinforcement bar lie either at any of the concrete element faces or within the element itself. The face ID (entries 4 and 8) of NKTP=31 is mentioned, if the point falls on the face of it. The face ID (entries 4 and 8) will be '-1' or '0' accordingly, if the start or end point lies within concrete element or coincides with concrete element node respectively.
2. Start and end points specified in this data card are different from *REINFORCEMENT. These points refer the part of the reinforcement bar which lie inside a concrete element, whereas the points in the *REINFORCEMENT data card defines the reinforcement bars.
3. If the concrete element contains more than one reinforcement, the data can be given as explained in the below example.

Example:

Assume that, two reinforcement bars (let 201 and 202 are IDs of the bars) lie inside a 3D concrete element (let the element ID is 101).

*RCELEM

101, 201, 1, 0.0, 0.25, 0.5, 6, 1.0, 0.25, 0.5

101, 202, 1, 0.0, 0.75, 0.5, 6, 1.0, 0.75, 0.5

6.4 Nodal Data

6.4.1 *LCSYSTEM Data Group - Definition of Local Coordinate Systems

Local coordinate systems may be specified in this group for the purpose of:

1. Defining a local coordinate system which may be referenced in the *NODES data group to specify nodal coordinates, and/or
2. Defining a local coordinate system which may be referenced in the *NODES data group as a local displacement coordinate system at any particular node.

Group ID card:

*LCSYSTEM

The free format data for a typical local coordinate system consists of one card set which may contain one or two cards depending on the method of definition.

Local system card set:

	\$			
Entry No:	1	2	3	4-12
Variable:	IDSYS	ITYPE	METHOD	Variable, depends on the value of 'METHOD'
Max Char:	6	1	1	max.10 characters each

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IDSYS	coordinate system ID, must be ≥ 3 . Systems 0, 1 and 2 are reserved for global cartesian, cylindrical and spherical systems, respectively.
2	ITYPE	type of local coordinate system = 0 - local system is cartesian = 1 - local system is cylindrical = 2 - local system is spherical
3	METHOD	coordinate system definition method, see Figure 6.11 = 0 - defined by 3 node numbers (These 3 nodes must be defined in the *NODES data group in the global coordinate system)

Model Data
Nodal Data

- = 1 - defined by the coordinates of 3 points(in local xy plane)
- = 2 - defined by the local system origin and 3 successive rotation angles
- = 3 - defined by the local system origin and two direction cosine vectors.
- = 4 - defined by the coordinates of 3 points (in local xz plane)

--- tab(\$) ---

4-12 -- definition parameters (see below for explanation). Entries 4 to 9 are entered on this card. Entries 10 to 12 are required only if METHOD = 1 or 3 and they are entered on a *continuation card* which must begin with a tab character (\$).

For METHOD = 0 (requires entries 4 - 6)

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
4	NODE1	a node number given in the *NODES group and defines the origin of the local system.
5	NODE2	a node number given in the *NODES group and defines a point on the +x axis of the local system.
6	NODE3	a node number given in the *NODES group and defines a point on the xy plane of the local system.

For METHOD = 1 (requires entries 4-9 on the first card, entries 10-12 on the second card)

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
4-6	XYZ1	global cartesian coordinates (X, Y, Z) of the origin of the local system.
7-9	XYZ2	global cartesian coordinates (X, Y, Z) of a point on the +x axis of the local system.
10-12	XYZ3	global cartesian coordinates (X, Y, Z) of a point on the xy plane of the local coordinate system. Entries 10-12 are entered on second card which must begin with a tab (\$).

For METHOD = 2 (requires entries 4-9)

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
4-6	XYZ0	global cartesian coordinates (X, Y, Z) of the origin of the local system.
7-9	THETAX, THETAY, THEATZ	three successive rotation angles (in degrees) about the global X, updated Y, and final Z axes, respectively, that define the orientation of the local system, see Figure 6.11(c) .

For METHOD = 3 (requires entries 4-9 on the first card, entries 10-12 on the second card)

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
4-6	XYZ0	global cartesian coordinates (X, Y, Z) of the origin of the local system.
7-9	DIRCOS1	direction cosines of a vector lying on the +x axis of the local system (3 entries).
10 - 12	DIRCOS2	direction cosines of a vector lying on the xy plane of the local system, other than the local x-axis itself (3 entries). Entries 10-12 are input on second card which must begin with a tab (\$).

For METHOD = 4 (requires entries 4 - 6)

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
4	NODE1	a node number given in the *NODES group and defines the origin of the local system.
5	NODE2	a node number given in the *NODES group and defines a point on the +z axis of the local system.
6	NODE3	a node number given in the *NODES group and defines a point on the xz plane of the local system.

Notes:

1. Refer to [Section 6.4.2](#) for an example of defining a local coordinate system.
2. If the local coordinate system is cylindrical (ITYPE = 1 on first card) and is referenced in the *NODES data group to define nodal coordinates, then the corresponding nodal coordinates must be cylindrical (R, θ , Z). The same holds for a local spherical coordinate system.
3. A local cylindrical (or spherical) coordinate system may be referenced in the *NODES data group to define both coordinates and local displacement coordinate systems at more than one node. The local displacement coordinate system directions will be dependent on the coordinates of each node.

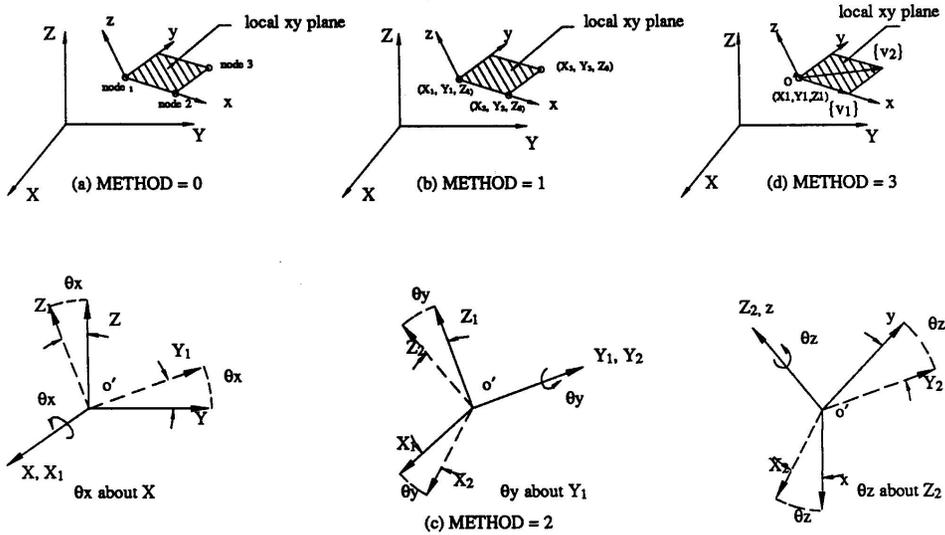


Figure 6.11: Definition of local coordinate systems, $(X, Y, Z) =$ global coordinate system, $(x, y, z) =$ local coordinate system, $o' =$ shifted origin: (a) using three nodes, (b) using coordinates of three points, (c) using shifted origin and successive three rotations, (d) using shifted origin and two vector $\{v1\}$ and $\{v2\}$.

6.4.2 *NODES Data Group - Nodal Coordinates

This data group is always required. It defines the nodal coordinates and may also be used to specify local displacement coordinate systems at nodes.

Group ID card:

*NODES

Nodal coordinates card set:

The nodal coordinates may be defined using *any* combination of three methods: individual definition, first level generation, and second level generation. Second level generation is activated whenever a base set, defined using individual nodes definition and/or first level generation, is preceded by the second level generation card as shown below. Examples for first and second level node generation are also given below.

	1	2	3	4	5	6	7	8
Entry No:								
Variable:	NODE	IDCSYS	INDEX	NSET	X	Y	Z	IDDSYS
Max Char:	6(8)	6	6	2	12	12	12	6

Entry Variable Description

- | | | |
|---|--------|---|
| 1 | NODE | <p>for individual definition and first level generation, this is the node number (positive integer up to 6 digits).</p> <p>for second level generation, a negative value of NODE indicates that this card is a second level generation card. The absolute value of NODE is the extrapolation increment for the node numbers defined in the base set. The nodes in the base set are defined immediately following this card.</p> |
| 2 | IDCSYS | <p>geometry coordinate system ID (in which the coordinates of the current node is defined)</p> <p>= 0 - global cartesian</p> <p>= 1 - global cylindrical</p> <p>= 2 - global spherical</p> <p>≥ 3 - local coordinate system, as defined in *LCSYSTEM data group</p> |
| 3 | INDEX | <p>- not used for individual node definition, enter zero</p> |

Model Data

Nodal Data

- for first level generation, this is the node number increment. If a card with INDEX > 0 follows an individually defined node, then first level generation will create nodes in increments of INDEX between the two NODEs and assign coordinates to these nodes, interpolating the coordinates data for the two NODEs.
 - for second level generation, this is the number of nodes in the base set (maximum of 100). Nodes in the base set are defined immediately following this card.
- 4 NSET - not used for individual definition and first level generation, enter zero.
- for second level generation, this is the number of sets to be generated from the base set, including the base set (maximum of 99).
- tab (\$)--
- 5-7 X, Y, Z nodal coordinates referred to the selected coordinate system (IDCSYS).
(For second level generation, these entries are the extrapolation increments for the coordinates).
- global cartesian coordinates (X, Y, Z) if IDCSYS = 0
 - global cylindrical coordinates (R, θ , Z) if IDCSYS = 1
 - global spherical coordinates (R, θ , ϕ) if IDCSYS = 2
 - consistent with the local coordinate system type if IDCSYS \geq 3
- tab (\$) ---
- 8 IDDSYS local displacement coordinate system ID.
- = 0 - same as global cartesian coordinate system.
 - = 1 - local displacement cylindrical coordinate system, dependent on the location of the node point with respect to the global axis, will be generated internally.
 - = 2 - local displacement spherical coordinate system, dependent on the location of the node point with respect to the global axis, will be generated internally.
 - \geq 3 - local coordinate system as defined in the *LCSYSTEM data group. The local displacement coordinate system will be dependent on the location of the node point if the referenced system is cylindrical or spherical.

Notes:

1. For second level generation, the nodes in the base set may be defined individually or by using first level generation. They may not be defined using second level generation.
2. All nodes defined using first or second level generation must have the same local displacement coordinate system ID (IDDSYS).
3. The *G2 data group provides an alternate form for defining local displacement coordinate system at nodes, and it *supersedes* any local system definition specified in this data group, if the definition corresponds to the same node.
4. Once a local displacement coordinate system is specified at a node, any subsequent reference to any degree of freedom at that node *must* be referred to that local displacement coordinate system. This includes data for coupled displacement, rigid elements, multi-point constraint equations, input loads and specified displacements.

Example 6.3.1: First level node generation

The six nodes shown in [Figure 6.12](#) are generated using first level generation. Nodes 1 and 6 are defined in the global cartesian coordinate system. The coordinates for the generated nodes (2, 3, 4 and 5) are linearly interpolated between nodes 1 and 6 in the same coordinate system.

```
**  
**Individual definition for node number 1  
1, 0, 0, 0, 0.0, 0.0, 0.0  
**  
**Definition of node number 6. First level generation is activated between nodes 1 and 6 with node  
**increment of 1  
6, 0, 1, 0, 5.0, 0.0, 0.0  
**
```

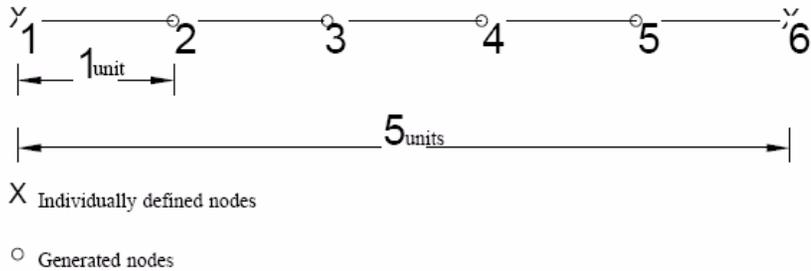


Figure 6.12: First level node generation

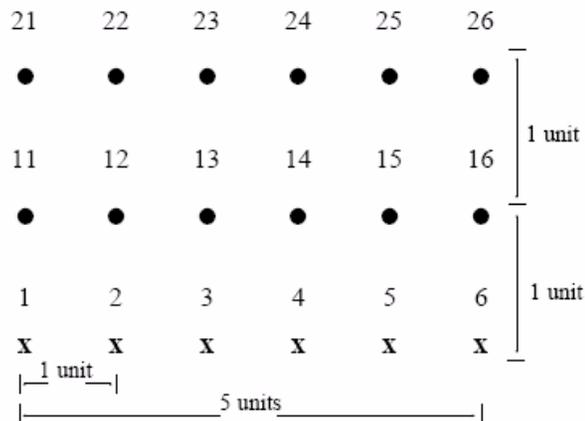
Example 6.3.2: Second level node generation

The 18 nodes in Figure 6.13 are generated using second level generation. The base set are generated via first level generation (individual definition may also be used). Three sets including the base set are generated.

```

**
**Define the second level generation parameters as well as the extrapolation increments
for nodal
**coordinates
-10, 0, 6, 3, 0.0, 1.0, 0.0
**
**Define nodes of the base set (nodes 1 to 6). First level generation is used
1, 0, 0, 0, 0.0, 0.0, 0.0
6, 0, 1, 0, 5.0, 0.0, 0.0
**

```



X Nodes in the master set

● Generated nodes

Figure 6.13: Second level node generation in the master set

Example 6.3.3: Use of local coordinate systems

Consider the geometry shown in Figure 6.14 where it is required to define the coordinates of nodes 1 through 17. It is convenient to define nodes 1-6 in the global cartesian coordinate system, nodes 7-10 in a local cartesian coordinate system (IDCSYS = 3) and nodes 13-17 in a local cylindrical coordinate system (IDCSYS = 4), as shown in the figure. In addition, it is required that nodes 13-17 have a local displacement coordinate system, consistent with the local cylindrical system in which the coordinates are defined. The input data is as follows:

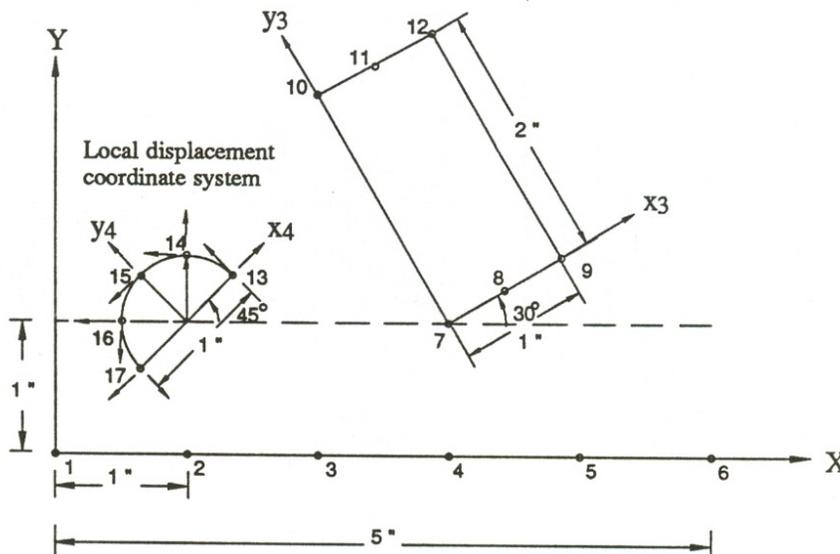


Figure 6.14: Geometry and node locations for example 6.3.3

```

**
*LCSYSTEM
**IDSYS      ITYPE      METHOD      —origin—      —angles—
3,           0,         2,         3.0, 1.0, 0.0,  0.0, 0.0, 30.0
4,           1,         2,         1.0, 1.0, 0.0,  0.0, 0.0, 45.0
**
*NODES
**NODE      IDCSYS      INDEX      NSET          X
**nodes 1-6 defined in global cartesian, using first level generation
1,          0,          0,          0,            0.0,
6,          0,          1,          0,            5.0,

```

** nodes 7-10 defined in local system no.3, using 1st and 2nd level generation

-3,	3,	3,	2,	0.0,
7,	3,	0,	0,	0.0,
9,	3,	1,	0,	1.0,

**nodes 13-17 defined in local system no.4, using first level generation,

**a local displacement coordinate system, IDDSYS = 4 is specified

13,	4,	0,	0,	0.5,
17,	4,	1,	0,	0.5,

**

6.4.3 *G2 Data Group - Alternate Form for Defining Local Displacement Coordinate Systems at Nodes

This data group may be used to define local displacement coordinate systems at nodes using successive rotation angles. Users are advised to use the standard form of definition, available in the *LCSYSTEM and *NODES data groups, since this data group (*G2) will not be supported in future versions.

Group ID card:

*G2

Nodal local displacement coordinate system card set:

Local displacement coordinate systems may be defined by using *any* combination of three methods: individual definition, first level generation, and second level generation. Second level generation is activated whenever a base set, defined using individual definition and/or first level generation, is preceded by the second level generation card as shown below.

	\$					
Entry No:	1	2	3	4	5	6
Variable:	NODE	INDEX	NSET	THETAX	THETAY	THETAZ
Max Char:	6	6	2	12	12	12

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NODE	<ul style="list-style-type: none"> - for individual definition and first level generation, this is the node number. - for second level generation, a negative value of NODE indicates that this card is a second level generation card. The absolute value of NODE is the extrapolation increment for the node numbers defined in the base set. The rotation angles for the nodes in the base set are defined immediately following this card.
2	INDEX	<ul style="list-style-type: none"> - not used for individual definition, enter zero.

-
- for first level generation, this is the node number increment. When a card with INDEX > 0 follows a card defining rotation angles at a single node, then the first level generation will select intermediate nodes in increments of INDEX between the two NODEs and assign angles to these intermediate nodes, interpolating the rotation angle data for the two NODEs.
 - for second level generation, this is the number of nodes in the base set (maximum of 100).
- 3 NSET
- not used for individual definition and first level generation, enter zero.
 - for second level generation, this is the number of sets to be generated from the base set, including the base set (maximum of 99).
- tab(\$) ---
- 4 - 6 THETAX, THETAY, THETAZ
- for individual definition and first level generation, these are three successive rotation angles (in degrees) about the global X, updated Y and final Z axes, respectively, that define the orientation of the local system, see [Figure 6.11\(c\)](#).
 - for second level generation, these are the extrapolation increments for the rotation angles defined for the nodes in the base set.

Notes:

1. In the second level generation method, nodes in the base set may be defined individually or by using first level generation. They may not be defined using second level generation.
2. A local displacement coordinate system defined at a node in the *G2 data group overrides any local displacement coordinate system assigned to the node in the *NODES data group.
3. If a local displacement coordinate system is defined at a node, then the input loads, specified displacements, and kinematic constraints (*CPDISP, *RIGLINK and *MPCEQN data groups) at this node will be in this local coordinate system.

6.5 Material Data

6.5.1 *MATERIAL Data Group - Material Property Data

This group is always required *unless*:

1. The analysis being performed is a steady state or transient heat transfer analysis. Material properties for heat transfer analyses are entered in the *MATHEAT data group.
2. The model consists entirely of elements which do not require material properties, e.g., springs and point masses - see element library.

For nonlinear material input used in nonlinear analysis, refer to the *PLASTIC, *APLASTIC and *HYPEREL data groups.

Group ID card: *MATERIAL

Material properties card set:

	\$							
Entry No:	1	2	3	4	5	6	7	8
Variable:	LABEL	MATID	KTEMP	COEF0	COEF1	COEF2	COEF3	COEF4
Max Char:	4	6	1	12	12	12	12	12

Entry Variable Description

- | | | |
|---|-------|---|
| 1 | LABEL | <p>label for the material property. For orthotropic material, the labels refer to the material axes. Allowable labels are:</p> <p>EX, EY, EZ - elastic moduli (force/area) in the direction of X, Y and Z axes, respectively.</p> <p>NUXY, NUXZ, NUYZ - Poisson's ratios such that NU_{ij} characterizes strain in the j direction produced by stress in the i direction (i, j = X, Y, Z).</p> |
|---|-------|---|

GXY, GXZ, GYZ - shear moduli (force/area) in the XY, XZ and YZ planes, respectively.
 - GXY, GXZ, and GYZ are treated as TMAX (maximum tensile), CMAX (maximum compressive) and SMAX (maximum shear) stresses respectively for weld elements.

DENS - mass density (force · time²/length⁴)

ALPX, ALPY, ALPZ - thermal expansion coefficients (strain/°temperature) in the direction of X, Y, and Z axes, respectively.

The following strength properties may be entered for composite elements (NKTP = 7, 32, 33 - refer to element library for definition):

FXC, FXT, FYC, FYT, FZC, FZT, FSXY, FSXZ, FSYZ, F12, F23, F13

2 MATID material identification number (must be an integer between 1 and 999999). This is the material ID referenced on the element definition group (*ELEMENTS).

3 KTEMP temperature dependence key
 = 0 - material property is constant
 = 1 - property is a polynomial function of temperature (see note 2)
 > 1 - for nonlinear analysis only, property is temperature dependent.
 KTEMP is the temperature interval at which property values are given, starting at temperature zero (see note 9)

--tab(\$)--

4 COEF0 base (constant) value of the material property

--- the following entries are used only if KTEMP > 0 ---

5 COEF1 coefficient of linear term of polynomial expression for variation of property with temperature.

6 COEF2 coefficient of quadratic term

7 COEF3 coefficient of cubic term

8 COEF4 coefficient of quartic term

Notes:

1. Not all the material properties listed in this group are required for all element types. Refer to the element library for the properties required for each element type.
2. COEF1, COEF2, COEF3 and COEF4 will not be used if KTEMP is set to zero. If KTEMP is set to 1, then the material property designated by IDLBL will be a polynomial function of temperature given by:

$$P(T) = \alpha_0 + \sum_{i=1}^4 \alpha_i T^i$$

where P is a typical property, T is the temperature and α_i ($i = 0, 1, \dots, 4$) are the polynomial coefficients (COEF0 through COEF4).

3. For isotropic materials (KISO = 0 in the *ELEMENTS data group), all elastic properties will be inferred from EX and NUXY, and other properties need not be entered. In this case, NISA will set

$$EX = EY = EZ$$

$$NUXY = NUXZ = NUYZ$$

$$ALPX = ALPY = ALPZ$$

4. For orthotropic materials (KISO = 1 in the *ELEMENTS data group) a full set of material properties consistent with the element type must be entered. Refer to the NISA element library for the properties required for each element type.
5. For orthotropic materials, the definition of Poisson's ratio is important. The convention used in NISA is defined by the reciprocity relationship:
$$NUXY * EY = NUYX * EX$$
6. In STATIC analysis, the mass density (DENS) is required only if a body force (*BODYFORCE data group) is applied. The mass density is expressed in units of (force. time²/length⁴). Weight density can be converted to mass density by dividing by the acceleration of gravity.
7. NISA is entirely independent of any system of units. The only requirement is that the units used must be self consistent.
8. For eigen value and buckling analyses the variable KTEMP should be set to zero.
9. For KTEMP > 1, COEF1 to COEF5 are the values of the property at T0, T0 + KTEMP, T0 + 2*KTEMP, etc., where T0 = 0.0. Not all five values are required but the last value given should be greater than or equal to the highest temperature at any point in the model. This option is only available in nonlinear analysis.

6.5.2 *MATDIR1 Data Group - Definition of Orthotropic Material Axes with Respect to Nodes

This data group may be used to define orthotropic material axes with respect to nodes. Alternatively, the material axes may be defined with respect to elements in the *MATDIR2 data group. Note that both the *MATDIR1 and *MATDIR2 *cannot* coexist in the same data deck. *Skip* this data group entirely if the orthotropic material directions coincide with the global directions everywhere. Orthotropic material directions at nodes which are not referenced in this data group are assumed to be coincident with the global directions.

Group ID card:

*MATDIR1

Material axes card set:

The orthotropic material axes may be defined using any combination of three methods: individual definition, first level generation and second level generation. Second level generation is activated whenever a base set, defined using individual definition and/or first level generation, is preceded by the second level generation card as shown below.

	\$					
Entry No:	1	2	3	4	5	6
Variable:	NODE	INDEX	NSET	THETAX	THETAY	THETAZ
Max Char:	6(8)	6	2	12	12	12

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NODE	<ul style="list-style-type: none"> - for individual definition and first level generation, this is the node number at which the material axes are defined on this card. - for second level generation, a negative value of NODE indicates that this card is a second level generation card. The absolute value of NODE is the extrapolation increment for the node numbers defined in the base set. Material axes at nodes in the base set are defined immediately following this card.
2	INDEX	<ul style="list-style-type: none"> - not used for individual definition, enter zero.

Model Data

Material Data

- for first level generation, this is the node number increment. When a card with INDEX > 0 follows a card defining material axes at a single node, then the first level generation will select intermediate nodes in increments of INDEX between the two NODEs and assign material axes to these intermediate nodes, interpolating the material axes data for the two NODEs.
 - for second level generation, this is the number of nodes in the base set (maximum of 100). Material axes at nodes in the base set are defined immediately following this card.
- 3 NSET
- not used for individual definition and first level generation, enter zero.
 - for second level generation, this is the number of sets to be generated from the base set, including the base set (maximum of 99).
- tab(\$) ---
- 4-6 THETAX, - for individual definition and first level generation, these are three successive rotation angles (in degrees) about the global X, updated Y and THETAY, final Z axes, respectively, that define the orientation of the local system, see [Figure 6.11\(c\)](#).
THETAZ
- for second level generation, these are the extrapolation increments for the rotation angles defining the material axes at a typical node in the base set.

Notes:

1. For second level generation, nodes in the base set may be defined individually or by using first level generation. They may not be defined using second level generation.
2. Orthotropic material axes at nodes which are not referenced in this group are assumed to be coincident with global axes.
3. Material properties entered in the *MATERIAL or *MATHEAT data groups are referred to the material axes defined in this data group.

6.5.3 *MATDIR2 Data Group - Definition of Orthotropic Material Axes with Respect to Elements

This data group may be used to define orthotropic material axes with respect to elements, wherein the material directions do not vary within a typical element. Alternatively, the material axes may be defined with respect to nodes in the *MATDIR1 data group. Note that both *MATDIR1 and *MATDIR2 *cannot* coexist in the same data deck. Skip this data group entirely if the orthotropic material directions coincide with the global directions everywhere. Orthotropic material directions at elements which are not referenced in this data group are assumed to be coincident with the global directions.

Group ID card: ***MATDIR2**

Material axes card set:

The orthotropic material axes may be defined using any combination of three methods: individual definition, first level generation and second level generation. Second level generation is activated whenever a base set, defined using individual definition and/or first level generation, is preceded by the second level generation card as shown below.

	\$						
Entry No:	1	2	3	4	5	6	7
Variable:	NELID	INDEX	NSET	THETAX	THETAY	THETAZ	ISXY
Max Char:	6(8)	6	2	12	12	12	1

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NELID	<ul style="list-style-type: none"> - for individual definition and first level generation, this is the node number at which the material axes are defined on this card. - for second level generation, a negative value of NELID indicates that this card is a second level generation card. The absolute value of NELID is the extrapolation increment for element numbers of the base set. Material axes at elements in the base set are defined immediately following this card.
2	INDEX	<ul style="list-style-type: none"> - not used for individual definition, enter zero.

Model Data

Material Data

- for first level generation, this is the element number increment. When a card with INDEX > 0 follows a card defining material axes at a single element, then the first level generation will select intermediate elements in increments of INDEX between the two NELIDs and assign material axes to these intermediate elements, interpolating the material axes data for the two NELIDs.
 - for second level generation, this is the number of elements in the base set (maximum of 100). Material axes at elements in the base set are defined immediately following this card.
- 3 NSET
- not used for individual definition and first level generation, enter zero.
 - for second level generation, this is the number of sets to be generated from the base set, including the base set (maximum of 99).
- tab(\$) ---
- 4-6 THETAX, - for individual definition and first level generation, these are three
 THETAY, successive rotation angles (in degrees) about the global X, updated
 THETAZ Y and final Z axes, respectively, that define the orientation of the
 local system, see [Figure 6.11\(c\)](#).
- for second level generation, these are the extrapolation increments for the rotation angles defining the material axes at a typical node in the base set.
- 7 ISXY
- rotation angles reference key
- = 0 - rotation angles referenced to global X, Y, Z axes
- = 1 - valid only for 3-D solid element (NKTP = 4). Rotation angles are referenced to local element axes, see note 4.

Notes:

1. For second level generation, material axes at elements in the base set may be defined individually or by using first level generation. They may not be defined using second level generation.
2. Orthotropic material axes at elements which are not referenced in this group are assumed to be coincident with global axes.
3. Material properties entered in the *MATERIAL or *MATHEAT data groups are referred to the material axes defined in this data group.

4. For the 3-D solid element (NKTP = 4), the rotation angles are referenced to the local element axes if the rotation angle reference key (ISXY) is set to 1. The local element axes are defined at the first node of the element such that the x axis is tangent to the line connecting nodes 1 and 2 of the element and the z axis is normal to the tangent plane of the bottom face of the element (face 1) at node 1 and is pointing in to the element. The y axis is orthogonal to both x and z axes.

6.5.4 *MATHEAT Data Group - Material Properties Data for Heat Transfer Analyses

This data group is always required in heat transfer analyses to specify the material properties. Group ID card:

Group ID card:

*MATHEAT

Material properties card set:

	\$							
Entry No:	1	2	3	4	5	6	7	8
Variable:	LABEL	MATID	KTEMP	IDCURV	COEF0	COEF1	COEF2	COEF3
Max Char:	4	6	1	6	12	12	12	12

Entry No:	9	10
Variable:	COEF4	IPCHNG
Max Char:	12	2

Entry Variable Description

- | | | |
|---|-------|---|
| 1 | LABEL | label for the material property. For orthotropic material, the labels refer to the material axes. Allowable labels are:
KXX, KYY, KZZ - thermal conductivities in the direction of X, Y and Z axes, respectively, heat/(length. time. degree)
C - specific heat, heat/(weight. degree) or heat/(mass. degree), depending on units of density
DENS - density, (weight/volume or mass/volume).consistent with units of specific heat |
| 2 | MATID | material identification number (must be an integer between 1 and 999999). This is the material ID referenced on the element definition group (*ELEMENTS). |
| 3 | KTEMP | temperature dependence key (see notes 2, 3)
= 0 - material property is constant or property vs. temperature curve is used.
= 1 - property is a polynomial function of temperature |

- 4 IDCURV identification number for property vs. temperature curve for a temperature dependent material. This is an integer number (up to 6 digits) which refers to a curve ID number defined in the *TEMPFN data group, see note 3.
- tab(\$) ---
- 5 COEF0 base (constant) value of the material property
 - - - - the following entries are used only if KTEMP > 1 - - -
- 6 COEF1 coefficient of linear term of polynomial expression for variation of property with temperature.
- 7 COEF2 coefficient of quadratic term
- 8 COEF3 coefficient of cubic term
- 9 COEF4 coefficient of quartic term
- 10 IPCHNG phase change identification key (see note 8)
 = 0 - material is not subjected to phase change
 = 1 - material is subjected to phase change

Notes:

1. Not all the material properties listed in this group are required for all element types. Refer to the heat transfer element library for the properties required for each element type.
2. COEF1, COEF2, COEF3 and COEF4 will not be used if KTEMP is set to zero. If KTEMP is set to 1, then the material property designated by IDLBL will be a polynomial function of temperature given by:

$$P(T) = \alpha_0 + \sum_{i=1}^4 \sigma_i T^i$$

where P is a typical property, T is the temperature and α_i (i = 0, 1,., 4) are the polynomial coefficients (COEF0 through COEF4).

3. If KTEMP = 1, then IDCURV will be ignored. IDCURV will be activated only when KTEMP = 0. If KTEMP = 0 and IDCURV = 0, the material property is not temperature dependent.
4. For isotropic materials (KISO = 0 in the *ELEMENTS data group), all three conductivities will be set to the value entered for KXX.

Model Data

Material Data

5. For orthotropic materials (KISO = 1 in the *ELEMENTS data group) a full set of material properties consistent with the element type must be entered. Refer to the NISA heat transfer element library for the properties required for each element type.
6. The specific heat (C) and mass density (DENS) do not enter into the equations of steady-state heat transfer. Hence, they are not required in SHEAT analysis. However, they are required for transient heat transfer (THEAT) analysis.
7. NISA is entirely independent of any system of units. The only requirement is that the units used must be self consistent.
8. If IPCHNG = 1, then material is assumed to be subjected to phase change, in which case enthalpy vs. temperature curve must be provided for the material in *PCHANGE1 data group.

6.5.5 *HYPEREL Data Group - Hyperelastic and Rubber-like Material Properties

This data group is required in nonlinear analysis to define hyperelastic material properties.

Various forms of hyperelastic constitutive relations (forms of the strain energy functions) have been discussed in [Chapter 2](#). Each hyperelastic material will have a unique material identification number, MATID. At least one property value should be given in the *MATERIAL group for the same MATID. User may specify density or Poisson's ratio under the *MATERIAL group. If Poisson's ratio is specified, it will be overwritten by the default value (0.499) or by the value for NU given in this group.

Group ID card:

*HYPEREL

The data for a typical hyperelastic material consists of two sets of cards. The first set consists of one card that describes the overall control parameters. The second set may consist of more than one card depending on the parameters selected in the first set.

Card set 1: One card, always required

Entry No:	1	2	3	4	5
Variable:	LABEL	MATID	NPOTN	NORDR1	NORDR2
Max Char:	4	6	2	1	1

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	LABEL	label for general parameters of hyperelastic materials = HYPE
2	MATID	material ID number. This is the same material ID referenced on the element definition group *ELEMENTS. At least one elastic property value should be given under the *MATERIAL group for each MATID.
3	NPOTN	selection of the potential or strain-energy function; see Chapter 2 for details: = 1 - Neo-Hookean = 2 - Classical Mooney-Rivlin = 3 - Generalized Mooney-Rivlin = 4 - Swanson

Model Data

Material Data

- = 5 - Blatz-Ko
- = 6 - IHT
- = 7 - Biderman
- = 8 - Klosner-Segal
- = 9 - Hart-Smith (exponential)
- = 10 - Alexander (exponential)

4-5 NORDR1, polynomial orders in the strain-energy functions. Only applicable to Generalized Mooney-Rivlin (NPOTN = 3) and Swanson (NPOTN = 4). NORDR1 gives the order of the first term in the polynomial and NORDR2 gives the order of the second term in the polynomial. The following restrictions apply to NORDR1 and NORDR2:

<u>CASE</u>	<u>NORDR1 or NORDR2</u>
Plane stress problems	≤ 3
Generalized Mooney-Rivlin	≤ 8
Swanson	≤ 4

Card set 2: May be more than one card, always required.

Each card begins with a label to identify the coefficient being defined. The coefficients required for each potential or strain energy function, NPOTN, are listed in [Table 6.4](#) (see notes 1-3). The *order* of the coefficients given in the table *must* be preserved when entering data. Definition of the potential or strain energy function and the coefficients are explained in [Chapter 2](#).

Table 6.4: Required coefficients for various hyperelastic material models

Material Type	NPOTN	Required Coefficient(S)
Neo-Hookean	1	CI11, NU
Classical Mooney-Rivlin	2	CI11, CI21, NU
Generalized Mooney-Rivlin	3	CI11, CI12, CI13,... CI18, CI21, CI22, CI23,...CI28, NU
Swanson	4	CI11, CI12,... CI14, ALP1, ALP2,... ALP4,CI21, CI22,... CI24, BET1, BET2,... BET4, D1
Blatz-Ko	5	MU, BETA
IHT	6	CI11, CI12, CI21, NU
Biderman	7	CI11, CI12, CI13, CI21, NU
Klosner-Segal	8	CI11, CI21, CI22, CI23, NU
Hart-Smith	9	CI12, CI21, G1, NU
Alexander	10	CI12, CI21, CI22, G1, G2, NU

\$

Entry No:	1	2	3	4	5	6	7	8
Variable:	LABEL	MATID	TEMPID	VAL1	VAL2	VAL3	VAL4	VAL5
Max Char:	4	6	10	10	10	10	10	10

Model Data

Material Data

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	LABEL	label identification, e.g. CI11, CI21, G1, BET1... The available list of labels is given in Table 6.4 . The <i>order</i> given in the table <i>must be preserved</i> when entering the coefficients.
2	MATID	material ID number. This is the same material ID number referenced on the element definition data group, *ELEMENTS. At least one elastic property value should be given under the *MATERIAL group for each MATID.
3	TEMPID	temperature dependence key = 0 - material coefficient is temperature independent. = 1 - material coefficient is a polynomial function of temperature > 1 - temperature intervals at which coefficients are given
--- tab(\$) ---		
4-8	VAL1 to VAL5	value of the material property designated by the IDLBL label. These entries have the following interpretation depending on the temperature dependence key TEMPID: <u>if TEMPID = 0:</u> VAL1 = value of the property. VAL2 to VAL5 are not required. <u>if TEMPID = 1:</u> VAL1 to VAL5 are the coefficients of the fourth order polynomial expression for variation of the property with temperature, i.e., constant term, linear term, quadratic term, etc. <u>if TEMPID > 1:</u> VAL1 to VAL5 are the values of the property at T0, T0+TEMPID, T0+2*TEMPID,etc., where T0 = 0.0. Not all the 5 values may be required, see note 4.

Notes:

1. In [Table 6.4](#), it should be noted that each coefficient is identified by a letter C followed by I1 or I2 followed by an integer indicating the order of the particular term in the polynomial. As an example, CI14 indicates the coefficient of the fourth order I1 term in the strain energy expression, and so on.
2. The material types of Neo-Hookean, classical Mooney-Rivlin, IHT, Biderman and Klosner-Segal maybe obtained as special cases of the generalized Mooney-Rivlin material. This is achieved by specifying appropriate values for NORDR1 and NORDR2 of the generalized polynomial.
3. In [Table 6.4](#), the Poisson's ratio NU is optional and if not specified, the near incompressibility case will be assumed, i.e. $NU = 0.499$. The value of NU may be controlled to give finite compressibility behavior.
4. If $TEMPID > 1$, not all the values of VAL1 to VAL5 have to be given. However, the last value given should be greater than or equal to the highest temperature at any point in the model.

6.5.6 *PLASTIC Data Group - Plasticity Properties for Elastoplastic Material Models

This data group is required in nonlinear analysis to describe the plasticity part of elastoplastic material models, *if any*. All other mechanical properties for the same material (elastic properties, mass density and thermal expansion coefficients) *must* be defined with the *same* material ID number in the *MATERIAL data group.

Group ID card:

*PLASTIC

The data for a typical elastoplastic material consists of two sets of cards. The first set consists of one card that describes the overall control parameters. The second set of cards may consist of more than one card depending on the parameters selected in the first set. Both sets should be repeated for additional elastoplastic materials, if any.

Card set 1: One card, always required

Entry No:	1	2	3	4
Variable:	LABEL	MATID	NTYSSC	NYCRIT
Max Char:	4	6	1	1

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	LABEL	card label to designate the beginning of the definition of a typical elastoplastic material. = PLAS or CONC
2	MATID	material identification number. This is the material ID number referenced on the element definition data group, *ELEMENTS. All other mechanical properties for the same material (elastic properties, mass density and thermal expansion coefficients) <i>must</i> be given in the *MATERIAL data group with the <i>same</i> material ID number.
3	NTYSSC	selection of stress-strain curve = 0 - elastic-perfectly plastic = 1 - Ramberg-Osgood representation (see note 1) = 2 - elastic-linear hardening

Model Data

Material Data

Table 6.5: Required plasticity properties for different combinations of the parameters NTYSSC and NYCRIT (see note 6)

NTYSSC \ NYCRIT	0	1	2	≥ 3
	(elastic-perfectly plastic)	(Ramberg-Osgood)	(elastic-linear hardening)	(elastic-piecewise linear hardening)
0, 1 (von Mises, Tresca, Ilyushin)	SY0	BETA, EXPO, SIG7, SY0	BETA, SY0, HARD	BETA, STRN, STRS, SY0
2, 3 (Mohr-Coloumb, Drucker-Prager)	COHE, FRIC	Not available	COHE, FRIC, HARD	COHE, FRIC, STRN, STRS
4, 5, 6 (Willam-Warnke 5-parameter model, Ottosen's criterion, Hsieh-Ting-Chen criterion)	SY0, CSIG, TSIG, CSTR, TSTR, FRAE, SHRC, CRIT	Not available	BETA, SY0, HARD, CSIG, TSIG, CSTR, TSTR, FRAE, SHRC, CRIT	BETA, STRN, STRS, SY0, CSIG, TSIG, CSTR, TSTR, FRAE, SHRC, CRIT, NHAR

\$

Entry No:	1	2	3	4	5	6	7	8
Variable:	LABEL	MATID	TEMPID	VAL1	VAL2	VAL3	VAL4	VAL5
Max Char:	4	6	10	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	LABEL	label indicating the property defined on this card. Refer to Table 6.5 for the properties required to complete the definition of the material (<i>labels required depend on the values of NTYSSC and NYCRIT defined on the first set</i>). Valid labels are: = BETA control parameter for hardening rule. The valid values of this parameter, that have to be specified in entry number 4 (VAL1), are: = 0.0-for purely kinematic hardening rule

- = 1.0-for purely isotropic hardening rule [*default*]. The default value of BETA = 1.0 applies only when the card label BETA is entirely omitted.
- > 0.0 and < 1.0 - mixed hardening rule (a combined isotropic- and kinematic hardening rule).
- = EXPO hardening index for Ramberg-Osgood stress-strain curve, see note 1.
- = COHE material cohesion (C).
- = FRIC angle of internal friction (f) in degrees.
- = SIG7 >0 parameter for yield offset (α) in Ramberg-Osgood formula (see note 1).
 - <0 uniaxial stress at a secant modulus of 0.7E for Ramberg-Osgood stress-strain curve ($\sigma_{0.7}$), see note 1.
- = STRN strain values *beyond* yield point.
- = STRS stress values *beyond* yield point.
- = SY0 initial uniaxial yield stress.
- = HARD work hardening parameter (H) for linear hardening material, see note 2.
- = CSIG Compressive strength of concrete (c)
- = TSIG Tensile strength of concrete (t)
- = CSTR Maximum compressive strain (c)
- = TSTR Maximum tensile strain (t)
- = NHAR Number of hardening segments (n)
- = FRAE Fracture energy (G^f)
- = SHRC Shear reduction coefficient (k)
- = CRIT Criteria code for tension failure
 - 1 - Linear softening
 - 2 - Bilinear softening
 - 3 - Quadratic softening
 - 4 - Cubic softening
- = CAP Parameters for CAP model
- = USR1 additional material constants for user-defined material model, see note 6.

Model Data

Material Data

- = USR2 additional material constants for user-defined material model, see note 6.
- 2 MATID material identification number. This is the material ID number referenced on the element definition data group, *ELEMENTS. All other mechanical properties for the same material (elastic properties, mass density and thermal expansion coefficients) *must* be given in the *MATERIAL data group with the *same* material ID number.
- 3 TEMPID temperature dependence key. Enter zero if entry number 1 is BETA, STRN or STRS.
- = 0 - material property is temperature independent.
- = 1 - material property is temperature dependent and is represented by a fourth order polynomial function of temperature.
- > 1 - material property is temperature dependent. TEMPID is the temperature increment at which the property is given, starting at a temperature of zero.
- tab (\$)--
- 4-8 VAL1 value of the material property designated by the LABEL. These entries to VAL5 have the following interpretation depending on the LABEL and the temperature dependence key TEMPID:
- if LABEL = STRN or STRS:
- VAL1 to VAL5 are 'NTYSSC' number of total strain or stress values of the uniaxial stress-strain curve describing the hardening behavior.
- if TEMPID = 0:
- VAL1 = value of the property.
- VAL2 to VAL5 are not required.
- if TEMPID = 1:
- VAL1 to VAL5 are the coefficients of the fourth order polynomial expression for variation of the property with temperature, i.e., constant term, linear term, quadratic term, etc.
- if TEMPID > 1:
- VAL1 to VAL5 are the values of the property at T0, T0+TEMPID, T0+2*TEMPID, etc., where T0 = 0.0. Not all the 5 values may be required, see note 3.

Notes:

- When SIG7 value is positive, NISA2 uses the Ramberg-Osgood stress-strain curve as

$$\varepsilon = \frac{\sigma}{E} + \alpha \frac{\sigma_0}{E} \left(\frac{\sigma}{\sigma_0}\right)^n$$

When SIG7 value is negative, NISA2 uses the Ramberg-Osgood stress-strain curve as

$$\varepsilon = \frac{\sigma}{E} + \frac{3}{7} \frac{\sigma_{0.7}}{E} \left(\frac{\sigma}{\sigma_{0.7}}\right)^n$$

where

ε, σ = the axial strain and stress respectively.

E = modulus elasticity

σ_0 = yield stress

$\sigma_{0.7}$ = uniaxial stress at a secant modulus of 0.7E (= |SIG7| and SIG7<0)

α = parameter for yield offset (= SIG7 and SIG7>0)

n = hardening index (EXPO)

- The work hardening parameter H (HARD) for a linear hardening material is given by:

$$H = \frac{E E t}{E - E t}$$

where, $E_{and} E_t$ are the elastic and elastoplastic tangent moduli (slopes of the uniaxial stress-strain curve) respectively.

- If $TEMPID > 1$, not all the values of VAL1 to VAL5 have to be given. However, the last value given should be greater than or equal to the highest temperature at any point in the model.
- The Ilyushin yield criterion (refer to [Section 2.7.3](#)) is used in triangular 3-node thin shell element (NKTP = 40, NORDR = 10) only. The parameter BETA always uses the default value of 1.0 in the Ilyushin model. The value of β (refer to [Equation 2.325](#)) is assumed to be zero.
- For user-defined material model, both NTYSSC and NYCRIT must be set to -1.
- For user-defined material model, i.e. when NTYSSC = -1 and NYCRIT = -1, the labels are no longer required. If assigned, however, NISA will take them as plain material constants and pass all of them to the user-defined material model to be re-assigned. USR1 and USR2 are supplemented to those existing labels. Therefore a total of 50 material constants may be given to each user-defined material type from NISA input file. Note either SY0 or COHE can be given. See [Appendix D.3](#) for details.

Model Data

Material Data

7. For defining stress-strain curve up to 1000 points in the model, set NTYSSC = -2 and NYCRIT = -1, and define STRSTRN.DAT in the working folder. The format for defining stress-strain points and related information in external file is given below.

Card set 1:

Entry No: 1
Variable:

MATTOT

Max Char: 6

Entry Variable Description

1 MATTOT Total number of materials for which Stress-Strain data given in STRSTRN.DAT

Card set 2:

Entry No: 1
Variable:

COMMENT

Max Char: 80

Card set 3:

Entry No: 1
Variable:

COMMENT

Max Char: 80

Card set 4:

Entry No: 1 1 1 1 1
Variable:

IDUSR	MAXSSC	NYCRIT	SY0	BETA
-------	--------	--------	-----	------

Max Char: 6 6 1 10 10

Entry Variable Description

1 IDUSR Stress-Strain curve ID, should be equal to MATID in the *MATERIAL card
2 MAXSSC Maximum number of points (Maximum 1000 points)

Model Data

Material Data

- 3 NYCRIT Yield Criteria (Refer *PLASTIC)
- 4 SY0 Initial uniaxial yield stress
- 5 BETA control parameter for hardening rule

Card set 5:

Entry No: 1 1
Variable:

VAL1	VAL2
------	------

Max Char: 10 10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	VAL1	Strain values beyond yield point.
2	VAL2	Stress values beyond yield point.

Stress at yield: 300.0MPa; Strain at yield: 0.00150

STRAIN	STRESS (MPa)
0.00321	310.000
0.00342	320.000
0.00365	330.000
0.00388	340.000
0.00413	350.000
0.00439	360.000
0.00466	370.000
0.00495	380.000
0.00525	390.000
0.00556	400.000

6.5.7 *APLASTIC Data Group - Anisotropic Plastic Properties for Elastoplastic Material Models.

This data group is required in nonlinear analysis to describe the anisotropic plasticity parameters and yield criteria for elastoplastic material models. All other mechanical properties for the same material (elastic properties, mass, density and thermal expansion coefficients) *must* be defined with the *same* material ID number in the *MATERIAL data group. Strength properties for orthotropic composite elements in *linear analysis* (defined in the *MATERIAL data group) are not required for nonlinear analysis, and they will be ignored if given, see notes 3 and 7.

Group ID card:

*APLASTIC

The data for a typical anisotropic elastoplastic material consists of two sets of cards. The first set consists of one card that describes the overall control parameters. The second set of cards consists of more than one card. The number of cards in the second set depends on the overall control parameters defined in the first set of cards. Both sets should be repeated for additional anisotropic elastoplastic materials, if needed.

Card set 1: One card, always required

Entry No:	1	2	3	4
Variable:	LABEL	MATID	NTYSSC	NYCRIT
Max Char:	4	6	1	1

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
--------------	-----------------	--------------------

1	LABEL	card label to designate the beginning of the definition of a typical elastoplastic material = APLA
---	-------	--

Model Data

Material Data

- 2 **MATID** material identification number. This is the material ID number referenced on the element definition data group *ELEMENT (or in case of 3-D composite shell element NKTP = 32, it is referenced on the lamination sequence definition data group, *LAMSQ2, see note 2). All other mechanical properties for the same material (elastic properties, mass density and thermal expansion coefficients) must be given in the *MATERIAL data group with the *same* material ID number. However, the strength properties which are required in the *MATERIAL data group for linear analysis need not be given for nonlinear analysis. If given, they will be ignored.
- 3 **NTYSSC** Selection of stress-strain curve
 = 0 elastic-perfectly plastic (see note 4)
 = 1 Ramberg-Osgood representation (see note 5)
 = 2 elastic-linear hardening
 ≥ 3 elastic piecewise linear hardening. NTYSSC, in this case, is the number of points used to define the stress-strain curve beyond yield (maximum of 5 points)
- 4 **NYCRIT** selection of yield criterion
 = 0 Hill's anisotropic theory [Default]
 = 1 Modified Hill's anisotropic theory to take into account different yield strengths in tension and compression.

Card set 2:

This set consists of more than one card. Each card begins with a label to indicate the property being defined. However, not all the cards listed in this set are required to complete the definition of the plasticity properties of typical elastoplastic material (see note 1).

For example in the case of 3-D laminated composite shell element (NKTP = 32), the stress normal to the shell is assumed to be zero, and it is not required to specify the plastic properties in the third principal material direction. However it is required to specify the plastic properties along the two principal material directions, along the inplane and transverse shear directions as well as along a direction making 45° to the fiber direction. Users should refer to the element library ([Chapter 4](#)) for the required properties for each element type.

For easy reference, the applicable property labels for all combinations of the two parameters NTYSSC and NYCRT are listed in [Table 6.6](#).

Table 6.6: *Applicable* anisotropic plasticity property labels for each direction or plane for various combinations of the parameters NTYSSC and NYCRT

NTYSSC / NYCRT	0	1	2	≥ 3
	(elastic-perfectly plastic)	(Ramberg-Osgood)	(elastic-linear hardening)	(elastic-piecewise linear hardening)
0, Hill's anisotropic yield function	SY0(see note 7)	SY0, EXPO, SIG7, BETA	SY0, HARD, BETA (see notes 6, 7)	SY0, STRN, STRS, BETA
1 Modified Hill's anisotropic yield function	Same property labels as above, but repeated for both tensile and compressive directions.			

\$

Entry No:	1	2	3	4	5	6	7	8
Variable:	LABEL	IDIR	TEMPID	VAL1	VAL2	VAL3	VAL4	VAL5
Max Char:	4	6	10	10	10	10	10	10

Entry Variable Description

- | | | |
|---|-------|--|
| 1 | LABEL | <p>label indicating the property defined on this card. Refer to Table 6.6 and to the element library (Chapter 4) for required property labels to complete the definition of the material. Valid labels are:</p> <ul style="list-style-type: none"> = BETA control parameter for hardening rule. The valid values of this parameter, that have to be specified in entry number 4 (VAL1), are: <ul style="list-style-type: none"> = 1.0-for purely isotropic hardening rule [default]. The default value of BETA = 1.0 applies only when the card label BETA is entirely omitted. = EXPO hardening index for Ramberg-Osgood stress-strain curve definition, see note 5 |
|---|-------|--|

Model Data

Material Data

= HARD work hardening parameter (H) for linear hardening material, see note 8

= SIG7 uniaxial stress at a secant modulus of 0.7E for Ramberg-Osgood stress-strain curve ($\sigma_{0.7}$), see note 5.

= STRN strain values *beyond* yield point.

= STRS stress values *beyond* yield point.

= SY0 initial yield stress.

2 IDIR direction ID indicating the principal material directions or 45° (except for 3-D solid elements) to the fiber direction for which the property defined in LABEL is given.

= 11 first principal material direction

= 22 second (or transverse) principal material direction

= 33 third principal material direction

= 12 in plane shear direction

= 23 transverse shear direction

= 13 another transverse shear direction

All the above directions will correspond to compressive properties if negative signs are added to them, i.e., -11, -22, -33. These compressive properties are required only for the modified Hill's theory where NYCRIT = 1.

3 TEMPID temperature dependence key. Enter zero if entry number 1 is BETA, STRN or STRS.

= 0 material property is temperature independent.

= 1 material property is temperature dependent and is represented by a fourth order polynomial function of temperature.

> 1 material property is temperature dependent. TEMPID is the temperature increment at which the property is given, starting at a temperature of zero.

--- tab(\$) ---

4-8 VAL1 value of the material property designated by the LABEL. These entries to VAL5 have the following interpretation depending on the LABEL and the temperature dependence key TEMPID:

if LABEL = STRN or STRS:

VAL1 to VAL5 are 'NTYSSC' number of *total* strain or stress values of the uniaxial stress-strain curve describing the hardening behavior.

if TEMPID = 0:

VAL1 = value of the property.

VAL2 to VAL5 are not required.

if TEMPID = 1:

VAL1 to VAL5 are the coefficients of the fourth order polynomial expression for variation of the property with temperature, i.e., constant term, linear term, quadratic term, etc.

if TEMPID > 1:

VAL1 to VAL5 are the values of the property at T0, T0+TEMPID, T0+2*TEMPID, etc., where T0 = 0.0. Not all the 5 values may be required, see note 9.

Notes:

1. Refer to element library ([Chapter 4](#)) for the required labels for each element.
2. For most NISA elements, the variable MATID in the *ELEMENTS data group refers to a material property table defined in *MATERIAL data group. However, this is inadequate for composite shells, since each layer may be a different material. Therefore, for this element, MATID is a pointer to an entry in the *LAMSQ2 (lamination sequence) data group. The *LAMSQ2 data group in turn contains pointers to the layer thicknesses, rotation angles, and materials for all layers (*RCTABLE, *LAMANGLE and *MATERIAL data groups, respectively). In this way, we retain the ability to model composite shells which have tapered thickness or variable rotation angles. There is no limitation on the number of layers or the lay up sequence. Note that the number of real constant tables, lamination angle tables, or material property tables may not necessarily be equal to the number of layers, since more than one layer may be of the same material type, thickness type, or lamination angle (orientation) type.
3. Strength properties for orthotropic composite elements in *linear analysis* are defined in the *MATERIAL data group, but they are not needed, and will be ignored, in nonlinear analysis.
4. It should be noted that for a specific material ID, the values of NTYSSC (selection of stress-strain curve) and NYCRIT (selection of the yield criteria) *cannot* change from one direction to another. First defined values (of NTYSSC and NYCRIT) for a given material ID will override other specifications.

Model Data

Material Data

5. The Ramberg-Osgood stress-strain curve is represented by the formula:

$$\varepsilon = \frac{\sigma}{E} + \frac{3}{7} \frac{\sigma_{0.7}}{E} \left(\frac{\sigma}{\sigma_{0.7}} \right)^n$$

where,

- ε, σ = the uniaxial strain and stress, respectively.
 - E = modulus of elasticity.
 - $\sigma_{0.7}$ = uniaxial stress at a secant modulus of 0.7E (SIG7).
 - n = hardening index (EXPO).
6. As an example, consider an anisotropic plastic material defined by material ID = 100, NTYSSC = 2 (elastic linear hardening) and assume that this material is going to be used with a general 3-D state of stress. The following input set-up will be needed to specify the required material properties:
- For NYCRIT = 0; Hill's anisotropic yield function,
and BETA = 1.0; pure isotropic hardening [Default]:

*APLASTIC
APLA, 100, 2, 0
SYO, 11,---
HARD, 11,---
SYO, 22,---
HARD, 22,---
SYO, 33,---
HARD, 33, ---
SYO, 12, ---
HARD, 12, ---
SYO, 23, ---
HARD, 23,---
SYO, 13,---
HARD, 13,---

- For NYCRIT = 1; Modified Hill's anisotropic yield function, the same input as above is required, except for the addition of properties in the compressive directions -11, -22,... et.
7. The work hardening parameter H (HARD) for a linear hardening material is given by:

$$H = \frac{EE_t}{E - E_t}$$

where E and E_t are the elastic and elastoplastic tangent moduli (slopes of the uniaxial stress-strain curve), respectively.

8. If TEMPID > 1, not all the values of VAL1 to VAL5 have to be given. However, the last value given should be greater than or equal to the highest temperature at any point in the model.

6.5.8 *CREEP Data Group - Creep Material Law

This data group is required in nonlinear analysis to define material creep law. Various forms of material creep or creep rate laws are available. The general form of creep laws considered here is:

$$\dot{\epsilon}^c = F_1(\sigma) F_2(t) F_3(T)$$

where, $\dot{\epsilon}^c$ is the uniaxial or effective creep strain, σ is the effective stress, t is the time and T is the temperature. The choice of the functions $F_1(\sigma)$, $F_2(t)$, and $F_3(T)$ are described in this data group. A different form of creep law, the ORNL law, is also described below. Creep strain rate, $\dot{\epsilon}^c$ may be specified by the same functions if the value of $n1$ on the CREEP parameter in the *EVENT data group is set to unity (see *EVENT data group). Each creep material will have a unique material identification number MATID. At least one property should be given in the *MATERIAL data group for the same material MATID.

Group ID card:

*CREEP

The data for a typical material consists of two sets of cards. The first set consists of one card that describes the overall control parameters. The second set may consist of more than one card depending upon the type of the creep law selected in the first card.

Card set 1: One card, always required

Entry No:	1	2	3	4	5	6	7	8
Variable:	LABEL	MATID	NSIG	NTIME	NCON1	NCON2	NCON3	NTEMP
Max Char:	4	6	1	1	1	1	1	1

Entry Variable Description

- | | | |
|---|-------|--|
| 1 | LABEL | label to designate the general parameters of creep law
= CRLW |
| 2 | MATID | material identification number. This is the same ID referenced in the element definition group *ELEMENTS. At least one elastic property value should be given under *MATERIAL for each material ID specified here. |
| 3 | NSIG | Selection of stress function type (F1 (s)) in the creep law. |

- = -1 User defined creep law (See note 1)
- = 1 Norton : $K\sigma^n$
- = 2 McVetty : $A \sinh(k\sigma)$
- = 3 Soderberg : $B [\exp(k\sigma)-1]$
- = 4 Dorn : $C \exp(k\sigma)$
- = 5 Johnson : $D_1\sigma^{m1} + D_2\sigma^{m2}$
- = 6 Garofalo : $A [\sinh(k\sigma)]^m$
- = 7 ORNL law : $A(\sigma) [1-e^{-R(\sigma)t}] + H(\sigma)t$ (see note 2)

where K, n, A, k, B, C, D1, D2, m1, m2, and m are the number of constants defined in NCON1 (entry #5)

4 NTIME Selection of function of time t in the creep law ($F_2(t)$):

- = 1 Andrade : $(1 + bt^{1/3}) \exp(kt)-1$ for NSIG = 1 to 6
: or ORNL law type-1 for NSIG = 7 (see note 2)
- = 2 Bailey : Ft^n for NSIG = 1 to 6
: or ORNL law type-2 for NSIG = 7 (see note 2)
- = 3 McVetty : $G(1-e^{-qt}) + Ht$
- = 4 Graham and Walles : $\sum a_i t^{n_i}$ i = 1,2

where b, k, F, n, G, q, H, a_i and n_i are constants (Note that if G, q and H are functions of the stresses, ORNL creep law should be used, see note 2).

- 5 NCON1 number of constants in the stress function $F_1(\sigma)$, maximum 4 constants
- 6 NCON2 number of constants in the time function $F_2(t)$, maximum 4 constants
- 7 NCON3 number of constants in the temperature function $F_3(T)$, maximum 4 constants
- 8 NTEMP selection of temperature function in creep law ($F_3(T)$):

- = 1 : $C_1 \exp\left(\frac{C_2}{C_3 T + C_4}\right)$ (see note 3)

Model Data

Material Data

Card set 2: More than one card, always required

Each card begins with a label to identify the coefficients being defined. The coefficients required for each function (stress, time or temperature) are listed in [Table 6.6](#). The *order* of the coefficients given in the table *must be preserved* with the coefficients for the stress function given first followed by the coefficients for the time function.

Table 6.7: Required coefficients for various functions defining the material creep law

Function Time	NSIG or NTIME	Required Constants	Equivalent labels
<u>Stress Function</u>			
Norton	1	K, n	FA1, FA2, FA3, and FA4 (depending on how many constants are required)
McVetty	2	A, k	
Soderberg	3	B, k	
Dorn	4	C, k	
Johnson	5	D1, m1, D2, m2	
Garofalo	6	A, k, m	
<u>Time Function</u>			
Andrade	1	b, k	FB1, FB2, FB3, and FB4 (depending on how many constants are required)
Bailey	2	F, n	
McVetty	3	G, q, h	
Graham and Walles	4	a1, n1, a2, n2	
<u>Temperature Function</u>			
Exponential	1	C1 C2 C3 C4	FC1, FC2, FC3, and FC4 (depending on how many constants are required)

\$

Entry No:	1	2	3	4	5	6	7	8
Variable:	LABEL	MATID	TEMPID	VAL1	VAL2	VAL3	VAL4	VAL5
Max Char:	4	6	10	10	10	10	10	10

<u>Entry Variable</u>	<u>Description</u>
1 LABEL	label identification, e.g. FA1, FA2, FA3, FA4, FB1,... The available list of labels is given in Table 6.6 . The order given in the table <i>must be preserved</i> when entering the coefficients.
2 MATID	material ID number. This is the same material ID number referenced on the element definition data group, *ELEMENTS. At least one elastic property value should be given under the *MATERIAL data group for each MATID.
3 TEMPID	temperature dependence key = 0 - material coefficient is temperature independent. = 1 - material coefficient is a polynomial function of temperature > 1 - temperature intervals at which coefficients are given
- - -tab(\$) - - -	
4-8 VAL1 to VAL5	value of the material property designated by the LABEL entry. These entries have the following interpretation depending on the temperature dependence key TEMPID: <u>if TEMPID = 0:</u> VAL1 = value of the property. VAL2 to VAL5 are not required. <u>if TEMPID = 1:</u> VAL1 to VAL5 are the coefficients of the fourth order polynomial expression for variation of the property with temperature, i.e., constant term, linear term, quadratic term, etc. <u>if TEMPID >1:</u> VAL1 to VAL5 are the values of the property at T0, T0+TEMPID, T0+2*TEMPID,etc., where T0 = 0.0. Not all the 5 values may be required, see note 4.

Notes:

1. Skip other variables.
2. The ORNL creep law is of the form:

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

To initiate this creep law, the user should set NSIG = 7 (the third entry on card set 1) and NTIME = 1 or 2 (fourth entry on card set 1). The value of NTIME define the form of the functions A(σ), H(σ) and R(σ) as shown in [Table 6.7](#).

3. The unit of this temperature is the same unit as the nodal temperature given in *NDTEMP data group. If different temperature units are needed in temperature function and nodal temperature, the temperature unit in the temperature function can be adjusted by changing constants C₂ and C₃.

Table 6.8: Function forms for ORNL creep law

Function	Type1 NTIME = 1	Type2 NTIME = 2	Comments
A (σ) R (σ)	a ₁ exp (b ₁ σ) k σ^n	k σ^n A ₁ exp (k ₁ σ)	Define constants under F ₁ σ i.e., FA1, FA2, FA3, and FA4
H (σ)	B ₁ exp (k ₂ σ)	B ₂ [sinh(k ₃ σ)] ^m	Define constants under F ₂ (t) i.e., FB1, FB2, and FB3

The constraints a₁, b₁, k, and n (or k, n, A₁, and k₁) are given as F₁ (σ), i.e., FA1, FA2, FA3, and FA4. The constants B₁ and k₂ (or B₂, k₃, and m) are given as F₂(t) using FB1, FB2, and FB3 constants.

6.5.9 *PDAMPING Data Group - Proportional Damping Coefficients

Applicable analysis type: LTRANSIENT, NLTRANSIENT

This data group can be used to define the constants for proportional damping (see note 1).

Group ID card:

*PDAMPING

Entry No:	1	2	3	4	5
Variable:	NELID	DAMP1	DAMP2	LASTEL	INC
Max Char:	6	12	12	6	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NELID	element ID as defined in *ELEMENT data group. (see note 2)
2	DAMP1	Proportional constant of element stiffness matrix (C1) for proportional viscous damping
3	DAMP2	Proportional constant of element mass matrix (C2) for proportional viscous damping
4	LASTEL	last element of a range of elements with the same damping coefficients DAMP1 and DAMP2. Enter 0 if this card defines damping coefficients for only one element.
5	INC	increment for the range of elements. Enter 0 if coefficients are defined for one element (see note 3)

Notes:

1. For proportional viscous damping, the damping matrix at the element level, $\mathbf{C}^{(e)}$, is defined here as a linear combination of the element stiffness matrix, $\mathbf{K}^{(e)}$ and the element mass matrix $\mathbf{M}^{(e)}$:

$$\mathbf{C}^{(e)} = C1 * \mathbf{K}^{(e)} + 2 * C2 * \mathbf{M}^{(e)}$$

2. If element ID is zero (i.e. no element ID is supplied), the proportional damping constants DAMP1 and DAMP2 are used for all the elements. In such case, the traditional proportional damping (Rayleigh's damping) is achieved:

$$\mathbf{C} = C1 * \mathbf{K} + 2 * C2 * \mathbf{M}$$

where \mathbf{C} , \mathbf{K} and \mathbf{M} are the global damping, stiffness and mass matrices, respectively.

Model Data

Material Data

3. Additional cards can be used to specify different constants for different elements. If an element ID is repeated, the last specification will be used for that element.

6.5.10 *MATPIEZO data group – Material properties for piezoelectric analyses

This data group is always required in piezoelectric analyses to specify coefficients of piezoelectric matrix and dielectric matrix.

Group ID card: *MATPIEZO

Piezo Material properties card set:

Entry No:	1	2	3	4	5
Variable:	LABEL	MATID	DUMMY	DUMMY	COEF0
Max char:	4	6	6	6	12

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	LABEL	label for the piezoelectric matrix coefficients and dielectric matrix coefficients. PZ11 PZ12 PZ13 -coefficients of piezoelectric matrix in X, Y, and Z axes (see note. 1) PZ21 PZ22 PZ23 PZ31 PZ32 PZ33 PZ41 PZ42 PZ43 PZ51 PZ52 PZ53 PZ61 PZ62 PZ63 EXX EYY EZZ - diagonal coefficients of dielectric matrix
2	MATID	material identification number (must be an integer between 1 and 999999). This is the material ID referenced on the element definition group (*ELEMENT). It should have the same material ID defined in *MATERIAL data group for mechanical properties of this material.
3	DUMMY	not use
4	DUMMY	not use
5	COEF0	value of the coefficient

Notes:

1. Piezoelectric matrix and dielectric matrix are defined as follows:

$$\{S\} = [C] \{\varepsilon\} - [P] \{E\}$$

$$[D] = [P]^T \{\varepsilon\} + [E] \{V\}$$

Where

$\{S\}$ = stress vector $\{S_{xx} S_{yy} S_{zz} S_{xy} S_{yz} S_{xz}\}^T$

$\{\varepsilon\}$ = strain vector $\{\varepsilon_{xx} \varepsilon_{yy} \varepsilon_{zz} \varepsilon_{xy} \varepsilon_{yz} \varepsilon_{xz}\}^T$

$[C]$ = elasticity matrix

$[P]$ = piezoelectric matrix

PZ11 PZ12 PZ13

PZ21 PZ22 PZ23

PZ31 PZ32 PZ33

PZ41 PZ42 PZ43

PZ51 PZ52 PZ53

PZ61 PZ62 PZ63

$\{V\}$ = electric field vector $\{V_x V_y V_z\}^T$

$\{D\}$ = electric flux density vector $\{D_x D_y D_z\}^T$

$[E]$ = dielectric matrix

EXX 0 0

0 EYY 0

0 0 EZZ

x, y, and z are the directions in a Cartesian coordinate system. Any component which is not defined in this data group will be set to zero.

2. It should be noted that the piezoelectric and dielectric properties are direction dependent. The value of these properties will vary with different reference frames. The global Cartesian frame is used as the default reference frame. If other reference frame is needed, *MATDIR1 or MATDIR2 data group should be used to define a proper reference frame for piezoelectric properties even though elastic material properties are isotropic. However, there is no need to activate orthotropic material key (KISO = 1 in *ELEMENT data group) if the mechanical properties are isotropic.

***REINFORCEMENT Data group – Reinforcement bar definition**

Applicable analysis type: NLSTATIC

This data group defines a reinforcement bar in the 3D concrete element (NKTP=31). It is neither element nor line geometry. This reinforcement bar can be defined as straight or curved bar depends upon the number of points. The cross section area of the bar is specified through real constant table. Elasto-plastic material properties are given through *MATERIAL and *PLASTIC data cards.

Group ID card:

REINFORCEMENT

Card set 1:

Entry no:	1	2	3	4				
Variable:	<table border="1"><tr><td>REINID</td></tr></table>	REINID	<table border="1"><tr><td>MATID</td></tr></table>	MATID	<table border="1"><tr><td>IDRC</td></tr></table>	IDRC	<table border="1"><tr><td>NPOINT</td></tr></table>	NPOINT
REINID								
MATID								
IDRC								
NPOINT								
Max Char:	8	6	6	6				

Entry no:	Variable:	Description:
1	REINID	-Reinforcement bar identification number
2	MATID	-Material identification for reinforcement bar
3	IDRC	-Real constant table identification number
4	NPOINT	-Number of points (or coordinates) required to represent reinforcement bar.

Model data

Reinforcement Data

Card set 2: (more than one card, if reinforcement bar require more than three points to be specified)

Entry no:	1	2	3	4	5	6	7	8	9
Variable:	X1	Y1	Z1	X2	Y2	Z2	X3	Y3	Z3
Max char:	12	12	12	12	12	12	12	12	12

Notes:

1. Two points (in global coordinates) are sufficiently required to specify a reinforcement bar. The first and second points coordinates are considered as start and end points of the reinforcement bar respectively.
2. If required, the reinforcement bar can be specified with three points, where first and third points are considered as start and end points respectively and the second point would be intermediate point of the curved reinforcement bar.
3. Up to five points can be given to specify reinforcement bar, where the first and last point would be start and end points of the reinforcement bar respectively and other points would be intermediate points of the curved reinforcement bar.

6.6 Kinematic Constraint Data

6.6.1 *RIGLINK Data Group - Rigid Element Data

This data group may be used to define rigid elements in structural analysis. A rigid element (or link) is a set of kinematic constraint equations which express a typical displacement degree of freedom at a slave (dependent) node in terms of the degrees of freedom at the master (independent) node. The corresponding multi-point constraint equations are generated *internally* using the kinematics of rigid link (see [Section 3.10](#) and note 10).

Group ID card:

*RIGLINK

Rigid element card set:

	\$						
Entry No:	1	2	3	4	5	6	7
Variable:	IDRE	NODM	NSLV	NSET	INC	ND1	K1
Max Char:	6	6(8)	2	3	3	6(8)	6

Entry No:	8	9	10-13	14	15
Variable:	ND2	K2	...	ND5	K5
Max Char:	6(8)	6	...	6(8)	6

Rigid elements may be defined using two methods: individual definition and first level generation, as shown below.

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IDRE	rigid element ID number
2	NODM	node number of the master node (node to which independent degrees of freedom of the element are assigned).
3	NSLV	total number of slave nodes (nodes whose degrees of freedom are dependent on the master node). This variable may be entered as zero, see note 1.
4	NSET	- not used for individually defined rigid elements, enter zero

- for first level generation, this is the number of rigid elements to be generated from the base element, including the base element.
- 5 INC - not used for individually defined rigid elements, enter zero.

- for first level generation, this is the node number increment. Additional rigid elements with the same ID number will be generated by adding INC to the node number of the base element.
- tab (\$) ---
- 6 ND1 node number of the first slave node
- 7 K1 identification code for the dependent degree(s) of freedom of the first slave node. This code identifies the degrees of freedom at the slave node for which the MPC equations corresponding to the rigid element are to be generated.

123456 generate 6 MPC equations for the rigid link. The dependent degrees of freedom of these MPC equations are degree of freedom 1 through 6 at the slave node. This represents 3-D rigid link.

126 generate 3 MPC equations for the rigid link. The dependent degrees of freedom in these MPC equations are UX, UY, and ROTZ at the slave node, respectively.

0 The number of MPC equations generated for the rigid link in this case will be equal to the number of degrees of freedom existing at the slave node (as indicated by the degrees of freedom of all elements attached to the slave node).

Other combinations of the number 1 through 6 may be used in any order. Degrees of freedom at the slave node which are not referenced in the identification code will not be affected by the rigid link, and are not considered as dependent degrees of freedom.
- 8 ND2 node number of the second slave node, if any
- 9 K2 identification code for the dependent degree(s) of freedom of the second slave node
-
-as required...
-

Model Data

Kinematic Constraint Data

- | | | |
|----|-----|---|
| 14 | ND5 | node number of the fifth slave node (if required) |
| 15 | K5 | identification code for the dependent degree(s) of freedom of the fifth slave node. |

Notes:

1. Up to 5 slave nodes may be entered on one card. The specification of the number of slave nodes (NSLV, entry 3) is optional since the number of slave nodes is counted internally. Additional cards needed to define additional slave nodes must start with a tab character (\$). For example, the 3-D rigid link 101 has node 100 as the master node and nodes 1, 2, 3, 4, 5, 6 as the slave nodes. The input may be entered as follows:

```
101, 100, 0 $1, 0, 2, 0, 3, 0, 4, 0, 5, 0
      $6, 0
```

Or alternatively as:

```
101, 100, 6 $1, 0, 2, 0, 3, 0
      $4, 0, 5, 0, 6, 0
```

2. Slave nodes used in the definition of rigid elements must also appear in the list of element connectivities (*ELEMENTS data group).
3. A slave node of a rigid link must be connected to NISA elements that possess structural stiffness in the directions of the dependent degrees of freedom at the slave node.
4. Dependent degrees of freedom assigned to a slave node of a rigid link (through the identification code) cannot be referenced again (as dependent) by any other rigid link, MPC data, or coupled displacement data. That is, a degree of freedom may be referenced as a slave only once.
5. Dependent degrees of freedom assigned to a slave node of a rigid link cannot be constrained in the specified displacement data.
6. An independent degree of freedom at a master node of a rigid element can be referenced by any subsequent rigid link, MPC data, or coupled displacement data as an independent or a dependent degree of freedom. However, once it becomes a dependent degree of freedom, all the above rules apply to it.

7. An independent degree of freedom at a master node (that remains an independent degree of freedom) can be constrained in the specified displacement data (non-zero specified displacement value is allowed).
8. If a displacement local coordinate system is defined at any node which is referenced in rigid element data, then the corresponding MPC equations for that node of the rigid link are in that local coordinate system.
9. The generated MPC equations in this group and the MPC equations defined directly in *MPCEQN are resequenced *internally* to eliminate any slave sequence dependency (a degree of freedom can be referenced as a slave only after it makes all of its appearances as a master, if any, in all the previous MPC equations - e.g., a chain of rigid links). A fatal error message is printed out if the resequencing process fails. This may happen if there is an inherent sequence dependency among the MPC equations that cannot be removed by shuffling the equations, e.g., rigid links in a closed loop. The resequencing process does not change a designated slave degree of freedom to a master, or vice-versa. The resequencing process does not include degrees of freedom listed in the coupled displacement data group (*CPDISP). The user must make sure that a slave degree of freedom defined in the *RIGLINK or the *MPCEQN does not appear in the coupled displacement data.
10. For nonlinear analysis, the MPC equations corresponding to the rigid link are generated in a linearized form ([Section 3.10](#)) based on the undeformed geometry. These MPC equations are not changed during load increments. The user is cautioned that this may lead to inaccurate results in the case of geometric nonlinearities if the degrees of freedom involved in the equations undergo large displacements or rotations.
11. The printout of rigid link forces are controlled by the KELFR quantity in the *LDCASE card and the RLFO key in the *PRINTCNTL card. In the case of coincident master and slave nodes, rigid link forces are reported in the global cartesian coordinate system or in the displacement local coordinate system (DLCS) at the slave node, if one is defined. When the master and slave nodes of a rigid link are distinct, rigid link forces are reported in the element local directions, where the local X direction is defined by a line from the master node to the slave node.

6.6.2 *MPCEQN Data Group - Definition of Linear Multi-Point Constraint (MPC) Equations

This data group may be used to define constraint equations involving any *admissible* linear combination of displacement degrees of freedom in the model.

Group ID card:

*MPCEQN

MPC card set:

Entry No:	1	2	3	4	5	6	7	8	9	10	11	12
Variable:	LB1	ND1	A1	LB2	ND2	A2	LB3	ND3	A3	LB4	ND4	A4
Max Char:	4	6	10	4	6	10	4	6	10	4	6	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	LB1	identification label for the first degree of freedom (dependent) in the MPC equation. Allowable labels are: UX, UY, UZ -translation along X, Y and Z axis, respectively ROTX, ROTY, -rotation about X, Y and Z axis, respectively ROTZ CONS -the right-hand-side constant NEXT -The definition of the MPC equation is continued on the subsequent card. All entries following the label 'NEXT' on the same card are ignored. This label must not be the first entry on any one card. See note 2
2	ND1	first node number in the MPC equation
3	A1	first coefficient in MPC equation
4	LB2	identification label for the second degree of freedom (independent) in the MPC equation
5	ND2	second node number in the MPC equation
6	A2	second coefficient in the MPC equation
:		
:		. as required...

- | | | |
|----|-----|---|
| 10 | LB4 | identification label for the fourth degree of freedom (independent) (if required) |
| 11 | ND4 | fourth node number (if required) |
| 12 | A4 | fourth coefficient in the MPC equation (if required) |

Notes:

1. Multi-point constraint equations are of the following form:

$$A1*X1 + A2*X2 + A3*X3 + \dots + An*Xn = CONS$$

Ai = coefficients represented by A1, A2, etc.

Xi = degrees of freedom defined by LBi and NODEi

CONS = the constant term

It should be noted that a nonzero constant term is allowed only in linear static and linear direct transient dynamic analyses.

2. Use as many cards as required to define each MPC equation. Up to four terms in the equation may be defined on one card. The label 'NEXT' should be used if only one to three terms of the equation are entered on one card and the definition of the equation has not been completed yet. For example, consider the MPC equation.:

$$3 UX_1 - UX_2 - UX_3 - UX_4 = 0.0$$

Input may be entered as

UX, 1, 3.0, UX, 2, -1.0, UX, 3, -1.0, UX, 4, -1.0

CONS, 0, 0.0

OR alternatively as

UX, 1, 3.0, UX, 2, -1.0, UX, 3, -1.0, NEXT

UX, 4, -1.0, CONS, 0, 0.0

3. The end of the definition of an MPC equation is determined by the occurrence of a blank label or the right hand side constant (CONS).
4. If the right hand side constant is zero, it need not be entered. If it is non-zero, it must be the last term in the equation and two commas must separate the label (CONS) from its value.

Model Data

Kinematic Constraint Data

5. The dependent degree of freedom in each equation is the first one mentioned, i.e., LBI at ND1.
6. The dependent degree of freedom in an MPC equation cannot be constrained (in *SPDISP data group). However, the independent degrees of freedom may be constrained
7. A dependent degree of freedom in one equation cannot be a dependent degree of freedom again in another equation. Nor can it be a dependent degree of freedom in a coupled displacement or rigid element data.
8. A dependent degree of freedom of an MPC equation must also be a degree of freedom of a NISA element that possesses structural stiffness.
9. If a local displacement coordinate system has been defined at any node which is referenced in any MPC equation, then the corresponding LBI designates a degree of freedom in this local coordinate system.
10. The right hand side constant (CONS) may be nonzero only in linear static and linear direct transient dynamic analyses.
11. The MPC equations entered here as well as those generated internally for the *RIGLINK data group are resequenced internally to eliminate any slave sequence dependency problems. (See note 9 in [Section 6.6.1](#)).

6.6.3 *CPDISP Data Group - Coupled Displacement Data

This data group may be used to represent a simple form of kinematic constraints, which equates specific components of displacement at more than one node to each other.

Group ID card: ***CPDISP**

Coupled displacement card set:

Coupled displacements sets may be defined using two methods: individual definition and first level generation as shown below

	\$							
Entry No:	1	2	3	4	5	6	7-13	14
Variable:	LABEL	NSET	NODINC	NODSET	NODE1	NODE2	...	NODE10
Max char	4	2	6	3	6(8)	6(8)	...	6(8)

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	LABEL	label representing displacement component. Allowable labels are: UX, UY, UZ - translations along X, Y and Z axes, respectively. ROTX, - rotations about X, Y and Z axes, respectively. ROTY, ROTZ UXYZ - all translations RXYZ - all rotations ALL - all translations and all rotations
2	NSET	- not used in individual definition, enter zero - for first level generation, this is the number of sets of coupled displacements to be generated from the base set, including the base set.
3	NODINC	- not used in individual definition, enter zero - for first level generation, this is the node number increment. Additional sets of coupled nodes will be generated by adding NODINC to the node numbers in the base set.
4	NODSET	number of coupled nodes in this set (or in the base set for first level generation). This variable may be entered as zero, see note 1.

Model Data

Kinematic Constraint Data

--- tab (\$) ---

5	NODE1	first node in this set (the referenced displacement component at the first node is the independent DOF)
6	NODE2	second node
...		
... as required...
...		
14	NODE10	tenth node in this set

Notes:

1. *Up to 10* coupled nodes may be entered on one card. The specification of the number of coupled nodes (NODSET, entry 4) is optional since the number of coupled nodes is counted internally. Additional cards needed to define additional coupled nodes must start with a tab character (\$). For example, if the UX displacement is the same at nodes 1 through 11, the input may be entered as follows:

```
UX $1, 2, 3, 4, 5, 6, 7, 8, 9, 10  
    $11
```

Or alternative as:

```
UX $1, 2, 3, 4  
    $5, 6, 7, 8  
    $9, 10, 11
```

2. The displacement component of the first node (NODE1) is the independent (master) degree of freedom; the others are dependent degrees of freedom.
3. A dependent degree of freedom must not be referenced again (as independent or dependent) in any subsequent coupled displacement definition. Moreover, it must not be designated as dependent in any rigid element data (*RIGLINK data group) or MPC data (*MPCEQN data group).
4. An independent degree of freedom must have not been designated as dependent in any rigid element data (*RIGLINK data group) or MPC data (*MPCEQN data group).

Model Data

Kinematic Constraint Data

5. Dependent degrees of freedom which appear in this data group may not be constrained in the *SPDISP data group.
6. If a local displacement coordinate system is defined at a node included in the coupled displacement list, then the local components of displacement are coupled.

6.6.4 *CPTEMP Data Group - Coupled Temperature Data for Heat Transfer Analyses

This data group may be used in heat transfer analyses to represent a temperature constraint, which equates the unknown temperatures at some nodes to each other.

Group ID card: ***CPTEMP**

Coupled temperature card set:

Coupled temperature sets may be defined using two methods: individual definition and first level generation, as shown below.

	\$							
Entry No:	1	2	3	4	5	6	7-13	14
Variable:	LABEL	NSET	NODINC	NODSET	NODE1	NODE2	...	NODE10
Max char:	4	2	6	3	6	6	...	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	LABEL	label representing temperature - must be TEMP
2	NSET	- not used in individual definition, enter zero - for first level generation, this is the number of sets of coupled temperatures to be generated from the base set, including the base set.
3	NODINC	- not used in individual definition, enter zero - for first level generation, this is the node number increment. Additional sets of coupled nodes will be generated by adding NODINC to the node numbers in the base set.
4	NODSET	number of coupled nodes in this set (or in the base set for first level generation). This variable may be entered as zero, see note 1.
--- tab (\$) ---		
5	NODE1	first node in this set (the temperature of the first node is the independent DOF)
6	NODE2	second node
:		
:		... as required...

Model Data

Kinematic Constraint Data

:

14 NODE10 tenth node in this set

Notes:

1. Up to 10 coupled nodes may be entered on one card. The specification of the number of coupled nodes (NODSET, entry 4) is optional since the number of coupled nodes is counted internally. Additional cards needed to define coupled nodes must start with a tab character (\$). For example, if the temperature is the same at nodes 1 through 11, the input may be entered as follows:

```
TEMP $1, 2, 3, 4, 5, 6, 7, 8, 9, 10
      $11
```

Or alternatively as:

```
TEMP $1, 2, 3, 4
      $5, 6, 7
      $8, 9, 10,11
```

2. The temperature of the first node (NODE1) is the independent (master) degree of freedom; the others are dependent degrees of freedom.
3. A dependent nodal temperature must not be referenced again (as independent or dependent) in any subsequent coupled temperature definition.
4. Nodal temperatures which appear in this data group may not be constrained in the *SPTMP data group.

6.6.5 *RIG2LINK Data Group – Weighted Average Link Element Data

This data group is used to define the movement of a dependent (slave) node as the weighted average movement of a set of independent (master) nodes. From the weighted factors and the participating degrees of freedom of the master nodes, a set of kinematic relations between displacements of the slave node and the displacements of master nodes will be created as multi-point constraint equations (MPC) internally. Opposite to rigid link element which has one master node and multiple slave nodes, this data group has only one slave node and multiple master nodes.

Group ID card:

*RIG2LINK

Rig2link element card set:

Card 1:

Entry No:	1	2	3	4	5	6	7	8	9	10
Variable:	IDRG2	SNOD	SDOF	NMA5	MNOD1	MDOF1	MWT1	MNOD2	MDOF2	MWT2
Max Char:	8	8	6	4	8	6	10	8	6	10

Card 2:

Entry No:	1	2	3	4	5	6	7	8	9
Variable:	MOD3	MDOF3	MWT3	MOD4	MDOF4	MWT4	MNOD5	MDOF5	MWT5
Max Char:	8	8	10	8	8	10	8	8	10

Card 2 can be repeated as many as it needs to specify all the independent (master) nodes. In case there is less than two master nodes, card 2 can be omitted.

Entry Variable Description

- | | | |
|---|-------|----------------------------------|
| 1 | IDRG2 | element ID number of RIG2LINK |
| 2 | SNOD | dependent (slave) node ID number |

- 3 SDOF identification code (123456) of the participating DOF of the dependent node.
- all number 1 through 6 stand for DOF of UX to ROTZ. Code 123456 means 6 DOF of the slave node participate and 6 MPC equations for DOF UX to ROTZ will be generated for the slave node. Code 126 will generate 3MPC equations for DOF UX, UY, ROTZ for the slave node. There should not be space or blank in between the numbers in the identification code
- 4 NMAS total number of independent (master) nodes in this RIG2LINK element
- 5 MNOD1 node ID of the first independent (master) node
- 6 MDOF1 identification code of the first master node
- 7 MWT1 weight factor used for the first master node. No default value for the weighted factor.

1.0 is recommended if there is no specific need for different value for different master nodes.

- 8 MNOD2 node ID of the second master node
- 9 MDOF2 identification code of the second master node
- 10 MWT2 weight factor for the second master node
- .
- .
- .
- ... MNODN node ID of the Nth master node
- ... MDOFN identification code of the Nth master node
- ... MWTN weight factor for the Nth master node

Notes:

1. It is recommend that for most applications only the translation degrees of freedom 123 are specified for the master nodes. In case the master nodes and slave node are collinear, at least one rotation degree of freedom should be specified to avoid singularity or rigid body motion of the element.
2. There is no default value for weight factor.
3. Independent (master) node used in the RIG2LINK element must be also appears in the list of the element connectivity (*ELEMENTS data group).
4. Dependent degrees of freedom of a slave node cannot be reference again as slave node of a rigid link , MPC data, or coupled displacement data. That means the slave degree of freedom can only be reference as slave once.
5. Dependent degrees of freedom of a slave node cannot be constrained in the specified displacement data.
6. An independent degrees of freedom at a master node can be referenced by any subsequent rigid link, MPC data or coupled displacement as an independent or a dependent degree of freedom. However, once it becomes a dependent degree of freedom, all the above rules apply to it.
7. The printout of RIG2LINK element forces are controlled by the option of KELER in the *LDCASE card and the R2FO option in the *PRINTCNTL card. Both of these two options have to be activated in order to get RIG2LINK element forces in the output file. For those nodes have displacement local coordinate system defined, the element forces at those nodes will be in their local coordinate system.

6.7 Miscellaneous Data

6.7.1 *SETS Data Group - Definition of Sets

Sets of integer numbers (nodes or element IDs) may be defined in this data group for subsequent reference in the *PRINCNTL data group, for output requests.

Group ID card:

*SETS

Sets definition card set:

	\$	\$									
Entry No.:	1	2	3	4	5	6	7	8	9	10	11
Variable	IDSET	LABEL	I1	J1	K1	I2	J2	K2	I3	J3	K3
Max char.:	6	1	6(8)	6(8)	6(8)	6(8)	6(8)	6(8)	6(8)	6(8)	6(8)

<u>Entry Variable</u>	<u>Description</u>
1 IDSET	set identification number, integer > 0
---\$---	
2 LABEL	card identification label (to interpret the remaining entries on this card). Allowable labels are: S = entries on this card are single numbers R = entries on this card define range of numbers E = entries on this card are single numbers to be excluded from any ranges of numbers defined in this set
---\$---	

Model Data

Miscellaneous Data

- 3-11 I1, J1, K1, interpretation of these entries depend on the card identification LABEL
.... K3 such that:
I1, J1, K1, ..., K3 are single number defined in the set, when LABEL is set
to 'S'
(I1,J1,K1), (I2,J2,K2), (I3,J3,K3) define three ranges of numbers begin-
ning with Ii, ending with Ji and in increment of Ki, for i = 1,2,3, respec-
tively, when LABEL is set to 'R'
I1,J1,K1, ..., K3 are single numbers to be excluded from any ranges of
numbers defined in this set, when LABEL is set to 'E'

Notes:

1. LABEL must not be blank for the first card used to define the set.
2. If LABEL is left blank for any subsequent card (defining the same set), the LABEL of the previous card is used.
3. Define as many sets as desired, each set must start with a unique non-zero 'IDSET'
4. If any set requires more than one card, continue with additional cards. Additional cards must start with a tab character \$. If any additional card starts with two consecutive tab characters \$\$, then the LABEL of the previous card is used.
5. Once a set is defined, a non-zero value of 'IDSET' on any subsequent card will begin the definition of a new set, if any.

Example 6.6.1:

Set identification number 101 defines the set of numbers:

1, 4, 8, 20 to 30, 50 to 60, 200 to 220, 1000 to 1030 (in increments of 10)

Set identification number 201 defines the set of numbers:

100 to 150, 152, 153, 155, 157, 163, 171, 182, 183, 191, 195, and excludes the numbers 101, 109, 140

Input is as follows:

**

*SETS

**	IDSET	LABEL	I1	J1	K1	I2	J2	K2	I3	J3	K3
**	101,	S	1,	4,	8,	50	60	1	200,	220,	1
	\$	R	20,	30,	1,						
	\$	S	1000	1030	10						
**											
	201,	R,	100,	150,	1						
	\$,	S,	152,	153,	155,	157	163	171	182,	183,	191
	\$,	\$,	195,								
	\$,	E	101,	109,	140						

6.7.2 *VECTORS Data Group - Definition of Vectors

This data group may be used to define vectors. These vectors may then be referenced in other data groups for miscellaneous use. One such use is in the *ELEMENTS data group to define an orientation vector for the determination of local coordinate systems for 3-D beam elements (NKTP = 11, NKTP = 12) or 3-D general spring element (NKTP = 38).

Group ID card:

*VECTORS

Vector definition card set:

Entry No.:	1	2	3	4
Variable	IDVEC	XCOMP	YCOMP	ZCOMP
Max char:	6	12	12	12

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IDVEC	vector ID number
2	XCOMP	global X-component of the vector
3	YCOMP	global Y-component of the vector
4	ZCOMP	global Z-component of the vector

Note:

1. Define as many vectors as desired, one vector per card.

6.7.3 *TEMPFN Group Data - Temperature Dependency Curves for Heat Transfer Analysis

This data group may be used in *heat transfer analyses* to define temperature dependency curves. These curves may be referenced in other data groups (e.g., *MATHEAT, *NDHEATGEN, *DFLUX) to indicate the variation of a quantity of interest with respect to temperature. For example, this includes temperature dependent material properties, internal heat generation, distributed heat flux, etc. Thus, a typical curve is used to compute the value of a quantity of interest at a given temperature. *Two* sets of cards are required for a typical curve. Repeat the same card set sequence for additional curves, if any.

Group ID card:

*TEMPFN

Card set 1: one card

Entry No.:	1	2
Variable	IDCURV	NPOINT
Max char:	6	2

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IDCURV	temperature curve identification number, integer > 0.
2	NPOINT	number of data points used to define the curve.

Card set 2: may require more than one card.

Entry No.:	1	2	3	4	5	6	7	8
Variable	T1	P1	T2	P2	T3	P3	T4	P4
Max char:	10	10	10	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1-8	(Ti-Pi)	temperature and scaling factor values (entered in pairs) for a typical point on the curve.

Model Data

Miscellaneous Data

Notes:

1. Use as many cards as necessary to define the NPOINT number of data points (4 points per card). The data points *must* be entered in ascending order of temperature.
2. A typical curve should cover all possible range of temperature values encountered in the analysis. Scaling factors corresponding to temperature values outside the specified range are assumed to be equal to the nearest value obtained from the curve, whereas amplitudes within the specified temperature range will be linearly interpolated.
3. Linear interpolation is used to compute the scaling factor for a given temperature. The exact value of a quantity of interest (e.g., distributed heat flux) at a given temperature is equal to the scaling factor times the base value given in the corresponding data group (e.g., *DFLUX).

6.7.4 *TIMEAMP Data Group - Definition of Time Amplitude Curves

This data group may be used in *nonlinear static, transient heat transfer or direct transient dynamic* analyses to define a time-amplitude curve. This curve may be referenced in some analysis data groups such as *PRESSURE, *SPDISP, *DFLUX, *NDHEATGEN, etc. The curve will specify the incremental and/or the total values of the pertinent quantity at each step or time instant. See example below.

Group ID card:

*TIMEAMP

Two sets of cards are required for *each* time amplitude curve to be defined. The first set consists of one card which defines the curve ID and the number of points on the curve. The second set defines time and amplitude values and may consist of more than one card if more than four points are specified on the curve. The maximum number of time amplitude curves allowed is 128.

Card set 1: one card

Entry No	1	2	3
Variable	IDCURV	NPOINT	IFORM
Max char	6	2	1

Entry Variable Description

- | | | |
|---|--------|--|
| 1 | IDCURV | curve identification number, integer > 0. This is the number that will be referenced by other analysis data groups. |
| 2 | NPOINT | number of data points used to define the curve |
| 3 | IFORM | data format input key
= 0: the data will be input as 4 pairs per card
= 1: the data will be input as 1 pair per card |

Model Data

Miscellaneous Data

Card set 2:

For IFORM = 0: more than one card if more than four pairs are specified

Entry No.:	1	2	3	4	5	6	7	8
Variable	T1	A1	T2	A2	T3	A3	T4	A4
Max char:	10	10	10	10	10	10	10	10

For IFORM = 1: more than one card if more than one pair is specified

Entry No.:	1	2
Variable	T1	A1
Max char:	20	20

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
--------------	-----------------	--------------------

1-8	(Ti, Ai)	pairs of time and amplitude values of a typical point on the curve. Use as many cards as required to define the NPOINT data points, four pairs per card if IFORM = 0 and one pair per card if IFORM = 1. Time values <i>must</i> be entered in ascending order.
-----	----------	---

Notes:

1. For nonlinear static analysis (excluding creep analysis), time is a pseudo time and time increments are equivalent to load increments or steps.
2. A typical curve should cover all possible range of time values encountered in the analysis. Amplitudes corresponding to time values outside the specified range will be assumed zero, whereas amplitudes within the specified time range will be linearly interpolated.
3. At a given time increment, the incremental value of a quantity (e.g., pressure, flux) will be equal to the change in amplitude, during this time increment, times the base value of the quantity specified on its corresponding data group. At a given instant of time, the total value of a quantity will be equal to the amplitude value, at this time, times the base value of the quantity specified on its corresponding data group. See example 6.6.2 for details.

4. Default time amplitude curves are internally defined by two points as follow:
- For nonlinear static analysis
(0.0,0.0),(t,1.0), where t = TIMEATEND specified in *EVENT data group
 - For heat transfer analysis
(0.0,1.0),(1.0,1.0)
 - For direct transient dynamic analysis
(0,1.0), (t,1.0), where t = TIMEATEND specified in *EVENT data groups.
 - Amplitude values beyond the highest time specified on the time-amplitude curve are taken as zero.

Example 6.6.2: Time Amplitude Curve Definition

The following cards define the time amplitude curve shown in [Figure 6.15](#). The curve identification number is 111 and 7 data points are specified. Analysis data groups may reference the curve identification number to specify changes in a specific quantity. Details of the program interpretation for incremental and total values of the pertinent quantity are shown after the card input.

**Card set 1, defines curve ID and number of points

111, 7, 0

**Card set 2, defines the time-amplitude pairs. Two cards are required since 7 points are specified

** (4 points per card).

0.0, 0.0, 2.0, 0.0, 5.0, 1.0, 6.0, 1.0

7.0, 0.0, 8.0, 0.0, 10.0, 0.5

To explain the interpretation of the program for the above curve, we assume the following case:

- An analysis group, e.g. *PRESSURE, is referring to the above curve ID.
- The base value of the pressure, specified on *PRESSURE, is 15 units.
- The analysis uses 10 steps or time increments.

Then, the incremental and total values of the pressure interpreted by the program at each step will be as follows: (see note 2 above).

Step no.	1	2	3	4	5	6	7	8	9	10
Incremental value	0.0	0.0	5.0	5.0	5.0	0.0	-15.0	0.0	3.75	3.75
Total value	0.0	0.0	5.0	10.0	15.0	15.0	0.0	0.0	3.75	7.5

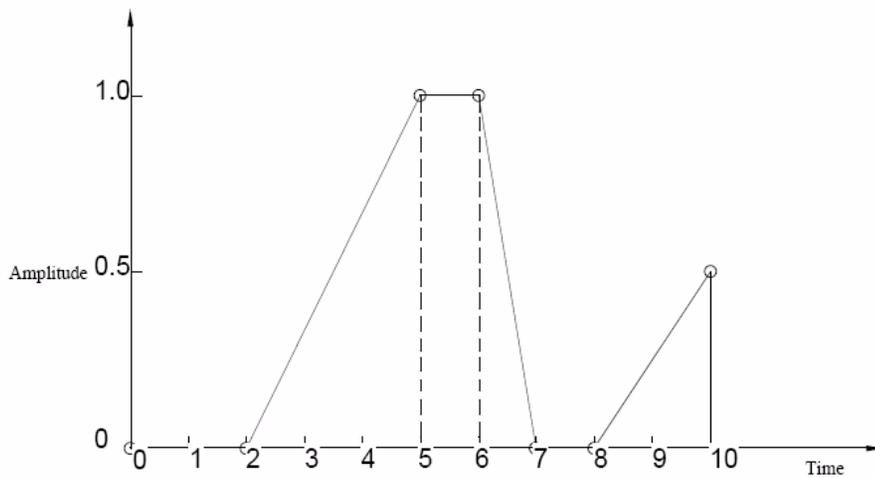


Figure 6.15: Time-amplitude curve for example 6.6.2

6.7.5 *PCHANGE1 - Enthalpy vs. Temperature Curve for Phase Change Analysis

This data group is required in *transient heat transfer* analysis if phase change is considered. This group describes the enthalpy value as a function of temperature.

Group ID card:

*PCHANGE1

Enthalpy vs. temperature definition card set:

Entry No.:	1	2	3	4	5	6	7	8
Variable	T1	H1	T2	H2	T3	H3	T4	H4
Max char:	10	10	10	10	10	10	10	10

<u>Entry Variable</u>	<u>Description</u>
1-8 (Ti, Hi)	Pairs of temperature and enthalpy values. Four pairs per card. Use as many cards as required to describe the curve.

Notes:

1. If the input of temperature and enthalpy point is not in ascending order of temperature, the program sorts out the data in ascending order of temperature values.
2. A typical curve should cover all possible ranges of temperature values encountered in the analysis. Enthalpy values corresponding to temperature values outside the specified range will be assumed to be equal to the nearest value obtained from the curve, whereas amplitudes within the specified temperature range will be linearly interpolated.
3. A typical enthalpy vs. temperature curve is shown in [Figure 6.16](#), where the transition or phase change region is represented by the temperature range T_L to T_S . Within the specified temperature range, the heat capacity (the product of material density, ρ and specific heat, c) is computed from the curve as the gradient of enthalpy with respect to temperature, (dH/dT) . On the other hand, outside the specified temperature range, the heat capacity values are calculated directly from the values of ρ and c given in the *MATHEAT data group.

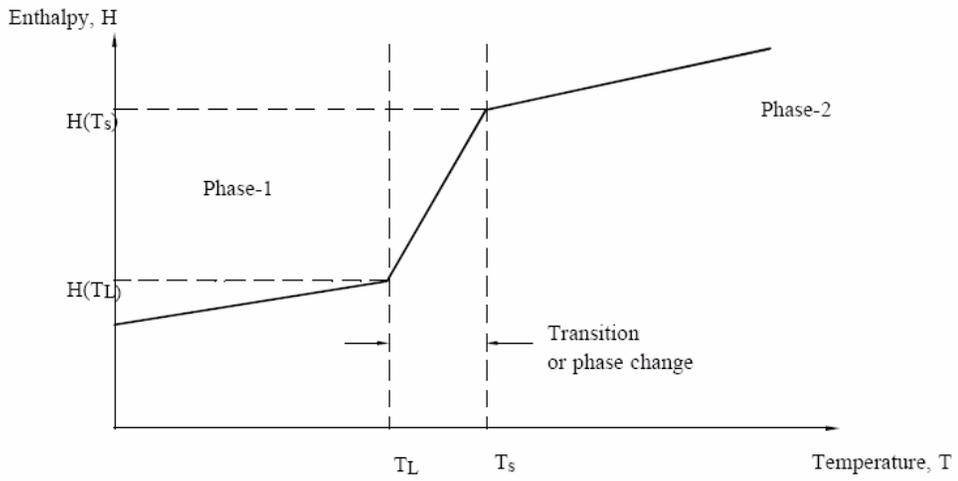


Figure 6.16: A typical enthalpy vs. temperature curve

6.7.6 *CRACKNODES Data Group – Crack Nodes Definition Data

Application analysis types: STATIC, NLSTATIC

This data group is optional and may be used to specify nodes lying on the crack mouth and crack front.

Card set 1: Crack Nodes Identification Card

Entry No:	1	2	\$3	4	5	6	7	8	9	10	11
Variable:	ID	LABEL	N1	N2	N3	N4	N5	N6	N7	N8	N9
Max char:	6	5	6(8)	6(8)	6(8)	6(8)	6(8)	6(8)	6(8)	6(8)	6(8)

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	ID	Set Identification number, Integer >0
2	LABEL	FRONT – Entries on this card are Node IDs on the Crack Front MOUTH – Entries on this card are Node IDs on the Crack Mouth
3, 4, ...	N1, N2, ..	Set of Node Ids on the Crack as one moves from one end to another. Up to 9 node Ids may be entered on a card

Notes:

1. LABEL must not be blank for the first card used to define the crack nodes
2. If LABEL is left blank for any subsequent card (defining same set of crack nodes) the LABEL of the previous card is used
3. Define as many as crack nodes desired, each set must start with a unique non-zero 'ID'
4. If any set requires more than one card, continue with additional cards. Additional cards must start with a tab character \$. If any additional card starts with two consecutive tab characters \$\$, then the LABEL of the previous card is used.
5. Once a set of crack nodes is defined, a non-zero value of 'ID' on any subsequent card will begin the definition of new set, if any.
6. The sequence of Crack nodes to be keyed in is from depth point of crack to the surface point. For the blunt crack node numbers to be keyed from free to fixed node along the blunt curve.

Example: 6.6.6

Cracks can be modeled as semi-elliptical surface cracks or through cracks, with blunt or sharp edges.

Semi elliptical Cracks:

The node IDs on crack front and crack mouth are represented using *CRACK-KNODES. The nodes lying on each semi circular crack front are represented by Ids such as 101,102,... starting from crack tip to crack center. The number of nodes, on the crack front, for the blunt crack will be more than one, where as for the sharp crack it will be only one node.

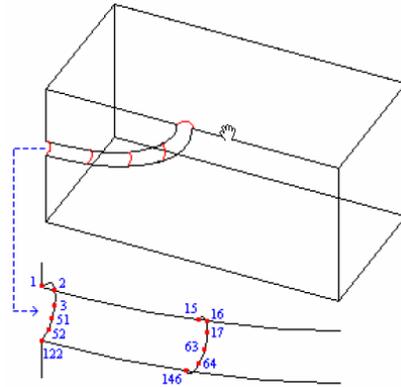


Figure 6.6.3a: Part-through (Blunt) Crack front nodes

Blunt Cracks: (Fig. 6.6.3a)

***CRACKNODES**

101, FRONT, 1, 2, 3, 51, 52, 122
 102, FRONT, 15, 16, 17, 63, 64, 146

 ...105, FRONT,,,,,

Sharp Cracks: (Fig. 6.6.3b)

***CRACKNODES**

101, FRONT, 143
 102, FRONT, 147

 107, FRONT, 458

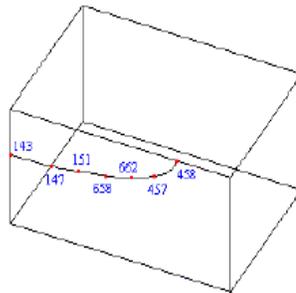


Figure 6.6.3b: Part-through (Sharp) Crack front nodes

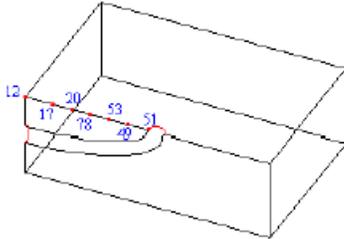


Figure 6.6.3c: Part-through (Blunt)
Crack mouth nodes
201, MOUTH, 12, 17, 20, 78, 49, 53,
49, 51

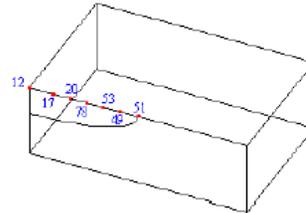


Figure 6.6.3d: Part-through (Sharp) Crack
mouth nodes
201, MOUTH, 12, 17, 20, 78, 53, 49, 51

Through Cracks:

The node IDs on crack front and crack mouth are represented using *cracknodes. The nodes lying on each semi circular crack front are represented by Ids such as 101, 102, ... starting from one end of the crack to the other end. The number of nodes for blunt crack will be more than one, where as for sharp crack it will have only one node

Blunt Cracks: (Fig 6.6.4a)

*CRACKNODES

101, FRONT, 398, 402, 366, ..., .., ...

\$, \$, ..., 862

102, FRONT, 382, 386, 370, ..., .., ...

\$, \$, ..., 842

.....
105, FRONT, ..., .., ...

201 MOUTH, 12, 14, 16, 398

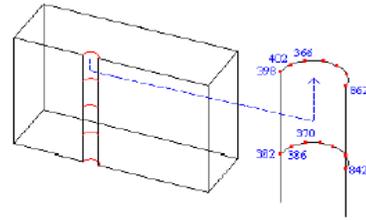


Figure 6.6.4a:Through (Blunt) Crack front nodes

Sharp Cracks (Fig 6.6.4c)

*CRACKNODES

101, FRONT, 398

102, FRONT, 382

.....
201 MOUTH, 12, 14, 16, 398

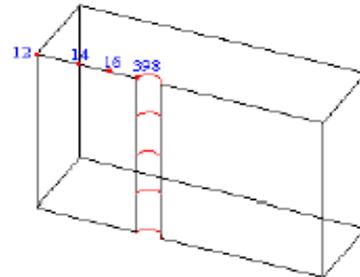


Figure 6.6.4b:Through (Blunt) Crack mouth nodes

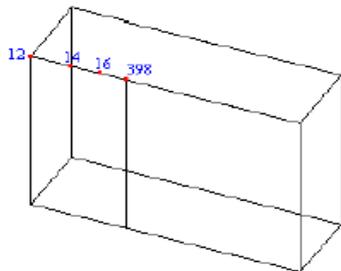


Figure 6.6.4d:Through (Sharp) Crack mouth nodes

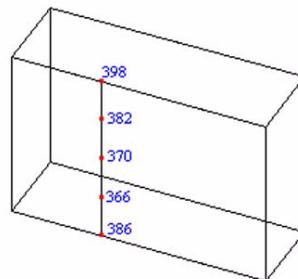


Figure 6.6.4c:Through (Sharp) Crack front nodes

6.7.7 *LDFILE data group – load history file data

This data group is used to input file names and the path for load history files. Load history files are snapshot data for different time frames for different type of loads related to *NDTEMPER, *NDTEMPDIF, *PRESSURE, and *BEAMLOAD data groups (see note 1 and 2). Different load type should have different snapshot file but only one snapshot file should be given for the same load type. Number of time frames and the sampling time for snapshot can vary in different load files. There is no limit how many time frames can be used in a load history file. Time frames of the snapshot file can be in any order. Sorting will be done internally to create a more efficient binary format file for computation. This binary file can be saved for future use to eliminate unnecessary data processing. During an analysis, linear interpolation scheme will be used to compute the values in between time frames. Load history files are not required to be in the working directory and the path card set can be used to specify the directory of the load history files. However, all the load history files should be in the same directory.

Group ID card:

*LDFILE

Two sets of cards may be used in this data group. Only the first set is required. The first set defines load type and the name of the load history file. If more than one load type is needed, more cards can be used to define for different load types. The second set is optional. It defines the path of the time history files.

Card set 1: May be more than one card. If four types of load are needed, four cards are required.

Entry No:	1	2	3	4	5	6
Variable	LDTYPE	DFILE	DDEL	BFILE	BDEL	KPRNT
Max char	9	20	1	20	1	1

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	LDTYPE	Name of data group will be used in the history data file. Four types of data group are allowed to use in load history file are NDTEMPER NDTEMPDIF PRESSURE

BEAMLOAD

- 2 DFILE Name of the history data file in text format.
- 3 DDEL Key to keep or delete DFILE after reading data.
 = 0 keep DFILE after reading data
 = 1 delete DFILE after reading data
- 4 BFILE Name of the history data file in binary format. (note 3)
- 5 DDEL Key to keep or delete BFILE after reading data.
 = 0 keep BFILE after reading data
 = 1 delete BFILE after reading data
- 6 KPRNT Key to print DFILE or BFILE content in NISA output file
 = 0 no printing
 = 1 printing file content

Card set 2: Optional card. If time history files are located in the working directory, this card set is required.

Entry No.	1	2
Variable	PATH	DIRPAT
Char	4	64

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	PATH	Identifier for card set 2 'path' should be used.
2	DIRPAT	path of the directory where time history files are sitting.(see note 4)

Notes:

1. Load history file is a list of same data groups (NDTEMPER, NDTEMPDIF, PRESSURE, BEAMLOAD) with specified time as second entry in the data group title card and the file end with *ENDDATA card. The data format of the data group is exactly the same as one NISA2 input file. For example pressure load history file:

```
*PRESSURE, TIME = 0.0
5,10,1,2,0,1,0.0
*PRESSURE, TIME = 0.2
5,10,1,2,0,1,0.2
*PRESSURE, TIME = 1.0
5,5,1,2,0,1,0.7
6,6,1,2,0,1,0.8
7,7,1,2,0,1,0.9
8,8,1,2,0,1,0.9
9,10,1,2,0,1,1.0
*ENDDATA
```

temperature load history file:

```
*NDTEMPER, TIME = 0.0
1,TEMP,0.0,25,1
*NDTEMPER, TIME = 0.5
1,TEMP,50.0,24,1
25,TEMP,20.0,25,1
*NDTEMPER, TIME = 1.0
1,TEMP,100.0,24,1
25,TEMP,80.0,25,1
*ENDDATA
```

2. Time frames given in load history file should at least cover the whole time interval in the analysis. One exception is at time zero. If no snapshot data is given at time zero and values at time zero are needed then zero will be used as the value at that time.
3. When a DFILE is given, NISA2 processes the data in DFILE and creates a binary file for computation. If the binary file is saved, it can be used in the future analysis to save data processing time. Only one of DFILE or BFILE is needed. In case of both are presented, DFILE will be used.

4. The format of DIRPAT is operating system dependent. Users should consult the system manual how to set path. The path name should include all characters just before the file name. For example, the path \USER\NISA\TMP in WINDOW98 operating system should be given as \USER\NISA\TMP\, the last back slash is required.

6.7.8 *STFUNCTION data group – Definition of standard periodic time function

This data group provides a convenient and efficient way to define a time amplitude curve for a periodic time function without using *TIMEAMP data group, which may require a large number of data points to cover the time length of an analysis. Only parameters of the periodic time function are needed regardless of time length in the analysis. This data group may be used in nonlinear static, direct linear transient dynamic, and nonlinear dynamic analyses. The time function may be referenced in analysis data groups such as *PRESSURE, *SPDISP, *CFORCE, and *ECHAR etc.

Group ID card:

*STFUNCTION

Two sets of cards are required for each standard time function. The first card consists of two data. The first entry defines the standard function ID, which will be referenced by the other data groups (see note 1), and the second entry selects the type of periodic time function. The second card defines the value of required parameters for the periodic time function. Number of data and the definition of the data in the second card vary for different types of function (see [Table 6.9](#) for details).

Card set 1: one card

Entry No:	1	2	3	4
Variable:	IDCURV	ITYPE	TSTRT	TEND
Max char:	6	3	10	10

Entry Variable Description

- 1 IDCRV standard function identification number, integer >0. This is the number that will be referenced by the other analysis data groups (see note 4)
- 2 ITYPE identification number for different type of standard time function.
- 3 TSTRT time at which the standard time function becomes active
- 4 TEND time at which the standard time function becomes inactive (note 5)

Card set 2:

Value of parameters for different types of standard time functions. Table lists

Entry No:	1	2	3	4	5	6	7
Variable:	P1	P2	P3	P4	P5	P6	P7
Max char:	10	10	10	10	10	10	10

Table 6.9: Definition of standard periodic time function

TYPE	FORMULA
1	$F = P1 * \sin(P2 * T + P3) + P4 + P5 * \cos(P6 * T + P7)$ Where T is the current time
2	$F = 4 * P2 * (T / P1) \text{ for } P1/4 > T > 0$ $F = P2 * (2 - 4T / P1) \text{ for } 3 * P1/4 > T > P1/4$ $F = 4 * P2 * (T / P1 - 1) \text{ for } P1 > T > 3 * P1/4$

Notes:

1. Standard time function can be seen as a convenient form of input for time amplitude function. The standard time function ID (IDCRV) is referenced by other NISA data groups such as *SPDISP, *CFORCE etc. as a time amplitude curve (TCRV). In order to tell the referenced curve (TCRV) is from a standard time function, a negative curve ID should be given in the time amplitude curve (TCRV) entry.
2. The standard time function is used for a periodic time function. The current time can be longer than one cycle. For any given time, it will compute the value at that cycle.
3. For nonlinear static analysis (excluding creep analysis), time is a pseudo time and time increments are equivalent to load increments or steps.
4. At a given time, the value of a quantity (e.g. pressure, flux etc.) will be equal to the value of the standard time function times the base value of the quantity specified on its corresponding data.
5. TSTRT and TEND are optional input. The standard time function is inactive (or zero value) before TSTRT and after TEND. If these two entries are not given, the standard time function is active at time zero and no specific end time. If TSTRT is given and TEND is zero, the standard time function is active after TSTRT. When both TSTRT and TEND are given, end time (TEND) should be later than start time (TSTRT).

6.7.9 *PRESTN data group – definition of pre-stresses/strains

Applicable analysis types: **NLSTATIC**, **NLTRANSIENT**

This data group is used for nonlinear static or nonlinear transient dynamic analysis to specify elastic pre-stresses/strains in elements before external loads are applied to the model. Current release supports this feature for four types of element (NKTP = 1,2,3,4). Six global components of pre-stresses/strains can be specified at each node of an element. Element shape functions are used to interpolate nodal values to Gauss points for computation. All pre-stresses/strains are assumed in elastic range. Elements not defined in this data group will have no pre-stresses/strains.

Group ID card:

*PRESTN

This data group uses two sets of cards to define pre-stresses/strains for each element. The first set of card contains one data card to specify element ID, number of cards in the second set of data, and type of input (pre-stresses or pre-strains). The second set of cards is used to define prestresses/ strains for each node. Number of cards in this set is defined in the first set. The maximum number of cards should not exceed the number of nodes for that element.

Card set 1: one card, always required

Entry No.:	1	2	3
Variable:	NELID	NCARDS	ITYPE
Max Char:	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NEL ID	element ID
2	NCARDS	number of cards in the second set
3	ITYPE	type of input in the second set. = 1 for pre-stresses = 2 for pre-strains

Card set 2: NCARDS cards, at least one card and should not be more than maximum number of nodes of the element.

Entry No.:	1	2	3	4	5	6	7
Variable:	NODE	SXX	SYY	SZZ	SXY	SYZ	SXZ
Max Char:	6	12	12	12	12	12	12

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NODE	nodal ID
2	SXX	pre-stress SXX or pre-strain EXX
3	SYY	pre-stress SYY or pre-strain EYY
4	SZZ	pre-stress SZZ or pre-strain EZZ
5	SXY	pre-stress SXY or pre-strain EXY
6	SYZ	pre-stress SYZ or pre-strain EYZ
7	SXZ	pre-stress SXZ or pre-strain EXZ

6.7.10 *FREQFUNCTION Data Group

Applicable analysis types: DFREQUENCY

This data group is always required. Both real valued and complex valued frequency functions may be defined using this data group.

Each frequency function is defined using three sets of cards. The first and the second sets contain one card each and are used to specify the title and control parameters, respectively. The third card set is required if the frequency function is not a constant value, and it may consist of one or more cards.

Group ID card:

*FREQFUNCTION

Card set 1: Frequency function title card required

Entry No.: 1
Variable:

TITLE

Max Char: 80

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	TITLE	alphanumeric description of frequency function.

Card set 2: Frequency function control card

\$

Entry No.: 1 2 3 4 5 6
Variable:

ID	NP	NTABLE	SFTR	DATA1	DATA2
----	----	--------	------	-------	-------

Max Char: 5 5 5 20 20 20

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	ID	Frequency function identification number to be referenced in *CFORCE, *PRESSURE, *SPDISP
2	NP	Number of points used to define the function (>2). Enter zero for constant frequency function (NTABLE = 0).

- 3 NTABLE table form to be used to input the forcing frequency function.
 = 0 – constant value frequency function is given, i.e., it has a constant value over the frequency range of interest (does not require card set 3).
 = 1 - frequency versus amplitude of real frequency function values.
 = 2 - frequency versus amplitude and phase values of complex frequency function.
 = 3 - frequency versus real and imaginary components of a complex frequency function

---tab(\$)---

- 4 SFTR scale factor used for the frequency function data (default = 1.0).
 5 DATA1 The amplitude value of constant frequency function.
 6 DATA2 The phase value of constant frequency function.

Card set 3: Frequency function definition cards

This card set is required for NTABLE = 1, 2 or 3. The abscissa (frequency) and the ordinates should be entered as one set per card. Enter only NP sets.

For NTABLE = 1

Entry No:	1	2
Variable:	FREQ	VAL
Max Char:	20	20

Entry	Variable	Description
1	FREQ	frequency in cycles/time at point 1
2	VAL	frequency function in unit of excitation at point 1

For NTABLE = 2

Entry No:	1	2	3
Variable:	FREQ	VAL1	VAL2
Max Char:	20	20	20

Model Data

Miscellaneous Data

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	FREQ	frequency in cycles/time at point 1
2	VAL1	amplitude of frequency function at point 1
3	VAL2	phase of frequency function at point 1 in unit of excitation

For NTABLE = 3

Entry No:	1	2	3
Variable:	FREQ	VAL1	VAL2
Max Char:	20	20	20

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	FREQ	frequency in cycles/time at point 1
2	VAL1	real component of frequency function at point 1 in unit of excitation
3	VAL2	imaginary component of frequency function at point 1 in unit of excitation

Notes:

1. The frequency values should be entered in ascending order. They must be positive, except for the first point which can be zero.
2. Any number of frequency functions may be defined by repeating card sets 1 to 3. Each frequency function should have a unique ID.

6.8 Surface Contact Data

6.8.1 *CSURFACES Data Group - Definition of Contact Surfaces

This data is required in nonlinear analysis whenever the model has contact surfaces.

Group ID card: *CSURFACES

The data for a typical contact surface (CS) consists of two sets of cards. The first set consists of a parameter card defining the surface parameters with a syntax similar to that used in the executive commands ([Section 5.3.1](#)), i.e., options are shown between square brackets and default values are shown between braces. The second set of cards gives the segments' information.

Parameter Card: One card, always required

$$\text{IDSURF} = n, \text{STYPE} = \begin{bmatrix} \text{POINTS} \\ \text{LINES} \\ \text{SHELLS} \end{bmatrix}, \text{NORDR} = \begin{bmatrix} 1 \\ 2 \\ 3 \end{bmatrix} \text{FORM} = \begin{bmatrix} \{0\} \\ 1 \\ 3 \end{bmatrix},$$

$$\text{REVERSE} = \begin{bmatrix} \{\text{NO}\} \\ \text{YES} \end{bmatrix}$$

where

- IDSURF : contact surface-id number, n is the id-number
- STYPE : surface type
- NORDR : indicates (in NISA convention) the segment order; with 1, 2, and 3 for linear, parabolic and cubic, respectively. This provides the *default* value for the segments given in card set 2.
- FORM : indicates the format used in card set 2 to define the segments' nodes and is applicable for POINTS, 2D LINES and SHELLS (3D surfaces).

Model Data

Surface Contact Data

REVERSE : indicates whether the normal orientation of the CS, as implied by the segment definition in card set 2, is to be reversed.

Segment card(s), FORM = 0

This is valid for any STYPE and defines the surface segment-by-segment.

	\$						
Entry No:	1	2	3	4	5	6	12
Variable:	ISEG	NORDR	NSET	NODINC	N1	N2	---- N8
Max char:	6	2	4	6	6	6	--- 6

Entry Variable Description

- 1 ISEG : Segment number, for user identification purposes only.
- 2 NORDR : Segment order, defaults to the value given on the parameter card.
- 3 NSET : Automatic generation parameters. NSET is the number of sets to be generated from the base set, including the base set.
- 4 NODINC : NODINC is the node number increment.
- tab (\$) -----
- 5-12 N1-N8 : The segment nodes consistent with NORDR value. If the segment has more than 8 nodes, continue on a subsequent card that begins with a tab (\$).

Segments Card(s) FORM = 1

This is valid for STYPE = LINES or POINTS, only when the segment order does not change on the entire CS.

Entry No:	1	2	3		10
Variable:	N1	N2	N3	----	N10
Max char:	6	6	6		6

Entry Variable Description

1, 2... N1, N2,... : Nodes of the CS as one moves from one end to another. Up to 10 nodes may be entered on a card, zero entries are ignored

Segments Card(s) FORM = 3

This is valid for STYPE = LINES or SHELLS (see ** Note 1, 2, 3 **)

Entry No:	1	2	3		10
Variable:	N1	N2	N3	----	N10
Max char:	6	6	6		6

Entry Variable Description

1, 2... N1, N2,... : Nodes of the CS can be in any order. Up to 10 nodes may be entered on a card. If there are more than 10 nodes, use more cards.

Notes:

1. At present, format FORM = 3 is available only for these element types: NKTP = 1, 2, 3, 4, 5, 7, 9, 13, 15, 18, 20, 22, 32, 33, 36, 40, 41.
2. For element types other than shell, such as NKTP = 1, 2, 3, 4, the REVERSE key is not used if the FORM = 3. However, if a contact surface is on shell elements, the REVERSE key is related to the local face number of a shell element. 'REVERSE = NO' indicates the contact surface is on the face 2 of a shell element, and 'REVERSE = YES' indicates the contact surface is on the face 1 of a shell element.

- When $FORM = 3$ is used, one should be careful about unexpected contact surfaces introduced by the set of nodes. For example, a contact surface between node 1 - 2 and 2 - 3 of a triangle element are expected. If one defined the contact surface by $FORM = 3$ as 1, 2, 3, then an unexpected segment 1 - 3 will be included as a segment of the contact surface. (See [Figure 6.17](#))

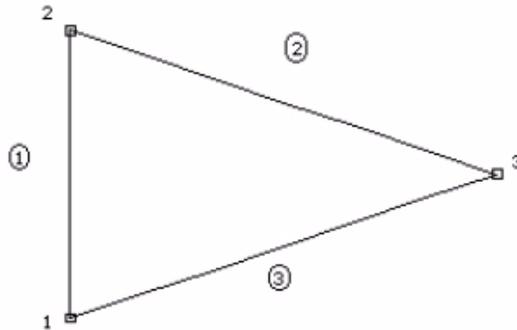


Figure 6.17: Use of $FORM = 3$ to define a contact surface on triangular element

- A Contact Surface (CS) may be a Slave Surface (SS) or a Master Surface (MS). It is made up of segments which usually are edges of 2-D elements or faces of 3-D solids or shells. Thus, depending on the application, a CS consists of LINES, SHELLS, or POINTS. The last type (POINTS) is actually a collection of points, with no implied connectivities, and therefore can only be SS. The line segment (LINES) may be straight or curved. The area segments (SHELLS) may be of quadrilateral or triangular shapes.
- A Contact Element (CE) combines 2 CSs, the first one is the SS and the second is the MS. The (SS, MS) in a CE should be compatible. Thus, in 2-D applications, a SS may be of the type POINTS or LINES, whereas a MS must be of the type LINES. In 3-D applications, the MS must be made of area segments, i.e. of the type SHELLS, whereas as SS may be of the type POINTS, LINES, or SHELLS. The above surface compatibility check, as well as many other data checking, are automatically performed by the program during the contact-data-processing phase.
- A CS may belong to more than one CE as either a SS or a MS. In particular, a typical SS may be associated with more than one MS in different CEs. However, at any given time a typical slave node in the SS may be in contact with only one MS.
- The impenetrability constraint is imposed on the slave nodes of an SS. However, the constraint is not imposed on the master nodes of the MS. Thus a single CE made up of SS and MS represents the usual single-pass analysis.

8. A symmetric-pass (two-pass) analysis may also be performed. This ensures that the nodes on the MS may not penetrate the SS. However, this approach is not without added cost, since the list of all slave nodes would include the nodes of the MS. The symmetric-pass treatment is not activated automatically for a single CE in NISA. However, it can be formed conveniently by the user by interchanging the SS and MS in a different CE (one line of data) (i.e., CE1 = (CS1, CS2) and CE2 = (CS2, CS1)). It should be noted that this procedure results in only some minor increase in the internal database associated with the surface and contact element definitions. However, the size of the result database will increase.
9. A typical slave node may belong to only one SS. However, it may be a master node on any other CS. But for a given CE, the SS and the MS should not be the same CS and the nodes on the SS should be different than those on the MS*.
10. Contact surfaces may be boundaries of deformable or rigid bodies. All nodes on all CSs must be attached to NISA elements. However, rigid surfaces may be modeled by using “dummy” elements. One way is to use as many point mass elements (with dummy properties) as needed to describe the geometry of the rigid surface. This will introduce nodes which may be used in the contact surfaces’ definition. A CE may not be composed of two rigid surfaces. A rigid surface may be an SS or an MS. However, it is recommended that a rigid surface is modeled as an MS. In this case the normal direction of the master surface would depend on the current orientation of the rigid surface, but not on the deformation of the model.
11. The definition of the normal direction to a CS is of utmost importance in contact analysis. The convention used in NISA is that the normal to the contact surface is the outward drawn normal. That is, the normal direction should be pointing towards the exterior of the body and not to its interior. An illustration is given in [Figure 6.18](#) where the correct normal (n_s) is used in SS of body S, and an incorrect normal (n_m) is used in MS of body M.

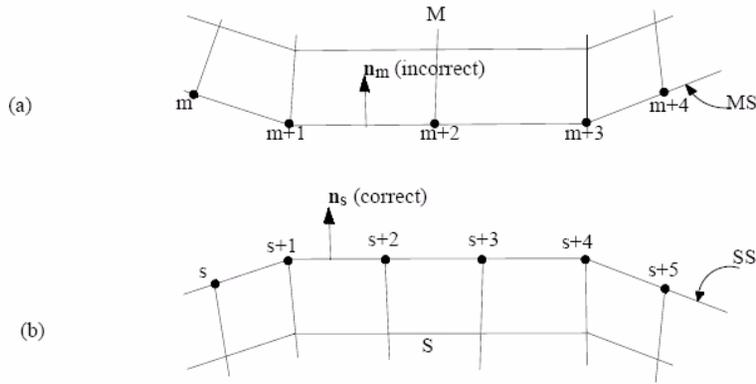


Figure 6.18: Definition of outer normal direction for contact surfaces, (a) Incorrect directions is used on MS, (b) Correct direction is used as SS.

12. The convention used for the normal definition does not apply for an SS of the STYPE = POINTS, since there are no connections implied by points (i.e., not segments). For this, and other reasons, it is recommended that this type of surfaces be used with care.
13. The outer normal to a CS is implied by the connections of its segment. Ordinarily, all segments on a CS should imply the same normal orientation (not necessarily the direction which may change). For 2-D contact, a rule of thumb is that as one moves along the CS, the outer normal is towards his left, and the interior of the continuum is towards his right. Because of the importance of the outer normal definition, a parameter may be given in the definition of a typical CS to internally reverse the normal direction of all its segments.
14. Initially, the two surfaces (SS, MS) may overlap each other at some points as in initial interference problems. The initial load implied by the initial interference may be the only load applied to the model, and other types of loading need not be applied.
15. Local displacement coordinate systems may be used at any node of a CS. Specified displacements (zero or nonzero) may be imposed on DOF's of nodes on a CS, irrespective of whether the CS is a boundary of a deformable or rigid body. It should also be noted that the contacted bodies should each be properly restrained against rigid body motion.
16. Degrees of freedom at nodes on a CS may appear in a coupled displacement set (*CPDISP, Chapter 6) as independent or dependent DOFs. However, they may not be dependent (slave) DOF's of MPC equations or rigid links (*MPCEQN and *RIGLINK).

17. Two solutions schemes are implemented for the wave front solution of the equilibrium equations. The first is an automatic sub structuring analysis. In the second scheme, the wave front parameters are dynamically updated according to the active set of nodes in the contact region.

Explanation of Output for 2-D Contact Problems:

For contact element, NISA prints out contact forces and tractions in two different tables. Each table contains several labels which are explained below.

(a) Contact Force Print Out

S node: Slave node id number

{XSI}c:

Normalized distance on a curve segment between -1 to 1 from the mid-curve. Let's illustrate {XSI}c with an example as given in [Figure 6.19](#). Distance between node M1 and node M3 on Master surface is normalized between -1 and 1. Mid-point between -1 and +1 is zero (0). Let's call it M2. {XSI}c is the normalized distance between M2 and contact point on Master surface. {XSI}c can be positive, zero or negative depending on the position of the contact point. If it is to the right side of M2 then it is positive. If it is to the left side of M2 then it is negative and if it coincides with M2 then it is zero. In [Figure 6.19](#) {XSI}c is a negative number.

SSid: Slave surface id number (see *CELEMENT data group)

MSid: Master surface id number (see *CELEMENT data group)

Mseg: Segment number (same as ISEG in *CSURFACE data group)

NORDR: Segment order (see *CSURFACE data group)

1st 2 Mnodes: Means 1st 2 master nodes on Mseg number

{n}c: Direction cosines for the normal direction at the contact point

{t}c: Direction cosines for the tangential direction at the contact point

{X}s: Global coordinate of the contact point

{Fn,Ft}s: normal and tangential contact force at contact point identified by Snode

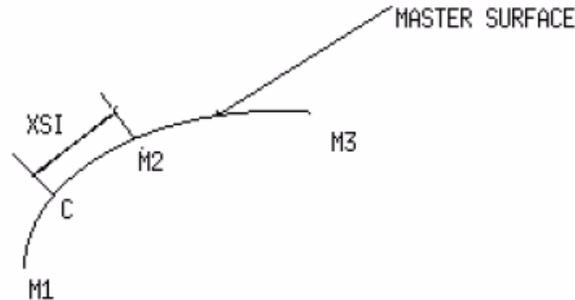


Figure 6.19: {XSI}c Illustration. c is the contact point on a curvilinear coordinate system on a generic contact segment.

(b) Nodal Transaction for Surface to Surface Contact

CSid: Contact surface id number (same as IDSURF in *CSURFACE data group)

Node: User's node id number

{X}: Deformed nodal coordinate for Node. Deformed means original coordinates plus associated displacement

{n}: Direction cosines for the normal direction at the node point

{t}: Direction cosines for the tangential direction at the node point

{Qx,Qy}: Resultant forces at node with respect to global X and Y coordinate system

{Pn,Pt}: Traction (pressure) forces at the Node in normal and tangential directions given by {n} and {t}, respectively

Explanation of Output for 3-D Contact Problems:

For three dimensional contact problems NISA prints out contact forces and tractions in two different tables. The output is similar to 2-D contact and the labels used in the tables are explained below.

(a) Contact Force Print Out

Snode: Slave node id number

{XSI}c:

Normalized coordinates on a face of the element identified by 3 corner nodes “Mnodes”. The 3 Mnodes form a curved area segment between -1 to 1 from the midpoint/centroid of the face. The face itself lies on the master surface MSid. {XSI}c is illustrated with an example as given in [Figure 6.20](#). The coordinates of the four nodal points M1, M2, M3 and M4 of the face which lies on the master surface are normalised to {1,1}, {1,-1}, {-1,-1}, and {-1,1} as shown. The origin of the normalised coordinates is at the centroidal point w.r.t. to the face as shown by point M0. {XSI}c specifies the normalized coordinate of the contact point on the face.

SSid: Slave surface id number (see *CELEMENT data group).

MSid: Master surface id number (see *CELEMENT data group).

Mseg: Segment number (same as ISEG in *CSURFACE data group).

NORDR: Segment order (see *CSURFACE data group).

Corner Mnodes: Means 1st 3 nodes on a face encompassed by Master surface.

{n}c: Direction cosines for the normal direction at contact point with respect to the global coordinate frame.

{t}c: Direction cosines for the tangential direction at the contact point with respect to the global coordinate frame.

{X}s: Global coordinate of the contact point.

{Fn,Ft}s: normal and tangential components of the contact force at contact point identified by Snode (see above). The unit vectors on the tangent plane at the contact point are output after {Fn, Ft}.

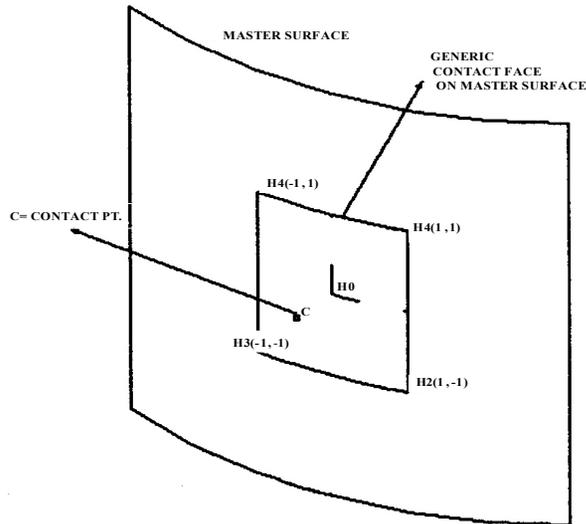


Figure 6.20: $\{XSI\}_c$ Illustration. c is the contact point on a generic contact face in a curvilinear coordinate system.

(b) Nodal Traction for Surface to Surface Contact

CSid: Contact surface id number (same as IDSURF in *CSURFACE data group).

Node: User's node id number .

$\{X\}$: Deformed nodal coordinate for Node. Deformed means original coordinates plus associated displacement .

$\{n\}$: Direction cosines for the normal direction at the node point with respect to global coordinate frame.

$\{t1\}$ and $\{t2\}$: Direction cosines for the tangential direction at the node point with respect to global coordinate frame.

$\{Q\}$: Traction (pressure) at node with respect to global coordinate system.

$\{P\}$: Traction (pressure) at the node in normal and tangential directions given by $\{n\}$ and $\{t\}$ respectively.

6.8.2 *CELEMENTS Data Group - Definition of Contact Elements

This data group is required in nonlinear analysis whenever the model has contact surfaces. The surfaces defined in the *CSURFACES are linked in pairs in this group, each pair forms a contact element (CE).

Group ID card:

*CELEMENTS

Contact elements card set:

Entry No:	1	2	3	4	5
Variable:	CEid	SSid	MSid	RCid	MATid
Max char:	6	6	6	6	6

Each CE is defined by a single line of data.

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	CEid	Identification number for the contact element (CE)
2	SSid	Identification number of a contact surface (CS) defined in the *CSURFACES and is designated as a slave surface in this CE
3	MSid	Identification number of a CS defined in *CSURFACES and is designated as a master surface in this CE
4	RCid	Identification number of a real constant table (defined in *RCTABLE) for this CE.
5	MATid	Identification number of a material table (defined in *MATERIAL) for this CE. The material table should define the coefficient of friction using the label NUXY. Set MATid to zero if frictionless contact is to be considered.

Model Data

Surface Contact Data

Real Constant Table (RCTABLE) for Surface contact

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	w_n	Penalty number for the normal direction, see note 7
2	w_t	Penalty number for the tangential direction
3	r_p	Pin ball radius, see note 8.
4	TOLxsi	Tolerance for end-segment's extension (default = 0.01), see note 1
5	THK	Effective beam/shell thickness used to offset the gap function, see note 2
6	TOLg	Tolerance on gap function for <i>initial</i> contact status determination (default = 10^{-8}) see note 3
7	TOLorn	Tolerance on validation of compatible contact in terms of orientation of both contact surfaces, (default = 0.02), see note 4
8	PTHK	Scale factor to adjust printout for contact pressure, see note 6

Note:

1. TOLxsi is used to extend an end-segment by a small amount, such that the limit on the isoparametric coordinate ξ is:

$$|\xi| \leq 1 + TOLxsi$$

This is useful for some special cases in modeling contact problems wherein contact may occur at the edges of a beginning- or and end-segment of a contact surface (e.g., at an axis of symmetry, see [Figure 6.21](#)). Note that this applies to end segments only. Intermediate segments are handled automatically by the program.

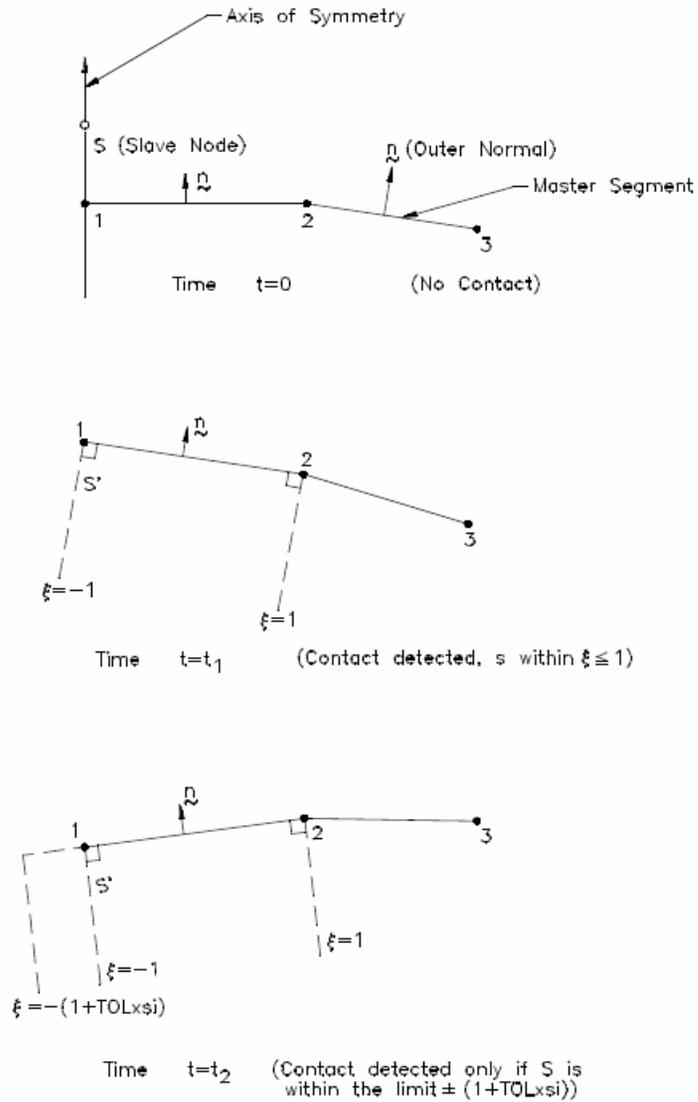


Figure 6.21: Explanation of TOLxsi

- Contact is detected when the gap function is negative

$$g_n = (\mathbf{x}_s - \mathbf{x}_c) \cdot \mathbf{n} \leq 0 \quad (\text{contact})$$

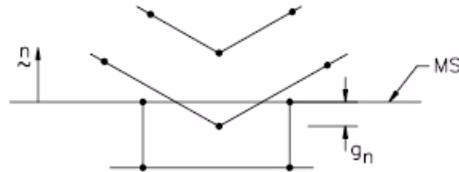


Figure 6.22: Modeling contact between continuum elements

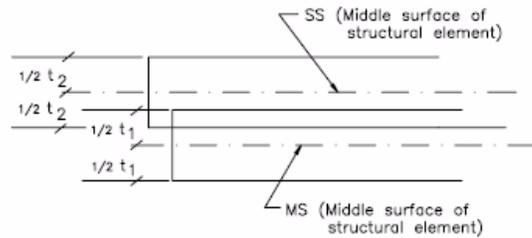


Figure 6.23: Modeling contact between structural (beams/shells) elements $THK = \frac{t_1 + t_2}{2}$

While this is readily applicable when modeling contact between continuum elements as shown in Figure 6.22, it needs some adjustment when modeling contact between structural elements *if their thicknesses are to be taken into account*. Since beams and shells are modeled by their middle surfaces, *physical contact* between two shells (or a beam and a continuum element) may not be detected by the contact algorithm as shown in Figure 6.23. In this case, the gap function may be altered such that it is defined as,

$$g_n = (\mathbf{x}_s - \mathbf{x}_c) \cdot \mathbf{n} - THK \quad (\text{Optional definition of gap function})$$

- A slave node is considered to be in contact *initially* if it lies on or below (as in initial interference fit problems) a master contact surface. Since the nodal coordinates are entered with a finite precision, the computed gap function may not be exactly zero or less than zero *initially* although the user may want it to be so. Therefore, for the initial search only (time 0), contact is assumed to have established if $g_n \leq TOLg$ (initial search only).

4. Contact is considered to be compatible only if

$$\mathbf{n}_s \cdot \mathbf{n}_m < 0 \text{ (for admissible contact)}$$

where, \mathbf{n}_s is the outer normal to the slave surface at the slave node and \mathbf{n}_m is the outer normal to the master surface at the contact point. In the program, the above condition is implemented such that

$$\mathbf{n}_s \cdot \mathbf{n}_m < TOLorn \text{ (for admissible contact)}$$

Note that this check is only possible when the two surfaces are of the same type (e.g., “LINE”). This does not apply to a contact surface of the type “POINT”. It should be noted that this check may be practically suppressed when TOLorn is given as a large number.

5. For the controlled release logic, a slave node already in contact is allowed to release after considering the status of its neighbors. This logic does not apply to surfaces of the type “POINTS”.
6. The contact tractions are computed from the contact forces using a lumped formulation. For axisymmetric problems (those involving NKTP = 3), no adjustment is required and PTHK can be left zero (reset to 1 internally). However, for plane stress problems, the contact tractions will be in terms of force/unit length if PTHK is 1. To get the contact tractions in units of force/length², PTHK should be entered as a representative thickness of the plane stress elements whose edges are the segments of the contact surfaces.
7. Selection of a penalty number is problem dependent, a very large penalty number may cause numerical instability such as oscillation and divergence. Guidelines for choosing a penalty number for penalty method and augmented Lagrangian multiplier method are as follows:

In penalty method, a penalty number controls the amount of penetration of the two contact surfaces at equilibrium. The contact force is the amount of penetration times the penalty number. Physically, a penalty number can be considered as a spring constant between two contact surfaces after contact. The accuracy of analysis depends on the magnitude of the penalty number. A small penalty number means deeper penetration and a less accurate result. A number about 1 to 3 order of magnitude higher than a representative stiffness (or Young’s modulus) of the contacting bodies in the contact region is recommended.

In augmented Lagrangian multiplier method, the product of penetration and a penalty number defines the incremental contact force. The contact forces are updated incrementally until the system is balanced. A small penalty number results in smaller incremental contact forces and more iterations to reach equilibrium. A penalty number about -1 to 1 order of magnitude of a representative stiffness in the contact region is suggested.

8. User can specify small regions around the slave nodes with a pinball radius parameter so that the search process is limited to only this region and the whole analysis process becomes faster.

6.9 Moving Load Data

6.9.1 *MLOAD Data Group - Moving Load Definition

Application analysis type: STATIC

This data group is optional and may be used to specify the loads moving along a moving load path.

Each moving load is defined by two sets of data cards. The first set contains one card to specify control parameters such as moving load path ID and starting node etc. The second card set consists of one or more cards to define the locations and magnitude of moving loads.

Group ID card: ***MLOAD**

Moving load definition card set 1: .

Entry No:	1	2	3	4	5	6	7	8
Variable:	IDMVLD	IDPATH	LABEL	NF	NTABLE	NSTART	NEND	NEND
Max char:	6	6	4	5	2	6(8)	6(8)	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IDMVLD	moving load identification number
2	IDPATH	moving load path identification number defined in *MPATH data group
3	LABEL	label to specify a component of moving force or moment. FX, FY, FZ - concentrated or distributed forces in global X, Y and Z axes, respectively MX, MY, MZ - concentrated or distributed moments in global X, Y, and Z axes, respectively
4	NF	number of forces or moments moving together in the moving load set.
5	NTABLE	selection of table format for concentrated load and distributed loads = 0 use concentrated load format = 1 use distributed load format

Model Data

Moving Load Data

6	NSTART	starting node ID for the moving load set
7	NEND	ending node ID for the moving load set
8	INC	number of node shift in each move

Card set 2: Moving load definition cards

For concentrated load (NTABLE = 0)

This format is used to define concentrated loads. The leading position of the moving load set starts moving at node ID NSTART and ends at node ID NEND. Each load in the load set should have two values. The first value specifies the distance between the load and the leading position. The second value defines the magnitude of the load. The distance should always be positive or all loads should not be in front of the leading position.

Entry No:	1	2	3	4	5	6
Variable:	P1	F1	P2	F2	P3	F3
Max Char:	10	10	10	10	10	10

Entry Variable Description

1	P1	distance between the first load and the leading position
2	F1	magnitude of first load
3	P2	distance between the second load and the leading position
4	F2	magnitude of second load
5	P3	distance between the third load and the leading position
6	F3	magnitude of the third load

For distributed load NTABLE = 1

This table format is used to define distributed moving loads. All distributed loads are assumed to follow linear distribution. The magnitude should be given in force per unit length. The distance from the leading position should always be positive or the distributed load should not be in front of the leading position of the moving load set.

Entry No.:	1	2	3	4
Variable:	P1	L1	F1	F2
Max Char:	10	10	10	10

Entry Variable Description

1	P1	distance between the beginning of the distributed load and the leading load
2	L1	length of the distributed load
3	F1	magnitude of distributed load at beginning point (force/length)
4	F2	magnitude of distributed load at end point

Notes:

1. The leading position of the moving load moves from node to node along the moving load path. It moves from the starting node NSTART to the end node NEND. INC value determines the moving pattern of how many nodes it jumps in each move. The default value of INC is to move one node at each move.
2. Any load outside of the moving path is inactive even though the leading position already is in the moving path. Once the leading position moves out of the end node (NEND), all loads in the same set of moving load are inactive.
3. All moving load sets are moving at the same time in each move regardless of the moving pattern. For example, two moving load sets start at the same node but with different moving pattern (different INC). After one move, the leading position of these two load sets locates at different locations.
4. When *MLOAD data is given in the analysis data, NISA will generate new load cases starting from the last load case given in the NISA input data. If no load is given in the last load case in the input data, only moving loads will be considered in the new load cases. Loads given in the last load case will be superimposed with the moving loads to generate new load cases.

6.9.2 *MPATH Data Group - Moving Load Path Definition

Applicable to: STATIC

This data group defines moving load path in a model to be referred in the *MLOAD (moving load) data group. The moving load path is a set of nodes which can form a continuous line along the edge of the elements (note 1). Two sets of cards are required to define this data group. The first set is the control parameters and the second set defines the nodes along the path. Nodes should be given in the order of moving load direction.

Group ID card:

*MPATH

Moving load path card set 1:

Entry No.:	1	2	3
Variable:	IDPATH	ISHAPE	IREVRS
Max Char:	6	2	2

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IDPATH	path identification number(0 < integer < 999999)
2	ISHAPE	selection of shape functions = 0 isoparametric shape function (default) = 1 beam shape functions (note 3)
3	IREVRS	reverse the direction of the moving load path as defined = 0 keep the direction of the moving load path as defined = 1 reverse the direction of the moving load path as defined

Moving load path card set 2:

Entry No.:	1	2	3	4	5	6	7	8	9	10
Variable:	LABEL	I1	J1	K1	I2	J2	K2	I3	J3	K3
Max Char:	1	6(8)	6(8)	6(8)	6(8)	6(8)	6(8)	6(8)	6(8)	6(8)

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	LABEL	identification label (to interpret the remaining entries on this card). Allowable labels are: S = entries on this card are single numbers R = entries on this card define range of numbers E = entries on this card are single numbers to be excluded from any ranges of numbers defined in this path.
2-10	I1,J1,K1...K3	interpretation of these entries depend on the identification LABEL such that I1, J1, K1,...K3 are single node numbers, when LABEL is 'S' (I1, J1, K1), (I2, J2, K2), (I3, J3, K3) define three ranges of numbers beginning with I _i , ending with J _i and in increment of K _i , for i = 1, 2, 3, respectively, when LABEL is set to 'R' I1, J1, K1,...K3 are single numbers to be excluded from any ranges of numbers defined in this path, when LABEL is set to 'E'

Notes:

1. The path of the moving load should follow along the element edges to form a continuous line to allow loads to move along. Any nodes present along the path should be specified in this card. The order of the nodes defined in the path defines the direction of the moving.
2. LABEL (S, R, E) must be given for the first card of card set 2 used. If a path requires more than one card, additional cards can be given in the form of starting with a LABEL or a tab character \$. Starting with a tab character indicates the label of the previous card is used.
3. When ISHAPE = 1 is used, shape functions of NKTP = 12 will be used to calculate equivalent load for that segment and the moving load is assumed acting on NKTP = 12 element whenever an element type of NKTP = 12 exists in the segment. If no NKTP = 12 is present in the segment, the default isoparametric shape functions will be used.

Model Data

Moving Load Data

Analysis Data

The analysis data for static, eigenvalue, buckling, nonlinear static, linear, and nonlinear direct transient dynamic, and heat transfer analyses are presented in this chapter. The modal dynamic analysis data (transient dynamics, frequency response, random vibration, and shock spectrum) are presented in [Chapter 8](#). The analysis data generally defines the constraints and the loads on the model and selects the output options. It must follow the model data block in a typical NISA data deck as shown in [Figure 5.1](#).

The analysis data block consists of distinct data groups. Each data group consists of a group identification card followed by its free format data, which may consist of one or more card sets, each card set may consist of one or more cards. The free format rules are given in [Section 5.2](#). Each group has a descriptive identification name indicating the function of the data group. For example, the specified displacement data is given in the *SPDISP data group, and the concentrated nodal force data is given in the *CFORCE data group.

The detailed description of the analysis data groups is presented in this chapter in the sequence shown in [Table 7.1](#). This sequence is adopted according to the function of each data group. For easy reference an alphabetical list of *all* data groups available in NISA is given in [Table 5.4](#). The input data setup for the above analysis types is discussed in [Section 5.4](#).

The following rules must be observed in the preparation of the analysis data:

1. For each analysis type, a specific data group flags the beginning of the analysis data block. This includes:

- (a) The *LDCASE data group for static and buckling analysis. For static analysis, each load case begins with the *LDCASE data groups, followed by the appropriate data groups defining the constraints, loadings and output options.
 - (b) The *EIGCNTL data group for eigenvalue analysis.
 - (c) The *EVENT data group for nonlinear static and direct transient analyses. After the constraints, loading and output options have been defined using the appropriate data groups, another *EVENT data group will begin the definition of the second event, if any, and so on. Currently, the specified displacement data cannot be changed from one event to another.
 - (d) The *HEATCNTL for steady state and transient heat transfer analysis.
2. For static analysis, after all load cases have been entered, the results may be combined through the load combination data. Each load combination case begins with the *LDCOMB data group, which may be followed only by the load combination title (*LCTITLE) and the appropriate data groups selecting the output options (e.g., *PRINTCNTL).
 3. Buckling analysis is a *one-run*, two-pass procedure. The first pass is a static analysis which computes the stresses for the specified loading conditions. The second pass is an eigenvalue analysis which determines the buckling load factors and the mode shapes. As such, the analysis data for buckling consists of a static load case (which begins with the *LDCASE group followed by the appropriate data groups defining the constraints, loading and output options) followed immediately by the eigenvalue analysis data (which begins with the *EIGCNTL data group). At present, only one loading condition can be specified for buckling analysis.
 4. The analysis data for a particular problem need not use all of the data groups described in this chapter. Some data groups are optional for particular analysis types. Whether the data group is required or not is explicitly stated in the description of the data group, along with the applicable analysis types.
 5. The data groups may be arranged in the analysis data quite arbitrarily but with few exceptions:
 - (a) As mentioned above in item 1, some data groups serve as delimiters between the model data block and the analysis data block, or between a static load case and another, e.g., the *LDCASE data group.

- (b) Eigenvalue analysis uses output blocks for controlling the group and continues with other pertinent data groups defining the printout options (e.g., the *PRINTCNTL data group).

Other than the above exceptions, the arrangement of the data groups is arbitrary. However, the sequence given in this manual is recommended for easier recognition.

6. The analysis data block must be followed by the data deck terminator, the *ENDDATA group ID card.

Table 7.1: List of Analysis Data Groups

Section No. and Group ID ⁽¹⁾	Description	Applicable Analysis Types ⁽²⁾
7.1 <i>Analysis Control Data</i>		
7.1.1 *LDCASE	Static load case control	ST, BU
7.1.2 *EIGCNTL	Eigenvalue analysis control	EV, BU
7.1.3 *EVENT	Nonlinear static event control	NL, LT, NT
7.1.4 *FREQCNTL	Direct frequency analysis control	DF
7.1.5 *HEATCNTL	Heat transfer analysis control	SH, TH
7.1.6 *LDCOMB	Load combination	ST
7.1.7 *LCITITLE	Load case title	ST, BU, NL, LT, NT
7.1.8 *MODEOUT	Mode selection for output	EV, BU
7.1.9 *TIMEINTEG	Time integration parameters	TH
7.1.10 *STEPSIZE	User specified time steps	NL, TH, NT
7.1.11 *CONVERGENCE	Convergence criteria	NL
7.2 <i>Fourier Analysis Data</i>		
7.2.1 *FRCNTL	Fourier analysis control	ST
7.2.2 *FRCOEF	Fourier coefficients	ST
7.2.3 *ANGSEC	Angular section for response	ST

Section No. and Group ID⁽¹⁾	Description	Applicable Analysis Types⁽²⁾
<i>7.3</i> <i>Boundary Conditions and Loading</i>		
7.3.1 *SPDISP	Specified displacements	ST, EV, BU, NL, LT, NT
7.3.2 *CFORCE	Concentrated nodal forces	ST, BU, NL, LT, NT
7.3.3 *CFOLLOWER	Follower concentrated forces	NL, NT
7.3.4 *PRESSURE	Pressure loads	ST, BU, NL, LT, NT
7.3.5 *BEAMLOAD	Beam loads	ST, BU, NL, LT, NT
7.3.6 *BODYFORCE	Body forces	ST, BU, NL
7.3.7 *NDTEMPER	Nodal temperatures	ST, BU, NL
7.3.8 *NDTEMPDIF	Nodal temperature differences	ST, BU, NL
7.3.9 *INITIAL	Initial condition	LT, NT
7.3.10 *TBODYFORCE	Dynamic body forces	LT, NT
7.3.11 *L1	Alternate form for pressure loads	ST, BU
7.3.12 *MASTER	Master degrees of freedom for Guyan reduction	EV, BU
7.3.13 *EDGFORCE	Edge force for shell element	ST, BU, NL, LT, NT
7.3.14 *ECDENSITY	Material properties for piezoelectric analyses	PST, PLT

Section No. and Group ID ⁽¹⁾	Description	Applicable Analysis Types ⁽²⁾
7.4	<i>Boundary Conditions and Loading for Heat Transfer</i>	
7.4.1	*INITEMP	TH
7.4.2	*SPTEMP	SH, TH
7.4.3	*CFLUX	SH, TH
7.4.4	*DFLUX	SH, TH
7.4.5	*ELHEATGEN	SH, TH
7.4.6	*NDHEATGEN	SH, TH
7.4.7	*CONVBC	SH, TH
7.4.8	*RADBC	SH, TH
7.4.9	*RADSURFACE	SH, TH
7.5	<i>Output Control Data</i>	
7.5.1	*EIGOUT	EV, BU
7.5.2	*NLOUT	NL, LT, NT
7.5.3	*PRINTCNTL	ALL
7.5.4	*REGIONS	ST, EV, BU, NL, NT
7.5.5	*STRSFILTER	ST, EV, BU, NL, NT
7.5.6	*SFDCOMP	ST, EV, BU
7.5.7	*TEMPHISTORY	TH
7.5.8	*TEMPOUT	TH
7.5.9	*HISTOUT	LT, NT
7.5.10	*I5	ST, EV, BU, NL
7.5.11	*N5	ST, EV, BU, NL
7.5.12	*POSTCNTL	ST, BU

Analysis Data

Section No. and Group ID ⁽¹⁾	Description	Applicable Analysis Types ⁽²⁾
7.6 Data Deck Terminator 7.6.1 *ENDDATA	Input data terminator	ALL

(1) **Acceptable minimum abbreviations are in bold face**

- (2) ST: Linear static EV: Eigenvalue BU: Buckling
LT: Linear direct transient NL: Nonlinear static SH: Steady state heat transfer
TH: Transient heat transfer NT: Nonlinear direct transient PST: Piezo electric linear static
PLT: Piezo electric linear direct
transient

ALL: All analyses (ST, EV, BU, NL, LT, NT, SH, TH)

7.1 Analysis Control Data

7.1.1 *LDCASE Data Group - Static Load Case Control

Applicable analysis types: STATIC, BUCKLING

This data group is always required in static and buckling analyses. It defines some computation keys needed in the analysis, and it must be the first data group in the analysis data set. Only one load case is allowed in buckling analysis. For static analysis, there is no limit to the number of load cases.

Each load case must start with this data group, followed by all other pertinent data groups which define boundary conditions, loading and printout control for this load case.

Group ID card: ***LDCASE, ID = n**

where, n is a *unique* load case identification number (up to 6 digits integer number). This load case ID number may be referenced by other data groups, e.g., the *LDCOMB and *PRINTCNTL data groups. If the load case ID number is not specified, it will default to the preceding load case ID number plus 1.

Load case control card set: one card

	\$								
Entry No:	1	2	3	4	5	6	7	8	9
Variable:	KELFR	KRCTN	KSTR	KSTN	LQ1	LQ2	LQ7	TSFRE	TOL
Max char	2	2	2	2	2	2	2	8	8

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	KELFR	element strain energy, element internal forces and rigid element forces calculation key (see note 1). The element internal forces are always provided in the global cartesian system, except for nodes which have local displacement coordinate system. = 0 - off = 1 - on

Analysis Data

Analysis Control Data

- 2 KRCTN reaction force calculation key (see notes 1, 2)
 = 0 - off
 = 1 - on
- 3 KSTR element stress calculation key (see notes 1, 3, 4)
 = 0 - no element stress calculation
 = 1 - calculate stresses at element Gauss points
 = 2 - calculate stresses at element node points
 = 3 - calculate stresses at Gauss points and node points
 = 4 - calculate the stresses at element centroid only
 = 5 - calculate stresses at element Gauss points and centroid
 = 6 - calculate stresses at element node points and centroid
 = 7 - calculate stresses at Gauss points, node points and centroid
- 4 KSTN element centroidal strain calculation key
 = 0 - off
 = 1 - on
- tab (\$)--
- 5 LQ1 element stress printout key (used only if KSTR > 0). This key selects the
 type of element stress printout.
 = -1 - suppress all element stresses printout
 = 0 - print all element stresses (local, global principal, and equivalent
 stresses)
 = 1 - print global components of stresses for each element
 = 2 - print principal stresses and equivalent stresses (e.g., von Mises
 stress) for each element
 = 3 - print element local stresses only (for shell elements or orthotropic
 materials) for each element
 = 4 - print global stresses, principal stresses, and equivalent stresses for
 each element
 = 5 - print global and local stresses for each element
 = 6 - print local stresses, principal stresses, and equivalent stresses for
 each element

- 6 LQ2 averaged nodal stress printout key, used only if KSTR >0 (see notes 1, 5, 6)
= 0 - do not print averaged nodal stresses
= 1 - print global stresses, principal stresses and equivalent stresses at the nodes.
= 2 - print global stresses at the nodes.
= 3 - print principal and equivalent stresses at the nodes
= 4 - calculate all nodal stresses and retain them on a file for post-processing, but do not print them
- 7 LQ7 coordinate system key for displacement printout (see notes 1, 7)
= 0 - print local components of displacement at nodes where local displacement coordinate systems have been defined. In addition, provide another printout listing global cartesian components of displacements at all nodes.
= 1 - print global cartesian components of displacements at all nodes.
= 2 - print global cartesian components of displacements at all nodes, but at nodes where local displacement coordinate systems have been defined, provide the local components only
- 8 TSFRE initial (stress-free) temperature
- 9 TOL tolerance for node point force unbalance printout (used only if KELFR > 0)
= 0.0- printout of node point force unbalance will be suppressed
> 0.0- printout of node point force unbalance will be given only at those nodes for which the absolute value of any components of the unbalanced forces is greater than TOL.

Notes:

1. The keys in this data group are used to initiate the computation of the output quantities, and to indicate the type of output request (e.g., global stresses or local stresses). The actual printout (excluding the nodal force unbalance printout), however, is controlled by the *PRINCNTL data group, which can also be used to provide a selective printout at a subset of nodes, elements, etc. Refer to the *PRINCNTL data group for available options and defaults.
2. Reaction forces at nodes with a local displacement coordinate system are provided in that system. The summation of reaction forces is always in the global cartesian coordinate system. Reaction forces printout at all constrained nodes is always provided when

Analysis Data

Analysis Control Data

- KRCTN = 1, unless the default output request is overridden in the *PRINTCNTL data group.
3. The variable KSTR selects the locations for stress calculations. Gauss points are located in the interior of the elements, and are the locations where the stresses are most accurate. Node points are located on the element boundaries. Nodal stresses obtained on the element level are in general discontinuous across element boundaries. Use the variable LQ2 (entry no. 6) if averaged nodal stresses are desired.
 4. For line elements (beams, springs), any nonzero value of KSTR can be used. Stress resultants in the element local coordinate system are provided for these elements, e.g., normal force, shear forces, and bending moments.
 5. The averaged nodal stress printout at all nodes is always provided when $KSTR > 0$ and $LQ2 > 0$, unless the default output request is overridden in the *PRINTCNTL data group.
 6. To obtain an averaged nodal stress printout for nodes lying within certain regions of the model, use the *REGIONS data group. To obtain a filtered printout of nodal principal stresses and equivalent stresses within specified limits, use the *STRSFILTER data group.
 7. The displacement printout at all nodes is always provided, unless the default output request is overridden in the *PRINTCNTL data group.

7.1.2 *EIGCNTL Data Group - Eigenvalue Analysis Control

Applicable analysis types: EIGENVALUE, BUCKLING

This data group is always required in eigenvalue and buckling analyses, and it specifies control parameters needed for eigenvalue extraction.

Group ID card: ***EIGCNTL**

Eigenvalue control card set: one card

Entry No:	1	2	3	4	5	6	7	8
Variable:	NNF	NEGK	MAXIT	NPA	COEFLO	COEFUP	RTOL	SHIFT1
Max char:	5	5	5	5	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NNF	- number of eigenvalues/eigenvectors to be calculated (see note 1)
2	NEGK	eigenvector printout key (see note 2) = -1- do not print eigenvectors = 0- print all eigenvectors (default) = n- print first n eigenvectors
3	MAXIT	- Block-Lanczos method: the maximum number of steps (see note 5) - Subspace or inverse iteration method: the maximum number of iterations (default = 20) - Guyan reduction method: not used - Shifted block Lanczos method: number of iteration for shift selection
4	NPA	- Lanczos method: block size of Block-Lanczos method (see note 5) - Subspace iteration methods: dimension of subspace (see note 3) - Inverse iteration method: the maximum number of iterations allowed for an out bound eigenvalue (see note 7) - Guyan reduction method: the number of master degrees of freedom (see note 6) - Enter zero for other methods

3. The default value for the dimension of the subspace (NPA) is computed from:

NPA = min (NEV+8, 2*NEV) for conventional subspace iteration

NPA = min (NEV+8, 2*NEV, MAXPA) for accelerated subspace iteration

where NEV is the number of eigenvalues, and MAXPA is the maximum wavefront.

4. For eigenvalue analysis the initial shift (SHIFT1) should be entered in cycles/time. The initial shift may be used for accelerating convergence of the lowest eigenvalues for the subspace iteration method by specifying a shift value close to but less than the lowest eigenvalue. In case of a free-free (unconstrained) structure a small negative initial shift should be specified. If the initial shift is not specified, it is automatically introduced whenever a singularity is detected.

Block-Lanczos algorithm allows only a negative shift for vibration analysis. It is used for solving free-free structure problems. If a very small value of $-1.0E-20 \leq \text{SHIFT1} < 0$ is specified a program computed negative initial shift will be used instead.

For buckling problems, Block-Lanczos algorithm ignores non-negative shift. If $\text{COEF0} = 0$ and $\text{SHIFT1} \geq -1.0E-20$, an internally computed shift will be used to avoid missing the first positive eigenvalue (buckling load factor). For given $\text{SHIFT1} < -1.0E-20$ and $\text{COEF0} = 0$, SHIFT1 value will be used as an initial shift.

5. In Block-Lanczos method, the maximum number of steps MAXIT is the number of block steps of Lanczos algorithm. One block step is in operation to process NPA lanczos vectors in one block. The larger the block size the lesser is the number of block steps required to extract the same number of eigenvalues because more Lanczos vectors are involved in one block step. However, in practice NPA should not be too big. The default value is 3. This is a recommended value. For special situations such as multiple eigenvalues are expected, $\text{NPA} > 3$ can be specified to avoid missing some repeated eigenvalues in Lanczos eigenvalue extraction. Because a large NPA may cause instability in Lanczos algorithm, $\text{NPA} > 10$ is not recommended.
6. In Guyan reduction, the master degrees of freedom specified in *EIGCNTL data is used as follows. If the master degrees of freedom selected by the user in *MASTER is less than the value given as NPA then the remaining masters are chosen automatically by the program. Otherwise, all the user specified master degrees of freedom are used.
7. In inverse iteration, the estimated eigenvalue may be out of bound at the first few iterations. The NPA is the allowable iterations for an estimated eigenvalue which remains out of bound. The default value of NPA is five iterations

7.1.3 *EVENT Data Group - Event Control

Applicable analysis type: NLSTATIC, LTRANSIENT, NLTRANSIENT

This data group is always required in nonlinear static and direct transient dynamic analyses and it must be the first data group in the analysis data set. The group defines parameters for time period, step size, number of steps, equilibrium checks and other control parameters for an event.

Generally an event should be defined when new load types are introduced or when an existing load changes its sign. An event can have one or more steps (or increments). Step size may be determined automatically or via user definition.

A specific type of load (e.g., pressure, point load, etc.) specified in an event remains in effect for the subsequent events unless it is redefined by its pertinent data group. It should be noted, however, that the magnitude of the load is controlled by the corresponding time-amplitude curve. To illustrate, consider that a *CFORCE data group is first introduced in an event to define point loads. Now, consider the following two cases in a subsequent event:

- ❑ The *CFORCE is not provided: In this case, the previously defined point loads will remain in effect with their magnitude controlled by the reference time-amplitude curve.
- ❑ The *CFORCE is redefined: In this case, the previously defined point loads will be deactivated and overwritten by the new definition.

Each event must start with this data group, followed by all other pertinent data groups that fully define the event.

Group ID card:

*EVENT, ID = n

where, n is a unique event identification number (up to 6 digits integer number). If the event ID number is not specified, it will default to the proceeding event ID number plus 1.

Definition of required event control parameters immediately follows the group ID card and has the following format:

Parameter = data1, data2, data3

Each parameter may be abbreviated to a minimum of first four unique letters. Most of the parameters have default values which will be applied *only* when the parameter is omitted. Whenever a parameter is specified, the user should specify its required option(s). The available options for a typical parameter are shown between two square brackets, [] whereas the default option is shown between two braces, { }. The description of the event control parameters is given below.

INCREMENTS - Mode and number of steps

INCREMENTS = $\left[\begin{array}{l} \text{EQUAL, n} \\ \text{USER, n} \\ \text{AUTO, n} \\ \text{ARCL, n} \\ \text{ADAPTIVE, n} \\ \{\text{EQUAL, 10}\} \end{array} \right]$

where,

- EQUAL, n : equal n steps. The step size will be equal to the time of the event divided by n. This option is available only for LTRANSIENT analysis (see notes 1, 2 and 5).
- USER, n : user specified n steps. User must give data for step sizes in *STEPsize data group (see notes 1 and 2).
- AUTO, n : automatically selected step sizes. Maximum of steps will not exceed n (see notes 1-3).
- ARCL, n : arc length automatic stepping scheme. Maximum of steps will not exceed n. Not available for NLTRANSIENT analysis (see note 7).
- ADAPTIVE, n : Adaptive calculation of step sizes. Maximum of steps will not exceed n.

TIMEATEND - Time at end of the event

$$\text{TIMEATEND} = \begin{bmatrix} t \\ \{1.0\} \end{bmatrix}$$

where,

t : the end time of the event (see note 1).

NEWTONRAPHSON - Selection of Newton-Raphson scheme

$$\text{NEWTONRAPHSON} = \begin{bmatrix} \text{MODI, } n \\ \{\text{FULL, } 1\} \end{bmatrix}$$

where,

MODI, n : modified Newton-Raphson iterative scheme. If n is positive, stiffness will be recalculated at every nth iteration, whereas if n is negative, stiffness will be recalculated at the first n iterations of each increment or step.

FULL, 1 : full Newton-Raphson iterative scheme. Stiffness will be recalculated at each iteration.

MAXITERATIONS - Maximum number of iterations

$$\text{MAXITERATIONS} = \begin{bmatrix} n \\ \{6\} \end{bmatrix}$$

where,

n : maximum number of iterations allowed per step.

TOLERANCES - Equilibrium check tolerances

$$\text{TOLERANCES} = \begin{bmatrix} \text{dtol}, & \text{etol}, & \text{ftol} \\ \{0.001, & 0.001, & 0.001\} \end{bmatrix}$$

where,

dtol : displacement tolerance. A load step will be assumed converged if the displacement

$$\text{norms } \delta_i \text{ and } \delta_{i-1} \text{ at iteration } i \text{ and } i-1 \text{ satisfy } \left| \frac{\delta_i - \delta_{i-1}}{\delta_i} \right| \leq \text{dtol}$$

etol : energy tolerance. A load step will be assumed converged when the ratio of the iterative energy to the energy at first iteration is less than or equal to 'etol'.

ftol : force tolerance. A load step will be assumed converged when the ratio of the Euclidean norm of the residual force vector to the Euclidean norm of the incremental force vector is less than or equal to 'ftol'.

If any one of the above criteria is satisfied, the step is considered converged. If the user wants to exclude one or more criterion, a zero tolerance value should be entered. If the user wants to satisfy simultaneously all criteria, 'dtol' value should be entered as a negative value.

TOLERANCE FOR ERROR - Error tolerance for adaptive steps

$$\text{ERRTOLERANCES} = \begin{bmatrix} \text{ERRTOL} \\ \{1\text{E-}10\} \end{bmatrix}$$

where,

ERRTOL : error tolerance for adaptive time stepping. This is a user given parameter for the computation of time steps.

DELTAIME - Step size and optimum factor for auto-steps

$$\text{DELTAIME} = \begin{bmatrix} \text{dt, opt} \\ \{\text{no default}\} \end{bmatrix}$$

where,

- dt : initial step size estimate for automatic time stepping, arc length scheme and adaptive time stepping.
- opt : desired number of iterations (or scaling factor) for arc length or auto-step size selection, see note 3 and 7.

STEPLength - Step size limits for auto-steps

$$\text{STEPLength} = \begin{bmatrix} \text{min, max} \\ \{0, 0\} \end{bmatrix}$$

where,

- min : minimum allowable step size for automatic time stepping. Analysis is terminated if calculated step size is smaller than 'min' (see note 3).
- max : maximum allowable step size for automatic time stepping. Will be used if calculated step size is larger (see note 3).

EQUILIBRIUM - Equilibrium iterations key

$$\text{EQUILIBRIUM} = \begin{bmatrix} \text{OFF} \\ \text{ON, n} \\ \{\text{ON, 1}\} \end{bmatrix}$$

where,

- OFF : equilibrium iterations will not be performed at any load step.
- ON, n : equilibrium iterations will be performed at every nth load step.

TSFRE - Initial temperature

$$\text{TSFRE} = \begin{bmatrix} \text{tval} \\ \{0.0\} \end{bmatrix}$$

where,

tval : initial (stress-free) temperature (see note 4).

LINESEARCH - Line search key

$$\text{LINESEARCH} = \begin{bmatrix} \text{ON, n} \\ \{\text{OFF}\} \end{bmatrix}$$

where,

ON, n : maximum 'n' searches per iteration will be performed. If not given, n defaults to 9.

OFF : line search will not be performed during any iteration.

CREEP - Creep analysis key

$$\text{CREEP} = \begin{bmatrix} \text{ON, n1, n2, n3, n4, n5, n6, n7, n8} \\ \{\text{OFF}\} \end{bmatrix}$$

where,

ON : the creep analysis is activated. This must be after the first event, i.e. second or later events. Unless this command is specified with an ON parameter, creep analysis will not be activated. See note 6.

n1 : = 0 Creep strain law is given
= 1 Creep strain rate ($\dot{\epsilon}_c$) law is given

n2 : = 0 Explicit integration scheme
= 1 Implicit - Crank - Nicholson scheme
= 2 Implicit - Galerkin scheme
= 3 Implicit - Backward scheme

Analysis Data

Analysis Control Data

- n3 : = 0 Stress criterion is used for auto timestep
 = 1 Strain criterion for auto timestep
 = 2 Both stress and strain criteria are used for auto time step
- n4 : tolerance for stress criterion $\left| \frac{\Delta\sigma}{\sigma} \right|$ (default = 0.1)
- n5 : tolerance for stress criterion $\left| \frac{\Delta\varepsilon^c}{\varepsilon^e} \right|$ (default = 0.1)
- n6 : allowable incremental effective stress per step if stress criterion is used
 = 0 no check for this criterion
 > 0 no incremental effective stress can be larger than this specified value
- n7 : allowable incremental creep strain per step if strain criterion is used
 = 0 no check for this criterion
 > 0 no incremental creep strain can be larger than this specified value
- n8 : starting creep time for this event
 ≤ 0 use the ending creep time of the previous event as the starting creep time
 > 0 use this value as the starting creep time for this event

ROTFORCE - Rotational force calculation (applicable to direct transient dynamic analysis only)

$$\text{ROTFORCE} = \begin{bmatrix} \text{icentr, icorio, itange} \\ \{ \text{ON, ON, ON} \} \end{bmatrix}$$

where,

- icentr : key to calculate centrifugal acceleration force. Set to OFF to suppress calculation.
- icorio : key to calculate Coriolis acceleration force. Set to OFF to suppress calculation.
- itange : key to calculate tangential acceleration force. Set to OFF to suppress calculation.

TINTEGRATION - Method used in time integration for transient analysis

$$\text{TINTEGRATION} = \begin{bmatrix} \text{NEWMARK, beta, gama} \\ \{ \text{NEWMARK, 0.3025, 0.6} \} \end{bmatrix}$$

where,

NEWMARK, beta, gama : Newmark's method is used. Beta and gama are the integration parameters for Newmark's method; the default values for beta and gama are 0.3025 and 0.6 respectively.

HYPERELASTIC - Method of formulation for hyperelastic material models

$$\text{HYPERELASTIC} = \begin{bmatrix} m, n \\ \{0, 0\} \end{bmatrix}$$

where,

m : an integer identifying the formulation method for the analysis of hyperelastic material models;
= 0 use average constraint approach for low order elements (NORDR = 1) and modified invariants for higher order elements (NORDR = 2) (see Section 2.7.4 and note 8).
= 1 use average constraint approach (see note 8).
= 2 use modified strain energy function (see note 9).
= 3 use alternate modification for strain energy function (see note 9).
n : not used in this version, enter 0.

CHARLENGTH - change arc length size when using RESTART run

$$\text{CHARLENGT} = \begin{bmatrix} \text{ratio} \\ \{ \text{no default} \} \end{bmatrix}$$

where,

ratio : ratio of current length to previous length, must be greater than zero
= 1.0 do not change length size

- < 1.0 reduce length size
- > 1.0 enlarge length size

Notes:

1. In static analysis (excluding creep analysis), time is a pseudo time, i.e. load increments or steps are the same as time increments or steps.
2. If the sum of the number of steps in all events exceeds 246, the user will not be able to save post processing data for all the steps. In such a case, the user is advised to limit the frequency of step outputs using the FRQOUTPUT parameter.
3. For automatic time stepping, if previous step has converged, current step size will be calculated as,

$$\Delta t_i = \Delta t_{i-1} \sqrt{\frac{n_d}{n_{i-1}}}$$

where,

- $\Delta t_i, \Delta t_{i-1}$: current and previous load step sizes
- n_{i-1} : number of iterations performed in previous load step.
- n_d : desired number of iterations (or scaling factor, can be real number).
This is the variable opt in the DELTAIME parameter.

If a load step does not converge within the specified number of iterations, the step size is reduced by one fourth of the current size and the load step is restarted. The reduction of the step sizes continues until one of the following criteria is satisfied:

- A step converges
 - The step size becomes less than the minimum step length specified on 'STEPLength = min, max' command
 - The total number of steps, including the restart steps, exceed the number specified on the 'INCREMENTS = AUTO, n' command
4. TSFRE parameter can be specified *only* in the *first* *EVENT data group. This parameter is ignored if it is specified in the subsequent *EVENT data groups.

5. For linear direct transient dynamic analysis, only equal time steps are allowed in each event; the step size can change from event to event.
6. It should be noted that the creep option is not supported with orthotropic or hyperelastic material models.
7. Arc length scheme is a method which can be applied to solve the problem of snap-through buckling phenomena.

For the arc-length method, three options for 'opt' are available.

- (i) 'opt' (default = 4) ≥ 2 , the current length size will be calculated as,

$$\Delta l_i = \Delta l_{i-1} \sqrt{\frac{n_d}{n_{i-1}}}$$

where, n_{i-1} is the number of iterations used in the previous step, $n_d = \text{'opt'}$.

- (ii) 'opt' = -1, the same 'dt' will be assigned every initial iteration to obtain a new length for the step regardless of the iteration number of the previous step.
- (iii) 'opt' = -2 or less, the arc length size will be kept constant throughout the event, regardless of the number of iterations used in the previous step.

Some limitations of arc length scheme are:

- follower loading is not allowed
- body force and thermal load are not allowed in geometric nonlinear analysis
- temperature dependent property such as E, ν must be linear during the event
- not available for creep analysis in this version

Default time amplitude curve is always used by arc length method.

For arc length scheme, time between beginning and ending point is not considered, therefore, in order to trace the loading path, a factor defined by current load/ending load is used. If this factor equals to one, the load at time-end is reached. In case this full load at time-end is not reached due to limitation of maximum steps given in arc length input statement, then a restart can be used to continue the solution.

In order to display time history in post-processor for arc length scheme, the equal time step is calculated after the analysis is completed.

8. The default formulation method for hyperelastic analysis is the average constraint approach for low order elements, NORDR = 1, and modified invariants for high order

Analysis Data

Analysis Control Data

elements, NORDR = 2. The low order elements with bubble functions (i.e., NKTP = 2, 3 and NORDR = 12 or NKTP = 4 and NORDR = 1) are treated as follows:

- (i) Two dimensional elements NKTP = 2, 3 and NORDR = 12: bubble function formulation is assumed (see [Section 2.7.4](#) for details). The average constraint approach may be invoked by using NORDR = 1.
- (ii) Three dimensional element NKTP = 4 and NORDR = 1: average constraint approach is assumed. The bubble function approach may be invoked by using NORDR = 12.

The above choice of defaults indicates a preference in using the average constraint approach over using the bubble function for the analysis of hyperelastic material models. The average constraint approach gives better convergence characteristics in most practical problems.

9. To enhance convergence of low order elements (NORDR = 1) in hyperelastic analysis, both the Mooney-Rivlin and the generalized Mooney-Rivlin hyperelastic models are modified by one of the following terms,

$$W = \hat{W} + \overset{\circ}{W}$$

where,

W : modified strain energy density function

\hat{W} : original Mooney-Rivlin or generalized Mooney-Rivlin function

$\overset{\circ}{W}$: modification term for strain energy given by:

- (i) For $m = 2$:

$$\overset{\circ}{W} = C \left(\frac{1}{I_3^2} - 1 \right) + D (I_3 - 1)^2$$

where,

$$C = \frac{C_{11}}{2} + C_{21}$$

$$D = \frac{1}{2} \left[\frac{C_{11}(5\nu-2) + C_{21}(11\nu-5)}{1-2\nu} \right]$$

and C_{11} , C_{21} are the coefficients of the Mooney-Rivlin model, ν is the Poisson's ratio and I_3 is the third invariant of the Cauchy-Green deformation tensor.

(ii) For $m = 3$:

$$\overset{\circ}{W} = \int_0^{I_3} \frac{g(\xi)}{2} d\xi - \frac{1}{2}(C_{11} + 2C_{21}) \ln I_3$$

where,

$$g(I_3) = D(I_3 - 1) \quad \text{and}$$

$$D = \frac{K}{2} - \frac{1}{3}(C_{11} + 4C_{21})$$

where, C_{11} and C_{21} are the coefficients of the Mooney-Rivlin model, K is the bulk modulus and I_3 is the third invariant of Cauchy-Green deformation tensor.

The choice and the convergence characteristics of strain energy functions with the above modifications depend on the actual problem and its constraints.

7.1.4 *FREQCNTL Data Group - Direct frequency analysis control

Applicable analysis types: DFREQUENCY

This data group is always required in direct frequency analyses, and it specifies control parameters needed for analysis.

Group ID card:

*FREQCNTL, ID = n

Direct frequency control card set:

Entry No: 1 2 3
Variable:

FREQ1	FREQ2	NINCR
-------	-------	-------

Max char:

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	FREQ1	value of starting frequency in cycles/time
2	FREQ2	value of ending frequency in cycles/time (default value is FREQ1)
3	NINCR	equal increments between FREQ1 and FREQ2 (default value is zero)

7.1.5 *HEATCNTL Data Group - Heat Transfer Analysis Control

Applicable analysis types: SHEAT, THEAT

This data group is always required in steady state and transient heat transfer analyses and must be the first data group in the analysis set. It defines a load case identification number and control parameters for iterations, tolerances, convergence, etc.

Group ID card:

*HEATCNTL, ID = n

where, n is a unique load case identification number (up to 6 digits integer > 0). Only one load case is allowed at the present time. In linear analysis, only the group ID card is needed.

Control parameters data set: one card, required only for nonlinear heat transfer analysis

Card set 1: one

Entry No:	1	2	3	4
Variable:	MAXITE	IREFM	NTRFM	RTOL
Max char:	5	5	5	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	MAXITE	maximum number of iterations allowed in each time step [default = 1]. If convergence is not achieved within this number of iterations, then step size will be reduced in transient analysis (see *TIMEINTEG), or execution will be terminated in steady state analysis.
2	IREFM	conductivity and capacity matrices will be recalculated every IREFM iterations within each time step [default = 1].
3	NTRFM	conductivity and capacity matrices will be recalculated every NTRFM time steps [default = 1].
4	RTOL	flux tolerance. Flux convergence criterion uses Euclidean norm [default = 1.0E-3].

7.1.6 *LDCOMB Data Group - Load Combination

Applicable analysis types: STATIC

This data group may be used to combine different load cases, and should be given after all load cases have been defined.

Group ID card: ***LDCOMB**, ID = n

where, n is a unique load combination identification number (up to 6 digits integer number). It must be different than all the load case ID numbers specified on the *LDCASE group ID card. If n is not specified, the load combination ID number is set to the previous load combination (or load case) ID number plus 1.

Each load combination must begin with a *LDCOMB data group and it may be followed by the *LCTITLE and/or the *PRINTCNTL data group.

The input data consists of two sets of cards as shown below.

Card set 1: Control card

Entry No:	1	2	3	4
Variable:	IDISP	ISTRS	IFORCE	IREACT
Max char:	1	1	1	1

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IDISP	key to combine displacement (only global components can be combined) =0 - off =1 - on
2	ISTRS	key to combine averaged nodal stresses =0 - off =1 - global, principal and equivalent stresses =2 - global stresses only =3 - principal and equivalent stresses

- 3 IFORCE key to combine element internal forces, strain energy and rigid element forces
 =0 - off
 =1 - on
- 4 IREACT key to combine reaction forces at constrained nodes
 = 0 – off
 = 1 - on

Card set 2: Load case ID number and scaling factor definition

Entry No :	1	2	3	4	5	6	7	8
Variable:	LCASE1	FACT1	LCASE2	FACT2	LCASE3	FACT3	LCASE4	FACT4
Max char:	6	12	6	12	6	12	6	12

Entry Variable Description

- | | | |
|-----|--------|---|
| 1 | LCASE1 | load case ID number of the first load case to be combined |
| 2 | FACT1 | scaling factor for the first load case to be combined |
| ... | | ... |
| ... | | ... as required... |
| ... | | ... |
| 7 | LCASE4 | load case ID number of the fourth load case to be combined (if any) |
| 8 | FACT4 | scaling factor for the fourth load case (if any) |

Notes:

1. The keys on card set 1 activate only a combination of the appropriate output quantities. The actual printout is controlled by the appropriate variables on the *PRINTCNTL data group.
2. Input as many cards as desired in card set 2 to define the load case ID numbers and scaling factor data. Any zero load case ID number is ignored, thus one set of data (load case ID number and scaling factor) may be specified on each card, see example below.

Example 7.1.1: A sample input for load combination data is shown below:

```
*LDCOMB, ID = 151
  1, 3, 1
  101, 1.0
  105, 2.5
  109, -2.0
*LCTITLE
Combination of load cases 101, 105 and 109
*PRINTCNTL
AVND, 0
DISP, 10, 24
RIGF, 104, 135
*LDCOMB, ID = 201
  1, 2, 1
  101, -3.0, 105, 1.5
*LCTITLE
Combination of load cases 101 and 105
*PRINTCNTL, IREF = 105
*ENDDATA
```

7.1.7 *LCTITLE Data Group - Load Case Title

Applicable analysis types: STATIC, BUCKLING, NLSTATIC, LTRANSIENT, NLTRANSIENT

This data group may be used to specify a title for a typical load case (load combination case, or an event).

Group ID card:

*LCTITLE

Load case title card set:

Entry No: 1
Variable:

load case title

Max char: 80

Entry Variable Description

1 - - alphanumeric title (upto 80 characters). This title will appear on the printout for the current load case as a second heading of each page.

Note:

1. If this data group is not provided for the current load case, the second heading line of each page of the printout for the current load case will be blank.

7.1.8 *MODEOUT Data Group - Mode Selection for Output Options

Applicable analysis types: EIGENVALUE, BUCKLING

This data group may be used to select the modes for which the output quantities (e.g., modal stresses, strain energy, etc.) are desired. The modes are selected in sets, each set refers to a specific output control block, which starts with *EIGOUT data group, where the output quantities are selected.

Group ID card:

*MODEOUT

Mode selection card set:

Entry No:	1	2	3	\$ 4				
Variable:	<table border="1"><tr><td>MODE1</td></tr></table>	MODE1	<table border="1"><tr><td>MODE2</td></tr></table>	MODE2	<table border="1"><tr><td>INC</td></tr></table>	INC	<table border="1"><tr><td>IDBLK</td></tr></table>	IDBLK
MODE1								
MODE2								
INC								
IDBLK								
Max char:	5	5	5	6				

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	MODE1	mode number of the first mode in the set to be processed as per the options given in the output control block (*EIGOUT), whose ID number is IDLBLK
2	MODE2	last mode number of a range of modes in the set
3	INC	increment for the range of modes
--- tab (\$) ---		
4	IDBLK	output control block identification number (up to 6 digits interger > 0). This number refers to an *EIGOUT data group with the same ID number.

Notes:

1. Provide as many sets of modes as desired, one set per card. Each set must refer to a nonzero output control block ID number, otherwise it will be ignored. Output quantities will be provided in ascending order of the modes selected in this group.
2. Modes not specified in this data group will not be processed for any output option. This excludes, however, the eigenvector printout, which is controlled by the parameter NEGK in *EIGCNTL data group.

3. If a modal superposition analysis (e.g., modal transient dynamic analysis) is to follow an eigenvalue analysis run, and if a certain response quantity (e.g., stress) is desired, the corresponding modal response quantity (e.g., Modal stress) should be requested for all modes to be included in the modal superposition analysis.

7.1.9 *TIMEINTEG Data Group - Time Integration Parameters for Transient Heat Transfer

Applicable analysis types: THEAT

This data group is always required for transient heat transfer analysis and must follow the *HEATCNTL data group in sequence. It defines the time integration control parameters needed in the analysis.

Group ID card:

*TIMEINTEG

Control parameters data set: one card

Entry No:	1	2	3	4	5	6	7
Variable:	ALPHA	DTO	TMAX	DTINC	DTMAX	DTDEC	NTDEC
Max char	10	10	10	10	10	10	5

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	ALPHA	time integration scheme control parameter = 0.0 - forward difference (explicit Euler) = 0.5 - trapezoidal (Crank-Nicholson) = 0.67 - Galerkin = 1.00 - backward difference (implicit Euler) [Default = 0.5]
2	DTO	initial time step size
3	TMAX	maximum time [default = N * DTO where N is the total number of time steps given in the executive commands]. Execution will be terminated when time reaches TMAX.
4	DTINC	multiplication factor for changing time step size. [Default = 1.0, fixed time steps]. $DT_i = DT_{i-1} * DTINC$, where DT_i and DT_{i-1} are the current and previous time step sizes, respectively.
5	DTMAX	maximum or minimum time step size allowed [Default = DTO * DTINC**5 (see note 1)].

- | | | |
|---|-------|--|
| 6 | DTDEC | time step size decreasing factor [Default = 0.5]. For nonlinear problems if solution does not converge at certain time step. Time step size will be decreased by $DT = DT * DTDEC$ |
| 7 | NTDEC | maximum number of times for which step size can be decreased within one time step. If solution does not converge after NTDEC times, program will be terminated. [Default = 5] |

Notes:

1. Time step size can be changed (increased or decreased) according to DTINC, but when the time step size reaches the value DTMAX, it will become a fixed time step $DT = DTMAX$.
2. User can define the variable time step size using *STEPSIZE data group. If *STEP-SIZE is given, it will override the time step size specification given in this group.

7.1.10 *STEPWISE Data Group - User Specified Time Steps

Applicable analysis types: NLSTATIC, THEAT, NLTRANSIENT

This data group may be used to specify user defined time step sizes for transient heat transfer analysis and nonlinear static and dynamic analysis. In transient heat transfer analysis, time step sizes may also be defined in *TIMEINTEG data group. Specifications given in this data group (*STEPWISE) will override, however, those given in the *TIMEINTEG group.

Group ID card:

*STEPWISE

Step size data set:

Entry No:	1	2	3	4 - 7	8
Variable:	DT1	DT2	DT3	...	DT8
Max char	10	10	10	...	10

<u>Entry Variable</u>	<u>Description</u>
-----------------------	--------------------

1 - 8 DT1 to DT8	user specified time step sizes. Use as many cards as necessary, 8 entries per card. See notes 1, 2.
------------------	---

Notes:

1. The sum of all specified time steps should be equal to the TIMEATEND value specified on the *EVENT data group for nonlinear static analysis, or to the TMAX value specified on the *TIMEINTEG data group for transient heat transfer analysis. If the sum of all user specified time step sizes is not equal to TIMEATEND specified in the *EVENT data group for nonlinear static analysis, or TMAX specified in the *TIMEINTEG data group for transient heat transfer analysis, then:

If TIMEATEND (or TMAX) > sum of user specified time steps; the program will use the last specified time step until TMAX is reached.

If TIMEATEND (or TMAX) < sum of user specified time steps; the program adjusts the last specified time step(s) to match TMAX.

2. For nonlinear static analysis, the total number of time steps *must* equal the number 'n' specified on the 'INCR = USER, n' parameter in the *EVENT data group.

7.1.11 *CONVERGENCE Data Group - Convergence Criteria

Applicable analysis types: NLSTATIC

- This data group overrides the convergence parameters specified (or defaulted) in the *EVENT group using the command TOLERANCES.
- Once this data group is given in an event, it remains active in all subsequent events unless it is overridden by a new *CONVERGENCE group.
- Supplying just the group-id card activates the program default values for all criteria.
- Program default values are used for a typical criterion if it is not specified.

Four convergence criteria are available: Displacement (Translation), Rotation, Force and Moment. The user may exclude any criterion. The program default values are given in [Table 7.2](#). Examples illustrating the use of this data group are given in the notes below.

Group ID card: ***CONVERGENCE**

Entry No:	1	2	3	4	5	6	7
Variable:	LABEL	TOL1	USRVAL	NRM	IREF	TOL2	LABEL2
Max char	12	12	12	3	3	12	12

Convergence parameters data set

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	LABEL	Convergence criterion label (Character). Valid labels are: DISPLACEMENT (or TRANSLATION) ROTATION FORCE MOMENT Only the first 3 characters of LABEL are needed. If a criterion is not compatible with the type of the degrees of freedom present in the model (e.g., there are no rotational dof), it is automatically excluded.

Analysis Data

Analysis Control Data

- 2 TOL1 Tolerance value (Real) for the convergence criterion identified by LABEL.
 > 0.0 this is the tolerance value, criterion is activated.
 ≤ 0.0 criterion is excluded.
 For a typical criterion, convergence is achieved when the ratio of a scalar measure (norm) of the iterative quantity defined by LABEL to a reference value is less or equal to TOL1(see note 1).
- 3 USRVAL User reference value (Real) to be used in evaluating the criterion, see note 1.
 > 0.0 this is the reference value used in evaluating the criterion.
 ≤ 0.0 the reference value is computed by the program according to LABEL, NRM (entry 4) and IREF (entry 5). The reference value used is: max (program reference value and abs (USRVAL)), see note 1.
- 4 NRM Norm type (Integer) to be used in computing the norms of both the iterative quantity defined by LABEL and its program reference value. Valid norm types are (see note 2):
 = 0, 2 P-2 norm (Euclidean norm)
 = 1 P-1 norm
 = 3 P-∞ norm (infinity norm)
- 5 IREF Type of program reference value (Integer). See notes 3-5. With the definitions:
 PGMVAL = program's reference value
 VINC = norm of the incremental vector associated with the quantity identified by LABEL
 VTOT = norm of the total vector-valued quantity (e.g., total applied force)
 RTOT = norm of reaction vector (for either FORCE or MOMENT)
 the valid values are:
 = 0, 1 PGMVAL = max (VINC, VTOT)
 = 2 PGMVAL = VINC
 = 3 PGMVAL = VTOT
 = 4 PGMVAL = max (VINC, VTOT, RTOT) (valid only if LABEL = FORCE or MOMENT)

- = 5 PGMVAL = norm of the residual force vector at end of first iteration of a typical step (valid only if LABEL = FORCE or MOMENT)
- 6 TOL2 Upper bound tolerance value (Real). This parameter is used in conjunction with the next entry, LABEL2, to guard against a tight tolerance specified by TOL1. If the criterion LABEL has not converged within the tolerance TOL1, it is checked against TOL2. If it satisfies TOL2, it is assumed to have converged provided that another criterion (designated by LABEL2 (next entry) has converged within its own TOL1 value (see note 6).
- 7 LABEL2 label for another criterion (Character) to be used in conjunction with TOL2. See note 6.
Valid labels are (only first 3 characters are needed):
DISPLACEMENT (or TRANSLATION)
ROTATION
FORCE
MOMENT
ALL
NONE (or blank field)
- if LABEL2 = ALL, all other criteria must be satisfied within their TOL1 values before TOL2 is used for the criterion designated by LABEL.
- if LABEL2 = ANY, any other criterion must be satisfied within its TOL1 value before TOL2 is used for LABEL.
- if LABEL2 = NONE, the upper bound tolerance TOL2 is not used for the criterion LABEL which must satisfy its TOL1 value.

Notes:

1. For a typical criterion, convergence is achieved whenever,

$$\text{RATIO} = \frac{\text{ITRVAL}}{\text{REFVAL}} \leq \text{TOL1} \qquad (7.1)$$

where, ITRVAL is the norm of the iterative vector representing the quantity identified by LABEL, and REFVAL is its reference value. REFVAL is computed as follows:

$$\begin{aligned} \text{REFVAL} &= \text{USRVAL}, & \text{if } \text{USRVAL} > 0.0 \\ \text{REFVAL} &= \text{PGMVAL}, & \text{otherwise} \end{aligned} \tag{7.2}$$

where, PGMVAL is the program's computed reference value for the criterion identified by LABEL, such that

$$\text{PGMVAL} = \max[\text{PGMVAL}, \text{abs}(\text{USRVAL})] \tag{7.3}$$

2. The p-norms of a vector \mathbf{x} with n components define scalar measures of the vector, and are expressed as,

$$\|\mathbf{x}\|_p = (|x_1|^p + |x_2|^p + \dots + |x_n|^p)^{1/p}, \quad p = 1, 2, \infty \tag{7.4}$$

For $p = 1, 2,$ and ∞ respectively, we have,

$$\|\mathbf{x}\|_1 = \sum_{i=1}^n |x_i| \tag{7.5}$$

$$\|\mathbf{x}\|_2 = \sqrt{\sum_{i=1}^n |x_i|^2} \tag{7.6}$$

$$\|\mathbf{x}\|_\infty = \max(|x_i|), \quad i = 1, 2, \dots, n \tag{7.7}$$

Thus, the P-1 norm is the sum of the absolute values of the n entries in \mathbf{x} , the P-2 norm (also known as the Euclidean norm) is the square root of the sum of the squares, and the infinity norm is the maximum absolute value among the entries in the vector. The p-norms are related by the inequalities,

$$\frac{1}{n} \|\mathbf{x}\|_1 \leq \|\mathbf{x}\|_2 \leq \sqrt{n} \|\mathbf{x}\|_\infty \tag{7.8}$$

3. Let $R_u, \|\delta \mathbf{u}^{(i)}\|, \bar{u}$ and ϵ_u be respectively the value of RATIO, ITRVAL, REFVAL and TOL1 given in [Equation 7.1](#) when LABEL is DISPLACEMENT (or TRANSLATION). Similarly, let $R_p, \|\mathbf{r}^{(i)}\|, \bar{P}$ and ϵ_p denote these variables when LABEL is FORCE. For these criteria, convergence is achieved when,

$$R_u = \frac{\|\delta \mathbf{u}^{(i)}\|}{\bar{u}} \leq \epsilon_u \tag{7.9}$$

$$R_p = \frac{\|\mathbf{r}^{(i)}\|}{\bar{p}} \leq \varepsilon_p \quad (7.10)$$

where,

- $\|\delta \mathbf{u}^{(i)}\|$: norm of the iterative displacement vector at iteration i of a typical load step (only translational DOFS are considered)
- $\|\mathbf{r}^{(i)}\|$: norm of the residual force vector at end of iteration i of a typical load step (only translational DOFS are considered), which is given by

$$\mathbf{r}^{(i)} = {}^t\mathbf{p} - {}^t\mathbf{f}^{(i)} \quad (7.11)$$

where ${}^t\mathbf{p}$ is the applied load vector and ${}^t\mathbf{f}^{(i)}$ is the internal force vector at end of iteration i .

4. For the ROTATION and MOMENT criteria, analogous expressions as in [Equation 7.9](#) through [Equation 7.11](#) are used, but only rotational DOFs are considered in the displacement and load vectors.
5. The program's reference value for a typical criterion is computed according to NRM (the norm type) and IREF (the program's options). Considering the DISPLACEMENT and FORCE criteria (the ROTATION and MOMENT criteria are treated similarly, with rotational DOFs in the associated vectors considered instead of the translational ones), let also,

- $\Delta \mathbf{u}^{(i)}$: incremental displacement vector (translational DOFs only) up to and including iteration i of a typical step
- ${}^t\mathbf{u}^{(i)}$: total displacement vector (translational DOFs only) up to and including iteration i of a typical step
- $\Delta \mathbf{p}$: incremental load vector (translational DOFs only) at a typical step
- ${}^t\mathbf{p}$: total load vector (translational DOFs only) at a typical step
- ${}^t\mathbf{q}^{(i)}$: reaction vector (translation DOFs only) at end of iteration i of a typical step.

Analysis Data

Analysis Control Data

$$\begin{aligned} \text{UINC} &= \|\Delta \mathbf{u}^{(i)}\|, & \text{UTOT} &= \|\mathbf{t}_{\mathbf{u}}^{(i)}\|, \\ \text{PINC} &= \|\Delta \mathbf{p}\|, & \text{PTOT} &= \|\mathbf{t}_{\mathbf{p}}\|, \end{aligned} \quad (7.12)$$

$$\begin{aligned} \text{RTOT} &= \|\mathbf{t}_{\mathbf{q}}^{(i)}\| \\ \text{RES1} &= \|\mathbf{r}^{(1)}\| \end{aligned} \quad (7.13)$$

ZTOT = RTOT, if any constrained dof (either translational or rotational) has a nonzero specified displacement value in the *SPDISP data
= 0.0, otherwise

With these definitions, the program reference values for \bar{u} and \bar{p} (see [Equation 7.9](#) and [Equation 7.10](#)) take the following values according to the input parameter IREF (entry 5):

IREF	Program's \bar{u}	Program's \bar{p}
0, 1	max (UINC, UTOT)	max (PINC, PTOT, ZTOT)
2	UINC	max (PINC, ZTOT)
3	UTOT	max (PTOT, ZTOT)
4	not valid	max (PINC, PTOT, RTOT)
5	not valid	RES1

The last option in the above table (IREF = 5) is included for compatibility with the force criterion used in the *EVENT data (with the command TOLERANCES), so that users may continue to use similar capability with new ones if they wish so. Other options shown above are included to provide the user with more control and to cover many situations. For example, problems with specified displacement control where no physical loads are applied, load-unload cases, etc.

- The parameters TOL2 and LABEL2 are provided for more flexibility in evaluating the criteria. They are primarily intended to guard against tight tolerance values set by TOL1. To illustrate their usage, consider [Equation 7.9](#) and [Equation 7.10](#) for the DISPLACEMENT and FORCE criteria, respectively with u and p as the TOL1 values. Let the input data be,

DISPLACEMENT, ε_u , 0.0, 0, 0, α_u , FORCE

FORCE, ε_p , 0.0, 0, 0, α_p , DISPLACEMENT

where, ε_u and α_u are the lower bound and upper bound tolerance values for DISPLACEMENT criteria. ε_p and α_p are the lower bound and upper bound tolerance values for FORCE criteria.

The following algorithm is performed:

- (a) Compute R_u and R_p
 - (b) If $R_u \leq \varepsilon_u$, u-criterion has converged, go to step (e).
 - (c) If $R_u \leq \alpha_u$ and $R_p \leq \varepsilon_p$ u-criterion has converged within its upper bound and p-criterion within its lower bound, go to step (e).
 - (d) u-criterion has not converged.
 - (e) Continue.
 - (f) If p-criterion has not been evaluated, go to step (a) and interchange u with p .
7. The following examples illustrate the use of this data group to show how the parameters may be manipulated.

Analysis Data

Analysis Control Data

Example 1:

```
** this is the analysis data block only
* EVENT, ID = 1
  INCREMENTS = EQUAL, 2
  TOLERANCES = 0.001, 0.001
** Include other pertinent data for loading and B.C.
* ENDDATA
```

In this example, the *CONVERGENCE data group is not provided, the convergence criteria used are those activated by the command TOLERANCES (or its defaults) in the * EVENT group.

Example 2:

```
* EVENT, ID = 1
INCREMENT = EQUAL,2
TOLERANCES = 0.001, 0.001, 0.001
TIMEATEND = 1.00
* CONVERGENCE
** No data are provided, just group-id, program defaults are activated
** Include other pertinent data for loading and B.C.
* EVENT, ID = 2
INCREMENTS = EQUAL, 2
TIMEATEND = 2.00
** Include other pertinent data for loading
* EVENT, ID = 3
INCREMENTS = EQUAL, 2
TIMEATEND = 3.00
* CONVERGENCE
DIS, 0.0
FOR, 0.05
*ENDDATA
```

In this example, 3 events are used. In the first event, the *CONVERGENCE group-id is given with no data following it. Therefore, the program default values for the *CONVERGENCE data is used. Note that this overrides the convergence criteria of the *EVENT group through its command TOLERANCES. The second event will continue using the convergence parameters of the preceding event since no new *CONVERGENCE group is given. In the third event, a new *CONVERGENCE is encountered. This overrides the previous data. Also, the ROTATION and MOMENT criteria are not specified and the program will use the default values for these criteria. In addition, the displacement criterion is excluded by the user ($TOL1 \leq 0.0$). For the force criterion, TOL1 is 5%, and all other parameters are zero (or blank).

Analysis Data

Analysis Control Data

Example 3

```
*CONVERGENCE  
ROT, 0.0  
DIS, 0.005, 0.0, 0, 0, 0.01, FORCE  
FOR, 0.001, 1000.0, 3, 0, 0.05, DISP  
MOM, 0.01, -1.0, 0, 4, 1.E8, ANY
```

In this example:

- ❑ The ROTATION criterion is excluded.
- ❑ The reference value for the DISPLACEMENT criterion is to be computed by the program, and if the FORCE (LABEL2) criterion is satisfied to within 0.001 tolerance, then the upper bound tolerance can be as high as 0.05 (TOL2).
- ❑ For the force criterion, the user is specifying the reference value (1000.0) directly and the infinity norm (NRM = 3). The line of data is read as FORCE convergence criterion (for translational DOF). This is translated whenever:

(a) The maximum residual force at any one DOF is greater than $0.001 \times 1000 = 10$

or

(b) The maximum residual force at any one DOF is less than $0.05 \times 1000 = 50$

AND

the DISP criterion is satisfied to within its lower bound tolerance of 0.005.

- ❑ For the MOMENT criterion, a 1% tolerance is specified, the reference value should be computed by the program with option 4 (IREF = 4) but it should not be less than 1.0 (abs (USRVAL) = 1.0). If any other (active) criterion is satisfied to within its lower bound tolerance, then consider that the MOMENT criterion is satisfied, since a very large TOL2 value (1.0E08) is given.

Table 7.2: Default values for convergence criteria (default values apply only if a criterion is not specified by the user).

LABEL	TOL1	USRAL	NRM	IREF	TOL2	LABEL2
DISPLACEMENT	0.001	0.0	2	1	0.05	FORCE
ROTATION	0.001	-0.0	2	1	0.05	DISPLACEMENT
FORCE	0.001	-1.0	2	1	0.05	MOMENT
MOMENT	0.001	-1.0	2	1	0.05	ROTATION

7.2 Fourier Analysis Data

(for axisymmetric elements with non-axisymmetric loading)

7.2.1 *FRCNTL Data Group - Fourier Analysis Control

Applicable analysis types: STATIC

This data group is required if the model contains element types NKTP = 34 or 37 (axisymmetric elements subjected to non-axisymmetric loads).

Group ID card:

*FRCNTL

Fourier analysis card set: one card

Entry No:	1	2	3	4
Variable:	NFOSET	MXFCN	MXFSN	NANGLE
Max char:	3	2	2	2

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
--------------	-----------------	--------------------

1	NFOSET	number of different Fourier distribution set
2	MXFCN	maximum number of cosine terms used in any Fourier set. This is the highest harmonic number in the cosine terms.
3	MXFSN	maximum number of sine terms used in any Fourier set. This is the highest harmonic number used in the sine terms.
4	NANGLE	number of sections at which output is desired. The list of angles is to be given in*ANGSEC group.

Note:

1. The Fourier coefficients for all Fourier distribution sets are to be given in *FRCOEF data group.

7.2.2 *FRCOEF Data Group - Fourier Coefficients

Applicable analysis types: STATIC

This data group is required if the model contains element types NKTP = 34 or 37 (axisymmetric elements subjected to non-axisymmetric loading).

The Fourier coefficients defining each Fourier set are defined in this data group. Each Fourier set will have at least two cards, as described below.

Group ID card:

*FRCOEF

Card set 1:

Entry No:	1	2	3
Variable:	LABEL	LFN	CONST
Max char:	5	3	12

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	LABEL	label to signal the beginning of the definition of a new Fourier distribution set = FRSET
2	LFN	identification number for this Fourier set (integer up to 999). This is the Fourier set ID number referenced in *CFORCE and *PRESSURE data groups.
3	CONST	value of the constant term of this series

Card set 2: (Defines the coefficients of the cosine and sine terms)

Entry No:	1	2	3	4	5	6	7	8	9
Variable:	LABEL	n	COEFn	n	COEFn	n	COEFn	n	COEFn
Max char:	3	3	12	3	12	3	12	3	12

Analysis Data

Fourier Analysis Data

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	LABEL	label to designate whether cosine terms or sine terms are entered on this card, see note 1 = COS - cosine terms are entered on this card = SIN - sine terms are entered on this card
2 - 9	n, COEFn	up to 4 pairs of harmonic number and associated coefficient representing either cosine terms or sine terms as indicated by LABEL. n should be greater than zero and the constant term should be defined in CONST.

Notes:

1. Use as many cards as needed in card set 2 to define the cosine and the sine terms. Each card in card set 2 must begin with 'COS' or 'SIN'. Repeat the sequence of card set 1 and 2 for each additional Fourier set.
2. Refer to the description of NKTP = 34 element ([Section 4.25](#)) for additional information on the procedure used to define the Fourier coefficients.
3. The types of non-axisymmetric loads implemented in NISA are concentrated nodal forces (*CFORCE data group) and pressure loads (*PRESSURE data group). In either case, the loads specified (in these data groups) are multipliers for the Fourier coefficients given in this data group (*FRCOEF), see example below.

Example 7.2.1:

A model is made up entirely of 4-noded (NORDR = 1) quadrilateral elements of type NKTP = 34. There is a nodal load applied in the radial direction (FX) at node 102. This is in reality a *ring load* acting on the nodal circle (corresponding to node 102) according to the Fourier distribution,

$$FX(\theta) = 100(1 + 2 \cos 3\theta + 5 \sin 3\theta)$$

This force is for the entire 360 degree circumferential of the nodal circle (i.e., the factor $2\pi r$ is already included in the definition).

There is also a pressure load on face 3 of element 52. This face has 2 nodes, and the pressure intensities (force/area) at the first and second nodes of the face are given, respectively by,

$$P1(\theta) = 4(1 + 6 \cos 3\theta + 8 \sin 2\theta)$$

$$P2(\theta) = 3(1 + 6 \cos 3\theta + 8 \sin 2\theta)$$

The Fourier coefficient and load data for this case will be given by,

```
*FRCNTL
    2, 3, 3, 6
*FRCOEF
    FRSET, 1, 1.0
    COS, 3, 2.0
    SIN, 3, 5.0
**
    FRSET, 2, 1.0
    COS, 3, 6.0
    SIN, 2, 8.0
*ANGSEC
    0.0, 22.5, 45.0, 90.0, 135., 180.
**
*PRESSURE
    52, 0, 0, 3, 1, 2
    4.0, 3.0
*CFORCE
    102, FX, 100.0 $1
```

7.2.3 *ANGSEC Data Group - Angular Sections for Response Calculation

Applicable analysis types: STATIC

This data group is required if the model contains element types NKTP = 34 or 37 (axisymmetric elements subjected to non-axisymmetric loads).

Group ID card:

*ANGSEC

Angular sections card set:

Entry No:	1	2	3	4	5	6	7	8
Variable:	ANG1	ANG2	ANG3	ANG4	ANG5	ANG6	ANG7	ANG8
Max char:	10	10	10	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	ANG1	first angle (in degrees) at which the response is desired
2 - 8	ANG2	second...
	to	to
	ANG8	ANG8 eighth angle at which the response is desired.

Notes:

1. If more than eight angles are to be specified, continue on additional cards, eight entries per card.
2. The number of angles given in this group must match the value of NANGLE as given in *FRCNTL data group.
3. Displacements will be printed at all these angular sections. If stresses are calculated they will be calculated at all the sections listed in this data group. Post-plotting may be done at only these sections.

7.3 Boundary Conditions and Loading

(for STATIC, EIGENVALUE, BUCKING, NLSTATIC, LTRANSIENT, and NLTRANSIENT)

7.3.1 *SPDISP Data Group - Specified Displacement Boundary Conditions

Applicable analysis types: STATIC, EIGENVALUE, BUCKLING, NLSTATIC, LTRANSIENT, NLTRANSIENT

This data group is used to define specified displacement boundary conditions at nodes. The requirement of this data group depends on the analysis type it is used in:

1. For *static* analysis, this data group is always required for the first load case. If this data group is not included in the definition of any subsequent load case, the boundary conditions of the preceding load case are used.
2. For *eigenvalue* and *direct transient dynamic* analyses, this data group is always required except for the case of free-free (unconstrained) structure. Only one set of boundary conditions is allowed in these analysis types.
3. For *buckling* analysis, this data group is always required in the static analysis pass and should not be repeated in the eigenvalue analysis pass. Only one set of boundary conditions is allowed in buckling analysis.
4. For *nonlinear static* analysis, this data group is always required for the first event, and it should not be specified again in the subsequent events, if any.

Group ID card: *SPDISP, TCRV = n

where the parameter TCRV = n is required in nonlinear static analysis and in direct transient dynamic analysis, 'n' is an integer designating a time-amplitude curve identification number defined in the *TIMEAMP data group (see note 4).

	\$							
Entry No:	1	2	3	4	5	6	7	8 - 9
Variable:	NODE	LABEL	DISP	LASTND	NODINC	LABEL	LABEL	...
Max char:	6	4	12	6	6	4	4	...

Analysis Data

Boundary Conditions and Loading

\$

Entry No:	10	11		
Variable:	<table border="1"><tr><td>LABEL</td><td>ITCRV</td></tr></table>	LABEL	ITCRV	
LABEL	ITCRV			
Max char:	4	6		

Specified displacement card set:

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NODE	node number
2	LABEL	label for degree of freedom (see note 2). Allowable labels are: UX, UY, UZ - translations along X, Y, and Z axes, respectively ROTX, ROTY, ROTZ - rotations about X,Y and Z axes, respectively, in radians UXYZ - all translations RXYZ - all rotations ALL - all translations and all rotations
3	DISP	specified displacement value
4	LASTND	last node of a range of nodes with the same specified displacement
5	NODINC	positive increment for the range of nodes
— tab (\$) —		
6 - 10	LABEL	the same specified displacement value can be assigned to the additional degrees of freedom by specifying the proper labels.
— tab (\$) —		
11	ITCRV	time-amplitude curve ID (applicable to NLSTATIC, LTRANSIENT, and NLTRANSIENT only, see note 4)

Notes:

1. In static analysis, boundary conditions may be changed from one load case to another. NISA checks the boundary conditions of a typical load case against those of the preceding load case. If the two sets of boundary conditions are similar (same constrained degrees of freedom with possibly different specified values), the decomposed stiffness matrix of the preceding load case is used, otherwise a new decomposition is initiated. Users are advised that multiple load cases should be ordered in the data deck such that load cases with similar boundary conditions are grouped together.
2. If a local displacement coordinate system has been defined at a node, then the displacement labels (UX, UY,..., ROTZ) refer to translations and rotations in the local system.
3. Specified nonzero displacement values are not allowed in eigenvalue analysis.
4. In nonlinear static and direct transient dynamic analyses, specified displacement values at any time/load step are computed using the displacement values and the time-amplitude curve (ITCRV) given in card set 1. If ITCRV variable is not given in the card set 1, then the time-amplitude curve ID given in group-ID parameter TCRV is used. A default time-amplitude curve as described in [Section 6.7.4](#) is used when the TCRV parameter is not supplied by the user.

7.3.2 *CFORCE Data Group - Concentrated Nodal Forces

Applicable analysis type: STATIC, BUCKLING, NLSTATIC, LTRANSIENT, NLTRANSIENT, DFREQUENCY

This data group may be used to define concentrated loads at nodes.

Group ID card: *CFORCE, TCRV = n

where the parameter ‘TCRV = n’ is required in nonlinear static and direct transient dynamic analyses, direct frequency analysis and n is an integer designating a time-amplitude curve or frequency function identification number defined in the *TIMEAMP data group (see note 3).

Nodal force card set:

	\$						
Entry No:	1	2	3	4	5	6	7
Variable:	NODE	LABEL	FORCE	LASTND	NODINC	LFN	ITCRV
Max char:	6	4	12	6	6	3	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NODE	node number
2	LABEL	label for the force or moment component. Allowable labels are: FX, FY, FZ - forces in the X, Y, and Z directions, respectively MX, MY, MZ - moments about the X, Y, and Z axes, respectively
3	FORCE	value of the force or moment.
4	LASTND	last node of a range of node numbers to be assigned the same value of force or moment.
5	NODINC	positive increment for the range of node numbers.
— tab (\$) —		
6	LFN	Fourier distribution set index. Required only for axisymmetric elements subjected to nonaxisymmetric loading, NKTP = 34 or 37, enter zero otherwise.

- 7 ITCRV time-amplitude curve identification number defined in the *TIMEAMP data group. (applicable to NLSTATIC, LTRANSIENT and NLTRANSIENT only, see note 3)
Frequency function identification number defined in the *FREQUENCY data group (applicable to DFREQUENCY only, see note 3)

Notes:

1. If a local displacement coordinate system is defined at a node, then specified concentrated forces and moments at this node are referred to the local coordinate system. Otherwise, they are referred to the global cartesian coordinate system.
2. A moment load (MX, MY, or MZ) can be applied at a node only if the corresponding rotational degree of freedom (ROTX, ROTY, or ROTZ) exists at that node. Thus for example, moment loads may not be applied to a solid element (NKTP = 4), since solids have only translational degrees of freedom.
3. In direct frequency analysis, nodal forces or moment values at any frequency are computed using the force or moment values and the frequency function (ITCRV) given in nodal force card set. If ITCRV variable is not given in the nodal force card set, then the time amplitude curve/frequency function ID given in group-ID parameter TCRV is used. A default timeamplitude curve/frequency function as described in [Section 6.7.4](#) respectively is used when the TCRV parameter is not supplied by the user.
4. In nonlinear static analysis, follower concentrated loads are defined in the *CFOLLOWER data group.

7.3.3 *CFOLLOWER Data Group - Follower Concentrated Forces

Applicable analysis type: NLSTATIC, NLTRANSIENT

This data group may be used in nonlinear static and dynamic analysis to specify follower concentrated forces. A follower force is deformation dependent and its direction is assumed to be normal to a specified element face at a given node. A concentrated moment cannot be applied directly in this group. However, it can be specified in terms of equivalent point forces.

Group ID card:

*CFOLLOWER, TCRV = n

where 'n' is an integer designating a time-amplitude curve identification number defined in the *TIMEAMP data group, see note 1. It should be noted that the curve ID number 'n' from the *PRESSURE data group overrides the curve ID number from this group (if both groups are specified in the same deck).

Follower force card set:

Entry No:	1	2	3	4	5					
Variable:	<table border="1"><tr><td>NODE</td></tr></table>	NODE	<table border="1"><tr><td>FORCE</td></tr></table>	FORCE	<table border="1"><tr><td>NELID</td></tr></table>	NELID	<table border="1"><tr><td>IDFACE</td></tr></table>	IDFACE	<table border="1"><tr><td>ITCRV</td></tr></table>	ITCRV
NODE										
FORCE										
NELID										
IDFACE										
ITCRV										
Max char:	6	12	6	1	6					

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NODE	node number
2	FORCE	value of the concentrated follower force. A positive value indicates a force acting towards the face in the normal direction.
3	NELID	element identification number
4	IDFACE	face number (of the referenced element) on which the force will be always acting in the normal direction at the specified node, see NISA element library for face numbering convention
5	ITCRV	time-amplitude curve ID, see note 1.

Note:

1. In nonlinear analysis, nodal follower force values at any time/load step are computed using the force values and the time-amplitude curve (ITCRV) given in follower force card set. If ITCRV variable is not given in the follower force card set, then the time-amplitude curve ID given in the group-ID parameter TCRV is used. A default time-amplitude curve as described in [Section 6.7.4](#) is used when the TCRV parameter is not supplied by the user.

7.3.4 *PRESSURE Data Group - Pressure Loads

Applicable analysis type - STATIC, BUCKLING, NLSTATIC, LTRANSIENT, NLTRANSIENT, DFREQUENCY.

This data group may be used to define pressure or distributed loads on element faces (or edges). Refer to NSA element library for elements for which pressure loading can be applied. Pressure loads or distributed loads can be in the direction of a vector specified in *VECTOR data group. For nonlinear analyses, the pressure may be specified as follower pressure. In this case, pressure is always normal to the surface and the direction given in *PRESSURE will be ignored.

Group ID card:

*PRESSURE, TCRV = n

where the parameter 'TCRV = n' is required in nonlinear static analysis and in direct transient dynamic analysis, 'n' is an integer designating a time-amplitude curve or frequency function, identification number defined in the *TIMEAMP or *FREQFUNCTION data group (see note 2).

Input data for pressure force on a typical element (or range of elements) is given in two card sets. The first set has flags for identifying element numbers, face number and pressure related parameters. The second set contains the pressure values for all the nodes on the element face.

Card set 1: one card; always required.

	\$								
Entry No:	1	2	3	4	5	6	7	8	9
Variable:	NELID	LASTEL	INC	IDFACE	NCARDS	LFN	UPRES	ITCRV	IVECT
Max char:	6	6	6	1	1	3	10	6	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NELID	element number
2	LASTEL	last element of a range of elements with the same pressure loading. Enter 0 if this card defines pressure loading on a single element.
3	INC	positive increment for the range of elements. Enter 0 or 1 if loading just a single element.
4	IDFACE	face number on which the pressure acts (see NISA element library for face numbering convention).
5	NCARDS	number of cards in card set 2 required to complete the pressure data for this face. If NCARDS = 0, pressure is assumed to be uniform with a value UPRES' (entry number 7 on this card), and the second card set is not required in this case.
6	LFN	<p>this entry has the following interpretation depending on the analysis type (see note 3).</p> <p><u>for STATIC analysis:</u></p> <p>= n - Fourier distribution index, only required for axisymmetric elements subjected to non axisymmetric loading; NKTP = 34, 37; enter 0 otherwise</p> <p><u>for NLSTATIC analysis:</u></p> <p>= 0 - pressures will not be treated as a follower pressure.</p> <p>= 1 - pressure will be treated as a follower pressure.</p> <p>not used for BUCKLING and LTRANSIENT analyses</p>
— tab (\$) —		
7	UPRES	uniform pressure value at all nodes of the face (used if NCARDS = 0), UPRES value is ignored if NCARDS > 0.
8	ITCRV	<p>time-amplitude curve ID (applicable to NLSTATIC, LTRANSIENT, and NLTRANSIENT see note 2).</p> <p>Frequency function identification number defined in the *FREQFUNCTION data group (applicable to DFREQUENCY only, see note 2)</p>
9	IVECT	vector ID defined in *VECTOR data group. The distributed load or pressure load is acting in direction of this vector. When IVECT is not given, pressure load is acting toward the element face. (see note 1)

Analysis Data

Boundary Conditions and Loading

Card set 2: required only if NCARD > 0

Entry No:	1	2	3	4	5	6	7	8
Variable:	P1	P2	P3	P4	P5	P6	P7	P8
Max char:	10	10	10	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1 - 8	P1 -P8	pressure values at all nodes of the face. If the face has more than 8 nodes, continue on additional cards as necessary, eight entries per card. Number of cards to define the pressure must be equal to NCARDS.

Notes:

1. If IVECT is given, the pressure load is acting in the direction of the vector of IVECT on the element face. When IVECT is not given, a positive pressure load corresponds to a surface traction normal to and acting towards the element face.
2. In nonlinear static and direct transient dynamic analyses, pressure values at any time/load step are computed using the pressure values and the time-amplitude curve (ITCRV) given in pressure loads card set. In direct frequency analysis, pressure values at any frequency are computed using the pressure values and the frequency function (ITCRV) given in pressure loads card set. If ITCRV variable is not given in card set 1, then the time-amplitude curve/frequency function ID given in the group- ID parameter TCRV is used. A default time-amplitude curve as described in [Section 6.7.4](#) is used when the TCRV parameter is not supplied by the user.
3. If the NISA II file is being prepared for use with SHAPE, then the pressure may be specified to continue acting normal to the changing boundary. This can be achieved by specifying a value of -1 for LFN, the sixth entry in card set 1.

7.3.5 *BEAMLOAD Data Group - Beam Loads

Applicable analysis types: STATIC, BUCKLING, NLSTATIC, LTRANSIENT, NLTRANSIENT

This data group may be used to define distributed and concentrated loads applied to 2-D or 3-D *beam* elements (NKTP = 11, 12 or 13, 39), at arbitrary locations along the beam axis. The applied loads can be defined in the *beam's local coordinate system*, in the global coordinate system, or in the direction of a defined vector. It includes transverse forces and bending moments. The forces and moments can be concentrated or distributed with uniform or linear variation. Refer to [Figure 7.1](#) for an example.

Group ID card: ***BEAMLOAD, TCRV = n**

where the parameter 'TCRV = n' is required in nonlinear static analysis and in direct transient dynamic analysis, 'n' is an integer designating a time-amplitude curve identification number defined in the *TIMEAMP data group (see note 3).

Beam load card set:

	\$											
Entry No:	1	2	3	4	5	6	7	8	9	10	11	12
Variable:	NELID	IPL	IRL	IFM	LASTEL	INC	X1	X2	P1	P2	LFN	ITCRV
Max char:	6	1	1	1	6	6	10	10	10	10	1	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NELID	element number
2	IPL	key for direction of load <ul style="list-style-type: none"> < 0 - the load is applied in the direction of a vector defined in *VECTOR datagroup. The absolute value of IPL is the corresponding vector ID for this load. = 0 - load is applied in the local xy plane (concentrated or distributed force is in the local y direction, moment is about the local z axis).

Analysis Data

Boundary Conditions and Loading

- = 1 - load is applied in the local xz plane (concentrated or distributed force is in the local z direction, moment is about the local y axis).
 - = 2 - load is applied in the beam axis
 - = 3 - load is applied in the global x-axis
 - = 4 - load is applied in the global y-axis
 - = 5 - load is applied in the global z-axis
- 3 IRL load location definition key (to interpret entries 7 and 8 on this card)
- = 0 - location of load is defined in terms of a ratio of the beam length measured from node 1 (i.e., normalized to the beam length)
 - = 1 - location of load is defined by actual distance measured from node 1 (i.e., not normalized to the beam length)
- 4 IFM type of load (direction is defined by entry 2, IPL)
- = 0 - applied load is a concentrated or distributed force
 - = 1 - applied load is a concentrated or distributed bending moment
- 5 LASTEL last element number of a range of elements with the same loading, enter zero if loading is on one element
- 6 INC positive increment for the range of elements, enter zero if loading is on one element
- tab (\$) -----
- 7 - 8 X1, X2 starting location and ending location of the load measured from node 1 ($X2 \geq X1 \geq 0$, for concentrated moment only X1 is needed, enter zero for X2). X1 and X2 are normalized distances (ratios) or actual distances, depending on entry 3 (IRL).
- 9 - 10 P1, P2 load values at locations X1 and X2. See notes 1, 2. For concentrated forces and moments, only P1 is needed, enter zero for P2.
- 11 LFN key for follower force.
if IFM (entry 4) = 0 and LFN = 0, forces will be treated as non-follower
if IFM (entry 4) = 0 and LFN = 1, forces will be treated as follower
- 12 ITCRV time-amplitude curve ID (applicable to NLSTATIC, LTRANSIENT and NLTRANSIENT only, see note 3).

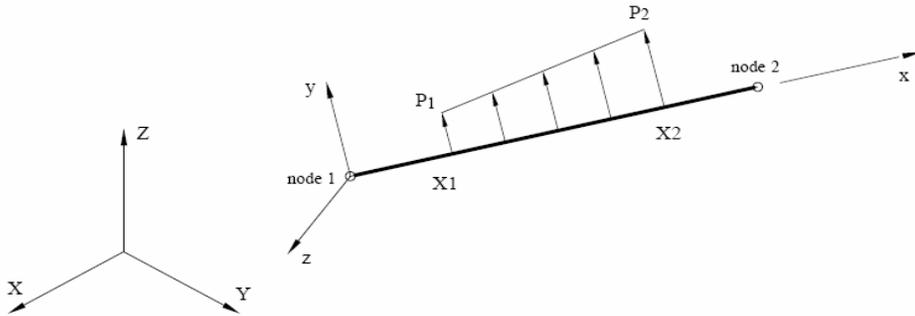


Figure 7.1: Beam element subjected to linearly varying distributed load acting on a segment of the beam, $(X, Y, Z) =$ Global System, $(x, y, z) =$ Beam Local Coordinate System

Notes:

1. If IFM (entry 4) = 0 and $X1$ (entries 7, 8) = $X2$, the applied load is a concentrated force. P2 is not used in this case.
2. The units of the load values P1 and P2 must be consistent with the type of load. P1 and P2 are in units of force per unit length if the applied load is a distributed force. P1 is in unit of force or moment if the applied load is a concentrated force or moment.
3. In nonlinear static and direct transient dynamic analyses, pressure values at any time/load step are computed using the pressure values and the time-amplitude curve (ITCRV) given in beam load card set. If ITCRV variable is not given in the beam load card set, then the time-amplitude curve ID given in the group-ID parameter TCRV is used. A default time-amplitude curve as described in [Section 6.7.4](#) is used when the TCRV parameter is not supplied by the user.

7.3.6 *BODYFORCE Data Group - Body Force Data

Applicable analysis type: STATIC, BUCKLING, NLSTATIC

This data group may be used to define gravity loads, and inertia loads (due to linear, centrifugal, and angular accelerations). Refer to NISA element library for elements for which body forces may be applied.

Group ID card:

*BODYFORCE, TCRV = n ₁ , n ₂ , n ₃

where the parameter TCRV is *only required in nonlinear static analysis*, and n₁, n₂, n₃ are integers designating time-amplitude curve identification numbers defined in the *TIMEAMP data group (see note 4 and 5).

The data in this group consists of two card sets. The first set is always required, and it defines the components of angular velocities and gravity (or linear) acceleration. The second card set is needed only if there are angular accelerations, and/or the point about which the model is rotating is different from the origin of the global system.

Card set 1: Angular velocity, and linear acceleration data, one card

	\$					
Entry No:	1	2	3	4	5	6
Variable:	OMEGAX	OMEGAY	OMEGAZ	ACCELX	ACCELY	ACCELZ
Max char:	10	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	OMEGAX	component of angular velocity in the X direction (radians/time)
2	OMEGAY	component of angular velocity in the Y direction (radians/time)
3	OMEGAZ	component of angular velocity in the Z direction (radians/time)
--- tab (\$) ---		
4	ACCELX	gravity or linear acceleration in X direction, (length/time ²), see note 3.
5	ACCELY	gravity or linear acceleration in Y direction, (length/time ²), see note 3.
6	ACCELZ	gravity or linear acceleration in Z direction, (length/time ²), see note 3.

Card set 2: One card, required only if there are angular accelerations, and/or there is a shift between the point of rotation and the global origin.

	\$					
Entry No:	1	2	3	4	5	6
Variable:	ALPHAX	ALPHAY	ALPHAZ	XO	YO	ZO
Max char:	10	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	ALPHAX	component of angular acceleration in the X direction (<i>radians/time²</i>)
2	ALPHAY	component of angular acceleration in the Y direction (<i>radians/time²</i>)
3	ALPHAZ	component of angular acceleration in the Z direction (<i>radians/time²</i>)
--- tab (\$) ---		
4	XO	X-coordinate of the point about which the model is rotating
5	YO	Y-coordinate of the point about which the model is rotating
6	ZO	Z-coordinate of the point about which the model is rotating

Notes:

1. For axisymmetric elements with non-axisymmetric loading (NKTP = 34, 37), the body forces must be axisymmetric.
2. Use the right hand rule to determine the direction of positive angular velocities and accelerations.
3. The *sign* of linear acceleration (excluding gravity) should be reversed. Acceleration due to gravity should, however, not be reversed in sign.
4. In nonlinear static analysis, body force values at any time/load step are computed using the force values and the time-amplitude curves in the group-ID parameter TCRV. The curve IDs n1, n2 and n3 in the TCRV parameter are associated with the OMEGA, ACCEL and ALPHA values, respectively. If TCRV parameter is not given or if any of the values n1, n2 or n3 is given as zero then a default time-amplitude curve as described in [Section 6.7.4](#) is used.

Analysis Data

Boundary Conditions and Loading

5. If the time-amplitude curve ID, n1, (in TCRV parameter) for OMEGA is negative then the amplitude values (defined in the referenced time-amplitude curve) correspond to the square of OMEGA values. This option is useful to provide linear increments in loading (i.e., proportional to increments in ω^2) rather than load increments proportional to increments in ω .

7.3.7 *NDTEMPER Data Group - Nodal Temperature Data

Applicable analysis type: STATIC, BUCKLING, NLSTATIC

This data group may be used to specify temperatures at the nodes. Thermal loads are computed using these temperatures and a reference temperature (TSFRE) given in *LDCASE group (or in *EVENT for NLSTATIC). Refer to NISA element library for elements for which thermal loads may be applied (see note 2).

Group ID card:

*NDTEMPER, TCRV = n, FN = file name, STEP = m

where,

- (i) The parameter TCRV = n is only required in nonlinear static analysis, 'n' is an integer designating a time-amplitude curve identification number defined in *TIMEAMP data group (see note 3)
- (ii) The parameters FN = file name and STEP = m are required if input data for this group is to be accessed from a steady state or a transient NISA II Heat Transfer run. The file name should represent the name of the file 39 of NISA II Heat Transfer analysis (ASCII format) and m is an integer (maximum characters of six) designating the time step identification number on that file to be accessed for the present load case (see note 4)

Nodal temperature card set:

Entry No:	1	2	3	4	5	6
Variable:	NODE	LABEL	TVAL	LASTND	NODINC	ITCRV
Max char:	6	4	12	6	6	6

Entry Variable Description

- 1 NODE node number
- 2 LABEL label, must be TEMP
- 3 TVAL temperature value
- 4 LASTND last node of a range of node numbers to be assigned the same temperature
- 5 NODINC positive increment for the range of node numbers
- 6 ITCRV time-amplitude curve ID (applicable to NLSTATIC only, see note 3)

Notes:

1. Nodes not referenced in this data group will be assigned a temperature value of TSFRE which is the stress-free temperature defined in *LDCASE data group.
2. For axisymmetric elements with non axisymmetric loading (NKTP = 34, 37), the thermal loads are assumed to be constant in the circumferential direction.
3. In the nonlinear analysis, nodal temperature values at any time/load step are computed using the temperature values and the time-amplitude curve (ITCRV) given in nodal temperature card set. Unlike other analysis data groups, use of the default time-amplitude curve is different in this group, and is explained below.

If the group-ID parameter TCRV is given, it will be used as the time-amplitude curve ID for the card sets without ITCRV variable. If the TCRV parameter and ITCRV variable in all the nodal temperature card sets are *not* given, then a default time-amplitude curve is *not used* in computation of the nodal temperature at a step. Instead, the nodal temperature is linearly interpolated between temperatures in previous event and current event. This option is useful if present analysis is using results of heat transfer analysis.

If ITCRV is not given and TCRV variable is given in one or more nodal temperature card sets, a default time-amplitude curve as described in [Section 6.7.4](#) is used for the card sets without time-amplitude curve ID.

4. To access the data from heat transfer analysis, the user is required to save file 39 during the NISA II heat transfer analysis run.
5. If temperature data for a particular node is redefined, either in the primary or secondary input file, the last specified value is used.

7.3.8 *NDTEMPDIF Data Group - Nodal Temperature Differences

Applicable analysis type: STATIC, BUCKLING, NLSTATIC

This data group may be used to specify the difference in temperature between the top and bottom surfaces of 3-D beam and shell elements. Refer to NISA element library for specific beam and shell elements that can have nodal temperature difference input.

Group ID card:

*NDTEMPDIF, TCRV = n, FN = file name, STEP = m
--

where,

- (i) the parameter TCRV = n is only required in nonlinear static analysis, *n* is an integer designating a time-amplitude curve identification number defined in *TIMEAMP data group (see note 2)
- (ii) the parameters FN = file name and STEP = m are required if input data for this group is to be accessed from a steady state or a transient NISA II Heat Transfer run. The file name should represent the name of the file 39 of NISA II Heat Transfer analysis (ASCII format) and m is an integer (maximum characters of six) designating the time step identification number in FILENM to be accessed for the present load case (see note 3).

Nodal temperature difference card set:

Entry No:	1	2	3	4	5	6
Variable:	NODE	LABEL	DELTAT	LASTND	NODINC	ITCRV
Max char:	6	4	12	6	6	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NODE	node number
2	LABEL	label, must be TEMP
3	DELTAT	temperature difference, defined as the top surface temperature minus the bottom surface temperature. (Top and bottom surfaces are defined by the element node numbering sequence, using the right-hand rule.) Refer to element library for the face numbers convention
4	LASTND	last node of a range of node numbers to be assigned the same DELTAT

Analysis Data

Boundary Conditions and Loading

- 5 NODINC positive increment for the range of node numbers
- 6 ITCRV time-amplitude curve ID (applicable to NLSTATIC only, see note 2)

Notes:

1. Nodes not referenced in this data group will be assigned a temperature difference value of zero.
2. In nonlinear analysis, nodal temperature values at any time/load step are computed using the temperature values and the time-amplitude curve (ITCRV) given in nodal temperature difference card set. Unlike other analysis data groups, use of the default time-amplitude curve is different in this group, and is explained below.

If the group-ID parameter TCRV is given, it will be used as the time-amplitude curve ID for the card sets without ITCRV variable. If the TCRV parameter and ITCRV variable in all the nodal temperature difference card sets are NOT given then a default time-amplitude curve is *not used* in computation of the nodal temperature at a step. Instead, the nodal temperature is linearly interpolated between temperatures in previous event and current event. This option is useful if present analysis is using results of heat transfer analysis.

IF TCRV is not given and ITCRV variable is given in one or more nodal temperature card sets, a default time-amplitude curve as described in [Section 6.7.4](#) is used for the card sets without time-amplitude curve ID.

3. To access the data from heat transfer analysis, the user is required to save file 39 during the NISA II heat transfer analysis run.
4. If temperature data for a particular node is redefined, either in the primary or secondary input file, the last specified value is used.

7.3.9 *INITIAL Data Group - Initial Condition Data

Applicable analysis types: LTRANSIENT, NLTRANSIENT

This data group is optional and may be used to specify initial displacements, velocities and accelerations at specific nodes of the structure for direct transient dynamic analysis.

Group ID card: *INITIAL

Initial condition card set:

Entry No:	1	2	3	4	5
Variable:	NODE	LABEL	VALUE	LASTND	INC
Max char:	6	4	12	6	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NODE	node number
2	LABEL	label for initial condition input (see the list of valid labels in Table 7.3)
3	VALUE	value for the initial condition
4	LASTND	last node of a range of node numbers to be assigned the same VALUE of the initial condition
5	INC	positive increment for the range of node numbers

Notes:

1. Initial conditions specified in this data group will be applied to the nodes of the structure at the starting time for the analysis specified.
2. Nodes not specified in this data group will be assigned a zero value for the initial displacement, velocity or acceleration.
3. If a local displacement coordinate system is defined at a particular node, then the initial conditions must be described in the local coordinate system, otherwise all initial displacements, velocities or accelerations are in the global coordinate system.
4. Additional cards may be provided, if necessary to specify initial displacements, velocities or accelerations at other nodes.

5. Initial conditions prescribed for slave DOFs and constrained DOFs will be ignored, and a warning message will be given. The corresponding initial conditions for slave DOFs will be calculated automatically by the program.
6. This data group is ignored in Restart 5 run.

Table 7.3: List of valid labels for *INITIAL group

UX	Displacement in X direction
UY	Displacement in Y direction
UZ	Displacement in Z direction
ROTX	Rotation about X axis
ROTY	Rotation about Y axis
ROTZ	Rotation about Z axis
VX	Velocity in X direction
VY	Velocity in Y direction
VZ	Velocity in Z direction
OMGX	Rotational velocity about X axis
OMGY	Rotational velocity about Y axis
OMGZ	Rotational velocity about Z axis
ACCELX	Accelerations in X direction
ACCELY	Accelerations in Y direction
ACCELZ	Accelerations in Z direction
ALPHAX	Rotational accelerations about X axis
ALPHAY	Rotational accelerations about Y axis
ALPHAZ	Rotational accelerations about Z axis

7.3.10 *TBODYFORCE -Time Dependent Body Force Data Group

Applicable analysis type: LTRANSIENT, NLTRANSIENT

This data group may be used to define gravity loads, and inertia loads (due to translation and/or rotational motions) for direct transient dynamic analysis.

Group ID card: ***TBODYFORCE**

Entry No:	1	2	3
Variable:	LABEL	VALUE	ICURV
Max char:	6	10	6

Entry Variable Description

1	LABEL	<p>component label of body force</p> <p>Allowable labels are:</p> <p>OMEGAX : angular velocity in X direction</p> <p>OMEGAY : angular velocity in Y direction</p> <p>OMEGAZ : angular velocity in Z direction</p> <p>ACCELX : linear acceleration in X direction</p> <p>ACCELY : linear acceleration in Y direction</p> <p>ACCELZ : linear acceleration in Z direction</p> <p>X0 : X-coordinate of the point of rotation</p> <p>Y0 : Y-coordinate of the point of rotation</p> <p>Z0 : Z-coordinate of the point of rotation</p> <p>ALPHAX : angular acceleration in X direction</p> <p>ALPHAY : angular acceleration in Y direction</p> <p>ALPHAZ : angular acceleration in Z direction</p>
2	VALUE	scalar value of the corresponding component
3	ICURV	time-amplitude curve identification number defined in the *TIMEAMP data group (see note 1).

Analysis Data

Boundary Conditions and Loading

Note:

1. In linear direct transient dynamic analysis, the values specified in this data group are applied in increments. The increments are calculated using the time amplitude curve defined in the *TIMEAMP data group. If the curve ID is omitted or blank (i.e., ICURV = 0), the default time-amplitude curve is used.

7.3.11 *L1 data group -Alternate Form for Specifying Pressure Loads

Applicable analysis type: STATIC, BUCKLING

This data group may be used to define pressure loads on any face (or edge) of an element. Refer to NISA element library for elements which may have pressure loading. Users are advised to use the standard form of pressure definition, the *PRESSURE data group, since this data group (*L1) will not be supported in future versions.

Group ID card:

*L1

Pressure force card set:

						\$						\$
Entry No:	1	2	3	4	5	6	7	8	9 - 11	12	13	
Variable:	NELID	NSRL	LASTEL	INC	IDFACE	NCARDS	P1	P2	...	P6	LFN	
Max char:	5	1	5	4	1	1	10	10	...	10	3	

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NELID	element number
2	NSRL	not used, enter zero or leave blank.
3	LASTEL	last element of a range of elements with the same pressure loading. Enter 0 if this card defines pressure loading on a single element.
4	INC	positive increment for the range of elements. Enter 0 or 1 if loading just a single element.
5	IDFACE	face number on which the pressure acts (see NISA element library for face number convention).
6	NCARDS	number of additional cards required to complete the pressure data for this face. (NCARDS = 0 unless the face has more than 6 nodes.)
--- tab (\$) ---		
7 - 12	P1 - P6	pressure values at all nodes of the face. If the face has more than 6 nodes, continue on additional cards as necessary, six entries per card. Additional cards must begin with a tab character (\$).
--- tab (\$) ---		

Analysis Data

Boundary Conditions and Loading

- 13 LFN Fourier distribution index, only required for axisymmetric elements subjected to non-axisymmetric loading; NKTP = 34, 37 elements; enter zero otherwise.

Notes:

1. Nodal pressure intensities must be specified at all nodes on a face. If the pressure is constant on a face, the horizontal repetition character (/) may be used to repeat the pressure value specified for the first node.
2. A positive pressure value corresponds to a surface traction normal to and acting towards the face.

7.3.12 *MASTER - Master Degrees of Freedom for Guyan Reduction

Applicable analysis type: EIGENVALUE, BUCKLING

This data group is used to specify master degrees of freedom for Guyan reduction.

Group ID card: *MASTER

Master degrees of freedom card set:

	\$						
Entry No:	1	2	3	4	5	6 - 8	9
Variable:	NODE	LABEL	LASTND	NODINC	LABEL	...	LABEL
Max char:	5	4	6	6	4	...	4

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NODE	node number
2	LABEL	label for degree of freedom Allowable labels are: UX, UY, UZ ROTX, ROTY, ROTZ UXYZ RXYZ ALL
3	LASTND	last node
4	NODINC	node increment for a range of nodes
5-9	LABEL	additional labels to be chosen as masters for Guyan reduction

Analysis Data

Boundary Conditions and Loading

Notes:

1. If a local displacement coordinate system is defined at a node, the displacement labels (UX...ROTZ) refer to translations and rotations in the local system.
2. If the number of master degrees of freedom specified in this data group is less than the value specified in *EIGCNTL data, the remaining master degrees of freedom are selected automatically by the program. Otherwise, all the master degrees specified in this data group are used.

7.3.13 *EDGFORCE - Edge Force for Shell Element Data Group

Applicable analysis type: STATIC, BUCKLING, NLSTATIC, LTRANSIENT, NLTRANSIENT

This data group may be used to define distributed and concentrated loads along the edges of the 3D shell elements (NKTP = 20, 32, 33, 41) at arbitrary locations. The applied loads can be tangent to the edge, normal to the edge, or in the direction of a given vector. The loads can be a force or a moment. The distributed loads can vary uniformly or linearly.

Group ID card: *EDGFORCE, TCRV = n

where the parameter 'TCRV = n' is required in nonlinear static, linear direct transient dynamic, and nonlinear direct transient dynamic analysis, 'n' is an integer designating a time-amplitude curve identification number defined in the *TIMEAMP data group (see note 3).

Edgeforce Card Set 1:

Entry No:	1	2	3	4	5	6	7	8
Variable	NELID	NODE1	NODE2	IVECT	IRL	IFM	ILFN	ITCRV
Max Char:	8	8	8	8	1	1	1	8

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NELID	Element number
2	NODE1	The first corner node of the edge
3	NODE2	The second corner node of the edge
4	IVECT	Definition of load direction = -1 – the applied load is in the direction along the edge from node1 to node2 = -2 – the applied load is in the direction of the outward normal of the edge which is tangent to the mid-surface of the shell element = -3 – the applied load is the direction of the normal of the shell surface. > 0 – vector ID number. The applied load is in the direction of the vector whose vector ID = IVECT in *VECTORS data group

Analysis Data

Boundary Conditions and Loading

- 5 IRL Form of load location definition
 = 0 – the locations of load (X1 and X2) are defined in terms of a ratio of the length of edge from node 1.
 = 1 – the locations of load (X1 and X2) are defined by the actual distance from node 1.
- 6 IFM Type of applied load
 = 0 – the applied load is a concentrated or distributed force
 = 1 – the applied load is a concentrated or distributed moment
- 7 LFN Key for follower force
 = 0 – if IFM = 0 and LEN = 0, force will be treated as a non-follower force.
 = 1 – if IFM = 0 and LEN = 1, force will be treated as a follower force.
- 8 ITCRV Time-amplitude curve ID (applicable to NLSTATIC, LTRANSIENT, or NLTRANSIENT)

Card Set 2:

Entry No.	1	2	3	4
Variable:	X1	X2	P1	P2
Max Char	10	10	10	10

Entry Variable Description

- 1 X1 Distance of the starting location of the distributed load (or the location of the concentrated load) from NODE1. X1 can be specified as the ratio of length (IRL = 0) or the actual distance (IRL = 1) from NODE1.
- 2 X2 Distance of the ending location of the distributed load from NODE1. X1 can be specified as the ratio of length (IRL = 0) or the actual distance (IRL = 1) from NODE1. For concentrated load, X2 can be given as X1 or zero.
- 3 P1 The load value at location X1
- 4 P2 The load value at location X2. For concentrated loads, only P1 is needed.

Notes:

1. If $IFM = 0$ and the load is concentrated force, P2 is not used in this case.
2. The units of the load values P1 and P2 must be consistent with the type of load. P1 and P2 are in units of force or moment per unit length for the distributed loads. For concentrated loads, P1 should be in units of force or moment.
3. In nonlinear static and direct transient dynamic analyses, load value at any time step are computed using the load values (P1 and P2) times the value of the time-amplitude curve (TCRV) at that time step. If ITCRV variable is not given in EDGFORCE data set 1, then the time-amplitude curve ID given in the group-ID parameter TCRV is used. A default time-amplitude curve as described in [Section 6.7.4](#) is used when the TCRV parameter is not supplied by the user.

7.3.14 *ECDENSITY data group – Material properties for piezoelectric analyses

Applicable analysis type: PZSTATIC, PLTRANSIENT

This data group is used to define electric charge density on faces of elements in piezoelectric analysis. Refer to NISA element library for elements for which electric charge density can be applied.

Group ID card

*ECDENSITY, TCRV = n

where the parameter ‘TCRV = n’ is required in direct transient dynamic analysis, ‘n’ is an integer designating a time-amplitude curve identification number defined in the *TIMEAMP data group.

Input data for electric charge density is given in two card sets. The first card is always required. The second card is optional. The first card has flags identifying element numbers, face number and electric charge density related parameters. The second card contains electric charge density values for all the nodes on the specified elements’ surface. For uniform electric charge density on an element face, a single value can be given in the first card for all the nodes without the second card.

Card set 1: one card; always required

Entry No:	1	2	3	4	5	6	7
Variable:	NELID	LASTEL	INC	IDFACE	NCARDS	UECHR	ITCRV
Max char:	6	6	6	1	1	10	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NELID	element number
2	LASTEL	last element of a range of elements with the same electric charge density loading. Enter 0 or blank if this card defines loading on a single element.
3	INC	positive increment for the range of elements. Enter 0, 1 or blank if loading just a single element.
4	IDFACE	face number on which the electric charge density loading acts (see element library for face numbering conventions).

- 5 NCARDS number of cards in card set 2 required to complete the electric charge density data for the face. If NCARDS = 0, uniform charge density value given in 'UECHR' is used for all nodes, and the second card set is not required.
- 6 UECHR uniform electric charge density value for all nodes of the face (used if NCARD=0). UECHR value is ignored if NCARD>0.
- 7 ITCRV time-amplitude curve ID (applicable to PLTRANSIENT)

Card set 2: required only if NCARD > 0

Entry No:	1	2	3	4	5	6	7	8
Variable:	EC1	EC2	EC3	EC4	EC5	EC6	EC7	EC8
Max char:	10	10	10	10	10	10	10	10

Entry Variable Description

- 1-8 EC1-EC8 electric charge density values at all nodes of the face. If the face has more than 8 nodes, continue on additional cards as necessary. Number of cards used to define the charge density must be equal to NCARDS.

Notes:

1. A positive electric charge density value corresponds to an electric charge into the face.
2. In direct transient dynamic analysis, electric charge density value at any time is computed using the charge density value at the nodes times the amplitude value at the time from the time amplitude curve (ITCRV). If ITCRV variable is not given in card set 1, then the time-amplitude curve ID given in the group-ID parameter TCRV is used. A default time-amplitude curve as described in [Section 6.7.4](#) is used when the TCRV parameter is not supplied by the user.

7.4 Boundary Condition and Loading for Heat Transfer

7.4.1 *INITEMP Data Group - Initial Temperatures

Applicable analysis types: THEAT

This data group may be used in transient heat transfer analysis to specify initial temperature values at nodes. If all nodes in the structure have the same initial temperature, that temperature may be entered as initial value in the executive commands and this data group is not required.

Group ID card:

*INITEMP

Initial temperature card set:

Entry No:	1	2	3	4				
Variable:	<table border="1"><tr><td>NODE</td></tr></table>	NODE	<table border="1"><tr><td>TEMP</td></tr></table>	TEMP	<table border="1"><tr><td>LASTND</td></tr></table>	LASTND	<table border="1"><tr><td>NODINC</td></tr></table>	NODINC
NODE								
TEMP								
LASTND								
NODINC								
Max char:	6	12	6	6				

<u>Entry Variable</u>	<u>Description</u>
1 NODE	node number
2 TEMP	initial temperature value.
3 LASTND	last node of a range of nodes with the same initial temperature
4 NODINC	increment for the range of nodes

Note:

1. Initial temperature values specified in this data group override the initial temperature value specified in the executive commands.

7.4.2 *SPTEMP Data Group - Specified Nodal Temperature Boundary Conditions

Applicable analysis types: SHEAT, THEAT

This data group may be used in heat transfer analyses to specify temperature boundary conditions at any node.

Group ID card:

*SPTEMP

Entry No:	1	2	3	4	5
Variable:	NODE	TEMP	LASTND	NODINC	IDCRV1
Max char:	6	12	6	6	6

Specified temperature card set:

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NODE	node number
2	TEMP	specified temperature value.
3	LASTND	last node of a range of nodes with the same specified temperature
4	NODINC	increment for the range of nodes
5	IDCRV1	time amplitude curve identification number in *TIMEAMP data group. Enter zero if specified nodal temperatures are not time dependent, see note 1.

Note:

1. Specified temperature value at any node becomes time dependent whenever a nonzero value of IDCRV1 is used. The IDCRV1 parameter refers to a time amplitude curve identification number defined in the *TIMEAMP data group. The value of the specified temperature at any time is the temperature value specified here (TEMP) times the scaling factor given in the curve at that time.

7.4.3 *CFLUX Data Group - Specified Concentrated Nodal Fluxes

Applicable analysis types: SHEAT, THEAT

This data group may be used in heat transfer analyses to specify concentrated nodal heat flux values at nodes.

Group ID card:

*CFLUX

Concentrated flux card set:

Entry No:	1	2	3	4	5	6
Variable:	NODE	FLUX	LASTND	INC	IDCRV1	IDCRV2
Max char:	6	12	6	6	6	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NODE	node number
2	FLUX	value of the nodal flux (heat/time)
3	LASTND	last node of a range of node numbers to be assigned the same value of nodal flux
4	INC	increment for the range of nodes
5	IDCRV1	temperature dependency curve identification number defined in *TEMPFN data group. Enter zero if heat flux is not temperature dependent, see note 1.
6	IDCRV2	time amplitude curve identification number defined in *TIMEAMP data group (used for THEAT only). Enter zero if nodal fluxes are not time dependent.

Note:

1. Specified nodal flux values at any node become temperature and/or time dependent whenever a nonzero value of IDCRV1 and/or IDCRV2 is used, respectively. The value of the specified nodal flux at any instant of time is the nodal flux value specified here (FLUX) times the scaling factor given in the temperature curve times the scaling factor given in the time curve.

7.4.4 *DFLUX Data Group - Specified Distributed Heat Fluxes

Applicable analysis types: SHEAT, THEAT

This data group may be used in heat transfer analyses to specify distributed heat flux on elements.

Group ID card:

*DFLUX

Free format data for a typical distributed flux input consists of two card sets

Card Set 1: Element identification card (one card)

Entry No:	1	2	3	4	5	6	7
Variable:	NELID	LASTEL	INC	IDFACE	KEY	IDCRV1	IDCRV2
Max char:	6	6	6	1	2	6	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NELID	element number
2	LASTEL	last element of a range of elements with the same flux.
3	INC	increment for the range of elements
4	IDFACE	face number on which the distributed heat flux acts (see NISA heat transfer element library for face numbering convention)
5	KEY	key for the flux values to be entered on card set 2; = -1 - constant flux, only the first entry on card set 2 is required = n - where, n is an integer = 0, 1, 2,... up to 8*(n+1) nodal flux values may be given on card set 2. Eight values per card.
6	IDCRV1	temperature dependency curve identification number defined in *TEMPFN data group. Enter zero if heat flux is not temperature dependent, see note 1.
7	IDCRV2	time amplitude curve identification number defined in *TIMEAMP data group. Enter zero if heat flux is not time dependent, see note 1.

Card Set 2: Heat flux card (more than one card if face has more than 8 nodes)

Entry No:	1	2	3	4	5	6	7	8
Variable:	P1	P2	P3	P4	P5	P6	P7	P8
Max char:	10	10	10	10	10	10	10	10

Entry Variable Description

1 - 8 P1 - P8 heat flux values at all nodes of the face, heat/(time.area). If the face has more than 8 nodes, continue on additional cards as necessary, eight entries per card.

Note:

1. Specified distributed heat flux on any element become temperature dependent and/or time dependent whenever a nonzero value of IDCRV1 and/or IDCRV2 is entered, respectively. The value of the specified distributed heat flux on an element face at any instant is the distributed heat flux value specified in this group times the scale factor given on the temperature curve times the scale factor given on the time amplitude curve.

7.4.5 *ELHEATGEN Data Group - Specified Internal Heat Generation at Elements

Applicable analysis types: SHEAT, THEAT

This data group may be used in heat transfer analyses to specify internal heat generation at elements.

Group ID card:

*ELHEATGEN

Internal heat generation card set:

	\$					
Entry No:	1	2	3	4	5	6
Variable:	NELID	HTGR	LASTEL	INCR	IDCRV1	IDCRV2
Max char:	6	12	6	6	6	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NELID	element number
2	HTGR	heat generation rate, heat/(time. volume)
3	LASTEL	last element of a range of elements with the same heat generation
4	INCR	increment for the range of elements
--- tab (\$) ---		
5	IDCRV1	temperature dependency curve identification number defined in *TEMPFN data group. Enter zero if heat generation rate is not temperature dependent, see note 1.
6	IDCRV2	time amplitude curve identification number defined in *TIMEAMP data group. Enter zero if heat generation is not time dependent, see note 1.

Note:

1. Specified heat generation rate at any element becomes temperature dependent and/or time dependent whenever a nonzero value of IDCRV1 and/or IDCRV2 is entered, respectively. The value of the specified element heat generation rate at any instant is the heat generation rate specified here (HTGR) times the scale factor given on the temperature curve times the scale factor given on the time amplitude curve.

7.4.6 *NDHEATGEN Data Group - Specified Internal Heat Generation at Nodes

Applicable analysis types: SHEAT, THEAT

This data group may be used in heat transfer analyses to specify internal heat generation at nodes.

Group ID card:

*NDHEATGEN

Internal heat generation card set:

Entry No:	1	2	3	4	5	6
Variable:	NODE	HTGR	LASTND	NODINC	IDCRV1	IDCRV2
Max char:	6	12	6	6	6	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NODE	node number
2	HTGR	heat generation rate, heat/(time. volume), see note 1
3	LASTND	last node of a range of nodes with the same heat generation
4	NODINC	increment for the range of nodes
--- tab (\$) ---		
5	IDCRV1	temperature dependency curve identification number defined in *TEMPFN data group. Enter zero if heat generation rate is not temperature dependent, see note 2.
6	IDCRV2	time amplitude curve identification number defined in *TIMEAMP data group. Enter zero if heat generation is not time dependent, see note 2.

Notes:

1. Nodal heat generation values given in this group are per unit volume. These values are integrated over the volume of all elements connected to the nodes given in this group.
2. Specified heat generation rate at any node becomes temperature dependent and/or time dependent whenever a nonzero value of IDCRV1 and/or IDCRV2 is entered, respectively. The value of the specified nodal heat generation rate at any instant is the heat generation rate specified here (HTGR) times the scale factor given on the temperature curve times the scale factor given on the time amplitude curve.

7.4.7 *CONVBC Data Group - Convection Boundary Conditions

Applicable analysis types: SHEAT, THEAT

This data group may be used in heat transfer analyses to specify convective boundary condition on an element.

Group ID card:

*CONVBC

A typical convection boundary condition data on an element consists of two card sets.

Card Set 1: Element identification card (one card)

Entry No:	1	2	3	4	5	6	7	8
Variable:	NELID	LASTEL	INC	IDFACE	KEY	IDCRV1	IDCRV2	IDCRV3
Max char:	6	6	6	1	2	6	6	6

Entry Variable Description

- 1 NELID element number
- 2 LASTEL last element of a range of elements with the same film coefficient.
- 3 INC increment for the range of elements
- 4 IDFACE face number on which convection boundary condition acts (see NISA heat transfer element library for face numbering convention)
- 5 KEY key for film coefficient and ambient temperature data to be entered on card set 2;
= -1 - constant film coefficient, only the first two entries are required on card set 2.
= n - where, n is an integer = 0, 1, 2,... up to 4*(n+1) pairs of film coefficients and temperatures may be given on card set 2. Four pairs per card.
- 6 IDCRV1 temperature dependency curve identification number (for film coefficient) defined in *TEMPFN data group. Enter zero if film coefficient is not temperature dependent, see note 1.

- 7 IDCRV2 time amplitude curve identification number (for film coefficient) defined in *TIMEAMP data group. Enter zero if film coefficient is not time dependent, see note 1.
- 8 IDCRV3 time amplitude curve identification number (for ambient temperature) defined in *TIMEAMP data group. Enter zero if ambient temperature is not time dependent, see note 2.

Card set 2: Film coefficient and ambient temperature data (more than one card if face has more than 4 nodes).

Entry No:	1	2	3	4	5	6	7	8
Variable:	H1	TEMP1	H2	TEMP2	H3	TEMP3	H4	TEMP4
Max char:	10	10	10	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	H1	film coefficient for the first node of the element face, heat/(time. area. degree).
2	TEMP1	temperature for the film coefficient H1
3 - 8	--	repeat pairs of film coefficient (Hi) and temperature (TEMPi) for nodes 2, 3,... of the element face. Eight entries (representing 4 nodes) per card.

Analysis Data

Boundary Condition and Loading for Heat Transfer

Notes:

1. The film coefficient becomes temperature dependent and/or time dependent whenever a nonzero value of IDCRV1 and/or IDCRV2 is entered, respectively. The value of the film coefficient at any instant is the film coefficient value specified here times the scale factor given on the temperature curve times the scale factor given on the time amplitude curve.
2. Specified ambient temperature values become time dependent whenever a nonzero value of IDCRV3 is entered. The value of the ambient temperature at any instant of time is the ambient temperature value specified here times the scale factor given on the time amplitude curve.

7.4.8 *RADBC Data Group - Radiation Boundary Conditions

Applicable analysis types: SHEAT, THEAT

This data group may be used in heat transfer analyses to specify radiation boundary condition on any face of an element.

Group ID card:

*RADBC, SIGMA = val

where, 'val' is the Stefan-Boltzmann constant, heat/(time. area. degree⁴).

The free format data for a typical radiation boundary condition on an element face consists of two card sets.

Card Set 1: Element identification card (one card)

Entry No:	1	2	3	4	5	6	7	8
Variable:	NELID	LASTEL	INC	IDFACE	KEY	IDCRV1	IDCRV2	IDCRV3
Max char:	6	6	6	1	2	6	6	6

Entry	Variable	Description
1	NELID	element number
2	LASTEL	last element of a range of elements with the same radiation boundary condition.
3	INC	increment for the range of elements
4	IDFACE	face number on which radiation boundary condition acts (see NISA heat transfer element library for face numbering convention)
5	KEY	key for emissivity and ambient temperature data to be entered on card set 2; = -1 - constant emissivity, only the first two entries are required on card set = n - where, n is an integer = 0, 1, 2,... up to 4*(n+1) pairs of emissivity and temperature may be given on card set 2. Four pairs per card.

Analysis Data

Boundary Condition and Loading for Heat Transfer

- 6 IDCRV1 temperature dependency curve identification number (for emissivity) defined in *TEMPFN data group. Enter zero if emissivity is not temperature dependent, see note 1
- 7 IDCRV2 time amplitude curve identification number (for emissivity) defined in *TIMEAMP data group. Enter zero if emissivity is not time dependent, see note 1.
- 8 IDCRV3 time amplitude curve identification number (for ambient temperature) defined in *TIMEAMP data group. Enter zero if ambient temperature is not time dependent, see note 2.

Card Set 2: Emissivity and ambient temperature data (more than one card if face has more than 4 nodes)

Entry No:	1	2	3	4	5	6	7	8
Variable:	E1	TEMP1	E2	TEMP2	E3	TEMP3	E4	TEMP4
Max char:	10	10	10	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
--------------	-----------------	--------------------

- | | | |
|-------|-------|---|
| 1 | E1 | emissivity for the first node of the element face, (dimensionless). |
| 2 | TEMP1 | ambient temperature for the emissivity E1 |
| 3 - 8 | -- | repeat pairs of emissivity (Ei) and ambient temperature (TEMPi) for nodes 2, 3,...of the element face. Eight entries (representing 4 nodes) per card. |

Notes:

1. The emissivity becomes temperature dependent and/or time dependent whenever a nonzero value of IDCRV1 and/or IDCRV2 is entered, respectively. The value of the emissivity at any instant is the emissivity value specified here times the scale factor given on the temperature curve times the scale factor given on the time amplitude curve.
2. Specified ambient temperature values becomes time dependent whenever a nonzero value of IDCRV3 is entered. The value of the ambient temperature at any instant of time is the ambient temperature value specified here times the scale factor given on the time amplitude curve.

7.4.9 *RADSURFACE Data Group - Surface Radiation Data

Applicable analysis types: SHEAT, THEAT

This data group may be used in heat transfer analysis to specify surface radiation data on any face of an element.

Group ID card: *RADSURFACE

Surface radiation card set:

Entry No:	1	2	3	4	5	6	7	8	9
Variable:	NELID	LASTEL	INC	IDFACE	EMISS	TAMB	IDCRV1	IDCRV2	NDIR
Max char:	6	6	6	1	12	12	6	6	2

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NELID	element number
2	LASTEL	last element of a range of elements with same surface radiation data
3	INC	increment for the range of elements
4	IDFACE	face number on which surface radiation data is specified (see NISA heat transfer element library for face numbering convention)
5	EMISS	emissivity of the radiating surface
6	TAMB	ambient temperature
7	IDCRV1	temperature dependency curve identification number (for emissivity) defined in *TEMPFN data group. Enter zero if emissivity is not temperature dependent (see note 1)
8	IDCRV2	time amplitude curve identification number (for temperature of surroundings) defined in *TIMEAMP data group. Enter zero if ambient temperature is not time dependent (see note 2).
9	NDIR	Direction of normal 0: outward (default) 1: inward

Analysis Data

Boundary Condition and Loading for Heat Transfer

Notes:

1. The emissivity becomes temperature dependent whenever a nonzero value of IDCRV1 is entered. The value of the emissivity at any instant is the emissivity value specified here times the scale factor given on the temperature curve.
2. Specified ambient temperature values becomes time dependent whenever a nonzero value of IDCRV2 is entered. The value of the ambient temperature at any instant of time is the ambient temperature value specified here times the scale factor given on the time amplitude curve.
3. The user should specify either *RADSURFACE or *RADBC card on an element face. If both the cards are specified, both the cards will be taken into account. Hence, effect of radiation on temperature distribution will be accounted for twice.
4. If a surface does not interact with other surfaces in surface radiation sense, the user is advised to make use of radiation boundary condition (*RADBC card). This will lead to reduction in disk space requirement as well as cpu time required to solve the problem.
5. If view factors with obstructions are to be evaluated, then all the radiation surfaces are taken as potential obstructors. Therefore, all possible obstructors must also be specified as radiation surfaces.
6. If radiating surfaces form a complete enclosure, ambient temperature is meaningless.

7.5 Output Control Data

7.5.1 *EIGOUT Data Group - Eigenvalue Output Control

Applicable analysis types: EIGENVALUE, BUCKLING

This data group defines some computation keys for output options. It must be the first data group in the output control block. There is no limit on the number of output control blocks.

Each output control block must start with this data group, followed by all other pertinent data groups which define printout control for this output control block.

Group ID card: *EIGOUT, ID = n

where, n is a unique output control block identification number (up to 6 digits). This ID can be referenced by other data groups like *MODEOUT and *PRINTCNTL. If the ID is not specified it will default to the previous output control block ID plus 1. In the case of the first output control block it will default to 1.

Eigenvalue Output Control Card Set:

	\$									
Entry No:	1	2	3	4	5	6	7	8	9	10
Variable	KELFR	KRCTN	KSTR	KSTN	LQ1	LQ2	LQ7	KPSEU	TOLPF	CFREQ
Max char	2	2	2	2	2	2	2	2	10	10

Entry Variable Description

- | | | |
|---|-------|---|
| 1 | KELFR | element modal strain energy and modal internal forces key (see note 1)
=0 - off
=1 - on |
| 2 | KRCTN | modal reaction force calculation key (see notes 2, 3)
=0 - off
=1 - on |
| 3 | KSTR | element modal stress calculation key (see notes 2, 4, 5) |

Analysis Data

Output Control Data

- = 0 - no element stress calculation
 - = 1 - calculate stresses at element Gauss points
 - = 2 - calculate stresses at element node points
 - = 3 - calculate stresses at Gauss points and node points
 - = 4 - calculate the stresses at element centroid only
 - = 5 - calculate stresses at element Gauss points and centroid
 - = 6 - calculate stresses at element node points and centroid
 - = 7 - calculate stresses at Gauss points, node points and centroid
- 4 KSTN KSTN element centroidal strain calculation key
- =0 - off
 - =1 - on
- tab (\$) ---
- 5 LQ1 element modal stress printout key (used only if KSTR >0). This key selects the type of element stress printout.
- = -1 - suppress all element stresses printout
 - = 0 - print all element stresses (local, global, principal and equivalent stresses)
 - = 1 - print global components of stresses for each element
 - = 2 - print principal stresses and equivalent stresses (e.g., von Mises stress) for each element
 - = 3 - print element local stresses only (for shell elements or orthotropic materials) for each element
 - = 4 - print global stresses, principal stresses, and equivalent stresses for each element
 - = 5 - print global and local stresses for each element
 - = 6 - print local stresses, principal stresses, and equivalent stresses for each element
- 6 LQ2 averaged nodal stress printout key, used only if KSTR >0 (see notes 2, 6, 7)
- = 0 - do not calculate or print averaged nodal stresses
 - = 1 - print global stresses, principal stresses and equivalent stresses at the nodes.
 - = 2 - print global stresses at the nodes.

- = 3 - print principal and equivalent stresses at the nodes
 - = 4 - calculate all nodal stresses and retain them on a file for post processing, but do not print them.
- 7 LQ7 coordinate system key for eigenvector (mode shape) printout, see note 1
 - = 0 - print local components of eigenvector at nodes where local displacement coordinate systems have been defined. In addition, provide another printout listing global cartesian components of eigenvector at all nodes.
 - = 1 - print global cartesian components of eigenvector at all nodes.
 - = 2 - print global cartesian components of eigenvector at all nodes, but at nodes where local displacement coordinate systems have been defined, provide the local components only.
- 8 KPSEU calculation key for pseudo static solution for missing mass correction, (see note 8) and/or influence matrix for multiple support excitation.
 - = 0 no calculation
 - = 1 calculate pseudo static solution
 - = 2 calculate influence matrix under support displacements
 - = 3 calculate pseudo static solution plus influence matrix under support displacements
- 9 TOLPF Threshold value to selectively print out modal mass ratio which exceeds the threshold (see note 9)
- 10 CFREQ Cutoff frequency (Hz) for modes to be included for missing mass correction. (See note 11).

Notes:

1. It is recommended to exercise caution in interpreting the modal response quantities (e.g., modal stresses), since the magnitudes of a typical response quantity are not meaningful. The mode shapes from which the response quantities are calculated, as implied by the name, are only shapes but not true displacements.
2. The keys in this data group are used to initiate the computation of the output quantities, and to indicate the type of output request (e.g., global stresses or local stresses). The actual printout, however, is controlled by the *PRINCNTL data group, which can also be used to provide a selective printout at a subset of nodes, elements, etc. Refer to the *PRINCNTL data group for available options and defaults.

Analysis Data

Output Control Data

3. Reaction forces at nodes with local displacement coordinate system are provided in that system. The summation of reaction forces is always in the global Cartesian coordinate system. Reaction forces printout at all constrained nodes is always provided when $KRCTN = 1$, unless the default output request is overridden in the *PRINTCNTL data group.
4. The variable KSTR selects the locations for stress calculations. Gauss points are located in the interior of the elements, where the stresses are most accurate. Node points are located on the element boundaries. Nodal point stresses obtained on the element level are in general discontinuous across element boundaries. Use the variable LQ2 (entry no. 6) if averaged nodal stresses are desired.
5. For line elements (beams, springs), any nonzero value of KSTR can be used. Stress resultants in the element local coordinate system are provided for these elements, e.g., normal force, shear forces, and bending moments.
6. The averaged nodal stress printout at all nodes is always provided when $KSTR > 0$ and $LQ2 > 0$, unless the default output request is overridden in the *PRINTCNTL data group.
7. To obtain averaged nodal stress printout for nodes lying within certain regions of the model, use the *REGIONS data group. To obtain a filtered printout of nodal principal stresses and equivalent stresses within specified limits, use the *STRSFILTER data group.
8. Pseudo static load solution is required for missing mass calculation in shock spectrum analysis. Without saving it to post-processing file, missing mass correction can not be calculated in shock spectrum analysis
9. The modal mass ratio is modal mass to system mass ratio. The threshold value is given in percentage. The default value is 1%.
10. Influence matrix used only for multiple support excitation captures the influence of support displacements on structural displacements.
11. This parameter is optional. If not used, all modes extracted in the eigenvalue run will be used to compute pseudo static solution. With a nonzero value, only those modes below the specified CFREQ value will be included in the computation of pseudo static solution.

7.5.2 *NLOUT Data Group - Selecting Output Parameters

Applicable analysis types: NLSTATIC, LTRANSIENT, NLTRANSIENT

This data group defines parameters for output selection of certain nodal and Gauss point quantities, like reaction forces, element stresses, etc. In case of multi-step analysis, the control parameters may be specified for a selected number of steps.

Group ID card:

*NLOUT

Output parameters card set:

Entry No:	1	2	3	4	5	6	7	8	9	10
Variable:	KRCTN	KSTR	LQ1	LQ2	LQ7	IFREQ	ISTYP	IRSFR	LQSN	KELFR
Max char:	2	2	2	2	2	5	2	5	2	2

Entry Variable Description

- | | | |
|---|-------|---|
| 1 | KRCTN | reaction force calculation key, see note 1 and *PRINTCNTL
= 0 - off
= 1 - on |
| 2 | KSTR | location selection for element stress/strain output, see note 2 and *PRINTCNTL
= 0 - no element stresses
= 1 - stresses at Gauss points only
= 2 - stresses at node points only
= 3 - stresses at Gauss and node points |

- 3 LQ1 type of element stress output, see note 2 and *PRINTCNTL
- = -1 - suppress all element stresses
 - = 0 - output all element stresses (local, global, principal, and equivalent stresses)
 - = 1 - output global stresses
 - = 2 - output principal and equivalent stresses
 - = 3 - output local stresses (for shell elements or orthotropic materials)
 - = 4 - output global, principal, and equivalent stresses
 - = 5 - output global and local stresses
 - = 6 - output local, principal, and equivalent stresses
- 4 LQ2 type of average nodal stress printout, see note 3 and 4
- = 0 - do not calculate or print averaged nodal stresses
 - = 1 - print global, principal and equivalent stresses at the nodes
 - = 2 - print global stresses at the nodes
 - = 3 - print principal and equivalent stresses at the nodes
 - = 4 - calculate all nodal stresses and retain them on a file for post-processing, but do not print them
- 5 LQ7 coordinate system key for displacement printout
- = 0 - print local components of displacement at nodes where local displacement coordinate systems have been defined. In addition, provide another printout listing global cartesian components of displacements at all nodes.
 - = 1 - print global cartesian components of displacements at all nodes
 - = 2 - print global cartesian components of eigenvector at all nodes, but at nodes where local displacement coordinate systems have been defined, provide the local components only.
- 6 IFREQ interval of steps at which the selected quantities are to be printed and saved for postprocessing
- = n - the requested output will be printed at the 1st and every nth step thereafter, default is n = 1, for nonlinear static analysis
 - = number of equal steps in the event, for direct transient analysis (See Note 5)

- 7 ISTYP selection of stress/strain measure for printout, see note 6 for large strain analysis.
 = 0 - all printed stresses will be 2nd Piola - Kirchhoff stress [default]
 = 1 - all printed stresses will be Cauchy stress
- 8 ISRFR interval of converged steps at which restart information is saved.
 restart data will be saved every nth converged step (note that the
 = n - restart data for the first and the last user specified step will be always
 saved). User must use SAVE and FILE executive commands (see [Section 5.3](#)) with this option.
- 9 LQSN type of element strain output, see note 2 and *PRINTCNTL
 = -1 - suppress all element strains
 = 0 - output all element strains (total, plastic, and creep strains)
 = 1 - output global total strains
 = 2 - output global plastic strains
 = 3 - output global creep strains
 = 4 - output global total and plastic strains
 = 5 - output global total and creep strains
 = 6 - output global plastic and creep strains
- 10 KELFR element strain energy, element internal forces and rigid element forces calculation key. The element internal forces are always provided in the global cartesian system, except for nodes which have local displacement co-ordinate system (see note 7) and *PRINTCNTL.
 =0 - Off
 =1 - On

Notes:

1. KRCTN controls the calculation of reaction forces. In order to have reactions for printout of history data, KRCTN must be set to one. Reaction forces at nodes with local displacement coordinate system are provided in that system, whereas the summation of reaction forces is given in the global cartesian coordinate system. Printout control of reactions for subset of nodes is available in the *PRINTCNTL data group.

Analysis Data

Output Control Data

2. Entries 2 and 3 (KSTR and LQ1) are used to control the location and the type of element stress output, respectively. The actual printout is controlled by the *PRINCNTL data group, which also may be used to provide selective printout at subset of elements.
3. To obtain averaged nodal stress printout for certain geometrical regions of the model, use the *REGION data group. To obtain filtered stress printout within specified limits, use the *STRSFILTER data group.
4. In nonlinear analysis, NISA outputs two values for averaged nodal effective stresses labeled as effective stress (EST) and equivalent von Mises stress. The effective stress is calculated at element integration point using stress tensor at the point. The value is projected to the node and averaged. Equivalent von Mises stress is calculated directly at the node from the stress tensor which is projected to the node and averaged.
5. The convergence of the default value of IFREQ being n, the number of equal steps in the event (specified in the INCREMENTS parameter of the *EVENT data group) is that, by default, snapshot results are saved only at last steps of every event. This parameter has no effect on the NFRQ parameter (5th entity of *HISTOUT group); by default, the histories are still stored at every time step.
6. When large strain analysis is opted, all printed stresses will be Cauchy stresses and all printed strains will be the logarithmic strains regardless of user's input. Otherwise all printed strains will be Green-Lagrangian strains.
7. This key is used to initiate the computation of element strain energy, element internal forces and rigid element forces. The actual printout however, is controlled by the *PRINCNTL data group, which can also be used to provide a selective printout at a subset of nodes, elements, etc. Refer to the *PRINCNTL data group for available option and defaults.

7.5.3 *PRINTCNTL Data Group - Selective Printout Control

Applicable analysis types: STATIC, EIGENVALUE, BUCKLING, NLSTATIC, SHEAR, THEAT, LTRANSIENT, NLTRANSIENT

This data group may be used to control the bulk of printout for the output quantities (displacements, reactions, stresses, etc.) The printout of a typical output quantity may be provided in its entirety, totally suppressed or provided for a subset(s) of nodes or elements. A default option applies for any output quantity not specified in this group.

Group ID card:

*PRINTCNTL

In some cases, only the group-ID card may be needed, see note 2.

Printout control card set:

	\$									
Entry No:	1	2	3	4	5	6	7	8	9	10
Variable:	LABEL	I1	I2	I3	I4	I5	I6	I7	I8	I9
Max char:	12	6	6	6	6	6	6	6	6	6

Default
option

The following labels are applicable to Heat Transfer analyses only (see Note 5)

HFLT	- total heat flux on element boundary face (excluding concentrated nodal heat fluxes)	-1 (none)
HFLC	- heat flux due to convection on element boundary face	-1 (none)
HFLR	- heat flux due to radiation on element boundary face	-1 (none)
NDFLUXES	- nodal heat flow at nodes with specified temperature	0 (all)
TEMPERATURE	- nodal temperatures	0 (all)

— tab (\$) —

- 2 I1 output option or set identification number
 < 0 - output designated by LABEL will be suppressed
 = 0 -output designated by LABEL will be provided in its entirety (e.g., element stresses for all elements)
 > 0 -output designated by LABEL will be provided for members of the set with an ID of I1, which has been defined in the *SETS data group.
- 3 - 10 I2 to I9 additional set ID numbers to complete the definition of the subset for which the output is requested, valid if I1 > 0 only.

Notes:

1. If more than 9 sets are required to define the subset for a typical output quantity, continue on additional cards. Additional cards must start with a tab (\$) character. Zero set ID numbers are ignored.
2. In some cases, the group ID card may be all that is needed to fully define the printout options as follows (see example 7.5.1):
 - Supplying just the group ID card has the effect of duplicating the printout options defined in the latest load case in static analysis (eigenvalue output block in eigenvalue analysis or event in nonlinear static analysis).

- The group ID card may take the form: *PRINTCNTL, IREF = n, where, n is an integer > 0,

referring to a previous load case ID in static analysis (eigenvalue output block in eigenvalue analysis or event in nonlinear static analysis). This has the effect of duplicating the printout options defined in the referenced load case. In restart runs, the load case IDs in previous runs cannot be referenced here.

3. This data group controls the printout of a typical output quantity provided that the computation of that quantity has been requested in the *LDCASE data group (for static analysis), in the *EIGOUT data group (for eigenvalue analysis), in the *NLOUT data group (for nonlinear static and direct transient dynamic analyses), or in the *LDCOMB data group (for load combination in static analysis).
4. For static, eigenvalue, buckling and nonlinear static analyses, output requests made in alternate data groups (*I5, *N5, *REGIONS) supersede those specified in this data group.
5. For heat transfer analyses, heat flux output is available for element boundary faces on which distributed heat flux, convection or radiation boundary conditions are specified. The *PRINTCNTL data group, in addition to the *TEMPOUT data group for transient heat transfer analysis, are the only data groups that control the computation and printout of the output quantities.
6. This option controls the printout of the summary of line element resultant forces for static, eigenvalue, buckling, and nonlinear static analyses. The allowed values for SLFO are -1 (no print), 0 (print all), and set number (print set). It should be noted that this print option can be activated only if the computation key (KSTR) is greater than zero.
7. For user-defined elastic-plastic material model up to 6 user-defined internal variables, if assigned, will be automatically printed along with the regular stress components. These quantities may be printed at either Gauss points or nodal points, or both, if applicable, in either local coordinate system or global system or both. See KSTR and LQ1 in *NLOUT and Appendix D.3.
8. AVND also controls the printout of averaged stress resultants for shell elements. The stress resultants for general shell elements (NKTP = 20, 40 and 41) are computed and averaged in element local coordinate system defined in NRCS of *ELEMENT data group. For layer composite shell elements (NKTP = 32, 33), stress resultants are computed and averaged in material directions of the first layer.

Example 7.5.1:

The following cards illustrate the use of the *PRINTCNTL data groups. Comment cards are included for explanation.

```
** — Begin a static load case —
*LDCASE, ID = 1001
1, 1, 3 $0, 1
**
...
** Insert data groups defining loads and B.C. here
**
...
*PRINT
LDVECTOR,          0
AVNDSTRESS,       11, 12, 13, 14, 15, 16,17
$,                18, 19, 20
ELSENERGY,        200
DISP,             11
** Note: set-ID no. 11-20 and 200 must be defined in *SETS data group
** — Begin 2nd load case —
*LDCASE, ID = 2002
1, 1, 3
**
...
** Insert data groups defining loads and B.C. here
**
...
*PRINT
** supplying just the group ID on the above card repeats the
** printout options of the preceding load case
** — Begin 3rd load case —
*LDCASE, ID = 3003
1, 1, 3
**
...
** Insert data groups defining loads and B.C. here
**
...
*PRINT, IREF = 1001
** The above card has the effect of repeating the printout options
** specified in load case ID no. 1001
**
*ENDDATA
```

7.5.4 *REGIONS Data Group - Regions for Averaged Nodal Stress Printout

Applicable analysis types: STATIC, EIGENVALUE, BUCKLING, NLSTATIC, LTRANSIENT, NLTRANSIENT

This data group may be used to specify the regions (consequently the nodes) of the model at which the averaged nodal stresses will be printed.

Group ID card:

*REGIONS

Regions definition card set:

Entry No:	1	2	3	4	5	6						
Variable:	<table border="1"><tr><td>XMIN</td></tr></table>	XMIN	<table border="1"><tr><td>XMAX</td></tr></table>	XMAX	<table border="1"><tr><td>YMIN</td></tr></table>	YMIN	<table border="1"><tr><td>YMAX</td></tr></table>	YMAX	<table border="1"><tr><td>ZMIN</td></tr></table>	ZMIN	<table border="1"><tr><td>ZMAX</td></tr></table>	ZMAX
XMIN												
XMAX												
YMIN												
YMAX												
ZMIN												
ZMAX												
Max char:	10	10	10	10	10	10						

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	XMIN	minimum X coordinate for the region
2	XMAX	maximum X coordinate for the region
3	YMIN	minimum Y coordinate for the region
4	YMAX	maximum Y coordinate for the region
5	ZMIN	minimum Z coordinate for the region
6	ZMAX	maximum Z coordinate for the region

Notes:

1. The region is specified in global cartesian coordinates.
2. Input as many regions as desired, one region per card.
3. Printout request defined in this data group overrides that made in the *PRINTCNTL data group.
4. This output is possible only if averaged nodal stresses are requested, i.e., KSTR > 0 and LQ2 > 0 on the *LDCASE data group (for STATIC and BUCKLING), on the *EIGOUT (for EIGENVALUE), or on the *NLOUT (for NLSTATIC, LTRANSIENT and NLTRANSIENT).

7.5.5 *STRSFILTER Data Group - Nodal Stress Filtering Data

Applicable analysis types: STATIC, EIGENVALUE, BUCKLING, NLSTATIC

This data group may be used to obtain a table of nodes where principal stresses, von Mises equivalent stress, maximum shear stress and octahedral shear are outside a specified range.

Group ID card: *STRSFILTER

Nodal stress filtering data card set:

Entry No:	1	2	3	4	5	6	7	8
Variable:	SIGMA1	SIGMA2	SIGMA3	SIGMA4	SIGMA5	SIGMA6	SIGMA7	SIGMA8
Max char:	10	10	10	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	SIGMA1	lower limit for principal stresses
2	SIGMA2	upper limit for principal stresses, SIGMA2 \geq SIGMA1
3	SIGMA3	lower limit for maximum shear stress
4	SIGMA4	upper limit for maximum shear stress, SIGMA4 \geq SIGMA3
5	SIGMA5	lower limit for von Mises equivalent stress
6	SIGMA6	upper limit for von Mises equivalent stress, SIGMA6 \geq SIGMA5
7	SIGMA7	lower limit for octahedral shear stress
8	SIGMA8	upper limit for octahedral shear stress, SIGMA8 \geq SIGMA7

Notes:

1. This output is possible only if averaged nodal stresses are requested, i.e., KSTR > 0 and LQ2 > 0 on the *LDCASE data group (for STATIC and BUCKLING), on the *EIGOUT (for EIGENVALUE), on the *NLOUT (for NLSTATIC).
2. If no table of filtered nodal stresses is desired for a typical stress quantity, set the corresponding upper limit equal to the lower limit.
3. If the model contains shell elements (e.g., NKTP = 20, 40), then separate tables will be printed for the middle, top, and bottom surfaces. In this case, three sequential *STRS-

Analysis Data

Output Control Data

FILTER groups may be used to filter the middle, top and bottom stresses, respectively. If only one *STRSFILTER data group is given, this data will be used for all three surfaces. If only two *STRSFILTER data groups are given, then the first group will be used to filter the middle surface stresses, the second group will be used for both the top and bottom surfaces.

4. Refer to the *SFDCOMP data group if the model contains composite elements (NKTP = 7, 32, 33) and stress filtering is desired.

7.5.6 *SFDCOMP Data Group - Composite Element Stress Calculation and Filtering Data

Applicable analysis types: STATIC, EIGENVALUE, BUCKLING, LTRANSIENT

Group ID card:

*SFDCOMP

This data group may be used for composite elements (NKTP = 7, 32, 33) to:

- (a) Specify stress ratios which can be used to filter the element stresses for composite elements (not including the interlaminar stresses, for NKTP = 7, 32, 33)
- (b) Calculate interlaminar shear stresses only (for NKTP = 32, 33 and NORDR = 2, 3, 5, 6, 7, and 11)
- (c) Print the effective stiffness coefficients for composite shell elements

Composite element stress calculation and filtering card set:

Entry No:	1	2	3
Variable:	SRATIO	ILSS	NSRL
Max char:	12	2	2

Entry Variable Description

- | | | |
|---|--------|--|
| 1 | SRATIO | filtering stress ratio. In composite elements, allowable stresses are input as material properties in the *MATERIAL data group, and the ratio of actual to allowable stress is computed at the mid surface of every layer. If SRATIO is set to zero, then all layer stresses and stress ratios will be printed. However if SRATIO is positive, then only stress ratios above SRATIO will be printed. (See notes 2 and 3) |
|---|--------|--|

Analysis Data

Output Control Data

- | | | |
|---|------|--|
| 2 | ILSS | element interlaminar shear stress calculation key (See notes 1 and 4)
= 0 no stress calculation
= 1 calculate stresses with respect to the material axes of the top layer
= 2 calculate stresses with respect to the material axes of each layer
= 3 calculate stresses with respect to the material axes of the top layer and material axes of each layer |
| 3 | NSRL | serial number of the element identification index - see *ELTYPE data group (see note 5) |

Notes:

1. This output is possible *only if* element stresses are computed, i.e., $KSTR > 0$ on the *LDCASE data group (for STATIC and BUCKLING), or on the *EIGOUT (for EIGENVALUE).
2. This data group specifies the stress ratio and interlaminar shear stress calculation for a given serial number of element identification index (see *ELTYPE data group).
3. If only a single value of the parameter **RATIO** is input, the program assumes the same **SRATIO** value for all composite elements.
4. The interlaminar stress averaging option is not available for composite elements.
5. If $NSRL > 50$, the actual NSRL is (NSRL-50) for which the effective stiffness coefficient will also be printed out.

7.5.7 *TEMPHISTORY Data Group - Temperature History Output for Transient Heat Transfer

Applicable analysis types: THEAT

This data group may be used in transient heat transfer analysis to specify the nodes for which temperature history plots and output is desired.

Group ID card:

*TEMPHISTORY

Node list card set:

Entry No:	1	2	3	4	5-10	11
Variable:	NT	NODE1	NODE2	NODE3	...	NODE10
Max char:	6	6	6	6	...	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NT	total number of nodes for which temperature vs. time plots are desired
2	NODE1	first node number
3	NODE2	second node number
4 - 10		... as required...
11	NODE10	10th node number (if required)

Note:

1. If more than 10 nodes are to be input for plotting and output, use additional cards as necessary, 11 entries per card.

7.5.8 *TEMPOUT Data Group - Time Steps for Transient Heat Transfer Output

Applicable analysis types: THEAT

This data may be used in transient heat transfer analysis to specify the time steps at which the temperature printout is desired.

Group ID card:

*TEMPOUT

Output time steps card set:

Entry No:	1	2	3			
Variable:	<table border="1"><tr><td>TSTART</td></tr></table>	TSTART	<table border="1"><tr><td>TEND</td></tr></table>	TEND	<table border="1"><tr><td>TINC</td></tr></table>	TINC
TSTART						
TEND						
TINC						
Max char:	10	10	10			

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	TSTART	the starting time for output and contour plotting
2	TEND	the ending time for output and contour plotting
3	TINC	time increment

Notes:

1. Input as many time step selection cards as desired to define the time steps for which output and contour plotting is required (each card contains three entries).
2. Post-processing for contour plotting can only be done at the time steps specified in this group.

7.5.9 *HISTOUT -History Output Request Data Group

Applicable analysis type: LTRANSIENT, NLTRANSIENT

This data group is optional and can be used to request history computation for printout and/or postprocessing. This data group is ignored in a Restart 5 run.

Group ID card:

*HISTOUT

Entry No:	1	2	3	4	5	6
Variable:	LABEL	IDSET	TSTART	TEND	NFRQ	IOPT
Max char:	12	6	12	12	5	2

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	LABEL	component label of time history. valid labels are: REACTIONS : reaction force (see note 2) DISPLACEMENT : displacement AVNDSTRESSES : averaged nodal stress (for shells, this label corresponds to middle surface) VELOCITY: : velocity ACCELERATION : acceleration BSTRESS : bottom surface nodal stress (applicable to shells only) TSTRESS : top surface nodal stress (applicable to shells only) RSTRESS : resultant stress (applicable to composite shell elements only) BEFORCE : beam end force
2	IDSET	node number set ID (defined in *SETS group) for history pick up. If LABEL = BEFORCE this entry indicates the element number set ID. Only one set is allowed.
3	TSTART	start time of history output. (default = starting time of first event)
4	TEND	end time of history output. (default = ending time of last event)

Analysis Data

Output Control Data

- | | | |
|---|------|--|
| 5 | NFRQ | frequency of steps for picking up the history, e.g., if NFRQ = 5, pick up history values at the 5th, 10th, 15th,... step within the specified time TSTART to TEND. |
| 6 | IOPT | 0: print and store the history output (default)
1: store the history output only. |

Notes:

1. All responses will be in the global coordinate system except at nodes where local coordinate systems are defined in which case, they will be in the local coordinate system corresponding to that node.
2. In order to save the history of reactions, reaction force calculation key (KRCTN) in *NLOUT data group should be set to 1.

7.5.10 *I5 Data Group - Alternate Form for Printout Control

Applicable analysis types: STATIC, EIGENVALUE, BUCKLING, NLSTATIC

This data group may be used to obtain partial printout of element stresses, element internal forces and strain energy and/or nodal displacements. It is recommended to use the *PRINTCNTL data group, since this group (*I5) will not be supported in future versions.

Group ID card:

*I5

Node (or element) list card set:

Entry No:	1	2	3	4
Variable:	NBEG	NEND	INCR	LABEL
Max char:	6	6	6	4

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NBEG	node (or element) number
2	NEND	last node (or element) of a range of node (or element) numbers
3	INCR	increment for the range of node (or element) numbers.
4	LABEL	control label stresses for elements <ul style="list-style-type: none"> = ELE - stresses for elements NBEG through NEND in increments of INCR will be printed = NOD - displacements for nodes NBEG through NEND in increments of INCR will be printed = ELF - internal forces and strain energy for elements NBEG through NEND in increments of INCR will be printed.

Analysis Data

Output Control Data

Notes:

1. Input as many cards as desired.
2. The computation of the output quantities listed in this group must be requested on the *LDCASE (for STATIC and BUCKLING), on the *EIGOUT (for EIGENVALUE), or on the *NLOUT (for NLSTATIC).
3. Printout requests defined in this data group override those made in the *PRINTCNTL data group.

7.5.11 *N5 Data Group - Alternate Form for Nodal Stress Printout Control

Applicable analysis types: STATIC, EIGENVALUE, BUCKLING, NLSTATIC

This data group may be used to obtain a partial printout of averaged nodal stresses for a subset of nodes. It is recommended to use the *PRINTCNTL data group, since this group (*N5) will not be supported in future versions.

Group ID card:

*N5

Node list card set:

Entry No:	1	2	3
Variable:	NBEG	NEND	INCR
Max char:	6	6	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NBEG	node number
2	NEND	last node number of a range of nodes
3	INCR	increment for the range of nodes

Notes:

1. Input as many cards as desired to complete the node list.
2. If both *REGIONS data and *N5 data are given, only those nodes existing in both data groups will be printed.
3. If *N5 data group is present within the load case then it will be used to control the printout of the averaged nodal stresses instead of the *PRINTCNTL data group.
4. This output is possible only if averaged nodal stresses are requested, i.e., KSTR > 0 and LQ2 > 0 on the *LDCASE data group (for STATIC and BUCKLING), on the *EIGOUT (for EIGENVALUE), or on the *NLOUT (for NLSTATIC).

7.5.12 *POSTCNTL Data Group - Selective saving of element output of quantities on the post-data file for Post Processing

Applicable analysis types: STATIC, EIGENVALUE, NLSTATIC and BUCKLING

This data group is used to control the bulk of element results (such as element Gauss point stresses, element nodal point stresses, element nodal strains etc.) to be saved in the post-data file (file 27) for buckling analysis and postprocessing. (For details see note 4 and 5.) The saving of a typical quantity may be in its entity or total suppression for all elements. A default option (no saving) applies for any quantity not specified in this group.

Group ID card:

*POSTCNTL

In some cases, only the group-ID card may be needed, see note 3.

Printout control card set:

Entry No:	1	2
Variable:	LABEL	I1
Max char:	12	6

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	LABEL	output label

Of the following labels, the first three are applicable to static, nonlinear static, buckling and eigenvalue analyses (see note 3 for details), while the next two apply to nonlinear static only.

		<u>Default option</u>
NDSTRS	- Element nodal stresses	-1 (none)
NDSTRN	- Element nodal total strains	-1 (none)
GPSTRS	- Element Gauss stresses	-1 (none)
NDSTNP	- Element nodal plastic strain	-1 (none)
NDSTNC	- Element nodal creep strain	-1 (none)
ERRIND	- Element error indicator	-1 (none)
CPSTRS	- Element Centroidal Stress	-1 (none)

		CPRFOR	- Element Centroidal resultant forces -1 (none)
		GPRFOR	- Element Resultant forces at Gauss -1 (none) (see Note Point 8)
2	II	Output option	
		< 0	suppress output any quantity designated by LABEL
		= 0	save all quantities designated by LABEL

Notes:

1. This group controls the saving of selective element results to the post-data file (file 27) provided that the computation of that quantity has been requested (e.g. set KSTR>0 in *LDCASE to activate the computation of element stress in static analysis).
2. The post-data file (file 27) is not automatically saved by NISA. The executive command 'SAVEfile' in (5.3.2) should be used in order to retain the file 27 after a NISA run.
3. The options NDSTRS, NDSTRN and GPSTRS will save nodal stress, nodal strain and Gauss point stress components in global coordinate system to the post-data file for the solid and shell elements (NKTP = 1, 2, 3, 4, 5, 9, 20, 34, 36, 37, 40, 41). For the composite and sandwich elements (NKTP = 7, 32, 33), stresses and strains saved in the file 27 are in material principal direction. For the line elements (NKTP = 11, 12, 13, 14, 15, 17, 18, 21, 22, 38, 46, 47), no stresses and strains will be saved. Instead, stress resultants will be saved if the option NDSTRS is active. [Table 7.4](#) lists details of the quantities saved for different element types with different options.

Table 7.4: Description of element results saved by *POSTCNTL

Option	Element Type	Quantities Saved
NDSTRS	solid and shell element composite solid (NKTP = 7) composite shell (NKTP = 32) sandwich shell (NKTP = 33) line elements	* global stresses * layer stresses in material principal directions * layer stress ratios in material principal directions * failure criteria (Tsai-Wu) * layer stresses in material principal directions * layer stress ratio in material principal directions * transverse shear stresses on top and bottom surface of each layer * stress resultants in material directions of the first layer * failure criteria (Tsai-Wu, Hill-Mises, Max stress) * layer stress in material principal directions * layer stress ratios in material principal directions * stress resultants in material directions of the first layer * failure criteria (Max. stress) * stress resultants in element local coordinate
GPSTRS	linear, nonlinear solid and shell elements composite solid (NKTP = 7) composite shell (NKTP = 32) sandwich shell (NKTP = 33) line elements	* global stresses * layer global stresses * layer stresses in material principal directions * layer stresses in material principal directions * stress resultant in element local coordinate

Option	Element Type	Quantities Saved	
NDSTRN	linear, nonlinear solid and shell elements	*	global strains
	composite solid (NKTP=7)	*	layer strains in material principal directions
	composite shell (NKTP=32)	*	layer strains in material principal directions
	sandwich shell (NKTP=33)	*	layer strains in material principal directions
	line elements	*	not available

4. Saving element nodal stress and/or strain results to the post-data file is required by DISPLAY-POST if user wants to post process unaverage nodal stresses plot, unaverage nodal strain plot and nodal stress error plot. Without saving these quantities to the post-data file, these plots will not be available.
5. For buckling analysis, Gauss point stresses will be saved to the post-data file automatically regardless the option in GPSTRS.
6. For multiple load cases, a single *POSTCNTL card may be sufficient to completely define saving options described as below:
 - Supplying just the group ID card alone has the effect of duplicating the saving options defined in the latest load case in static analysis.
 - The group ID card may take the form: *POSTCNTL, IREF=n, where n is a previous load case ID. This has the effect of duplicating the saving options defined in the previous load case ID = n. In restart runs, those load case IDs in previous runs can not be referred.
7. For nonlinear static analysis, the options NDSTNP and NDSTNC will save nodal plastic strain and nodal creep strain components in global coordinate system to the post-data file for NKTP = 1, 2, 3, 4, 20, 40 elements.
8. The element error indicator is based on stress errors in energy norm $(1/2 \int e_{\sigma}^T \cdot D^{-1} e_{\sigma} d(vol))^{1/2}$. The stress errors e_{σ} are defined as the difference between the calculated stresses and the smooth stresses. The average nodal stresses are used as a basis for projecting the smooth stresses. At present, error indicator calculation is available for element type NKTP = 1, 2, 3, 4, 5, 20, 32, 33, 40 in linear static analysis.

7.6 Data Terminator

7.6.1 *ENDDATA Data Group - Input Data Terminator

Applicable analysis types: ALL

Group ID card:

*ENDDATA

The group ID card must be the last card in the data deck.

Modal Dynamic Analysis Data

8.1 Introduction

The input data for all modal dynamic analyses is presented in this chapter. Modal dynamic analysis types include the following:

1. Transient dynamic analysis (TRANSIENT)
2. Random vibration analysis (RANDOM)
3. Frequency response analysis (FREQUENCY)
4. Shock spectrum analysis (SHOCK)

A theoretical overview of these analyses is presented in [Section 2.5](#), and a description of the analysis capabilities and output features is given in [Section 3.5](#). A modal dynamic analysis is a two-step procedure. The first step is an eigenvalue analysis run which determines the dynamic characteristics of the model in terms of its natural frequencies, mode shapes, participation factors, modal stresses, etc. The second step is the modal analysis run of any of the analysis types listed above. The normal mode method, which is employed for modal dynamic analysis, includes forming the modal load, solving for the modal (generalized coordinate) responses of each mode of vibration, and obtaining the physical responses such as displacements and stresses through modal superposition.

The input data for a typical modal analysis consists of two data blocks (an executive command data block, and an analysis data block) and a data terminator as shown in [Figure 8.1](#). The executive command data block specifies overall control parameters and is described in [Section 8.3](#). An alphabetical list of the executive commands for modal dynamic analysis is given in [Table 8.1](#).

The analysis data block consists of distinct data groups, each performing a specific function. The details of the data groups for analysis control, load function description, load definition and output control are given in [Section 8.5](#) through [Section 8.8](#). The data groups are presented in the sequence given in [Table 8.2](#). For easy reference, an alphabetical list of *all* data groups available in NISA is given in [Table 5.4](#).

Each data group consists of the group identification card followed by its free format data, which may consist of one or more card sets. Each card set may consist of one or more cards. The rules for free format input are given in [Section 5.2](#). Each group has a descriptive identification name indicating the function of the data group. For example, the damping values are specified in the *DAMPING data group. The group identification name may be abbreviated to a minimum of the first four characters.

It should be noted that the data groups may be arranged arbitrarily. In addition, *not* all the data groups are required for all of the analysis types, i.e. some data groups are optional. The distinction is explicitly stated in the description of each data group.

The format of presentation in this chapter is intended to familiarize the user with various aspects of the four analysis types simultaneously. However, the input data is classified by analysis type in [Section 8.2](#).

After understanding the contents of this chapter, the user may employ [Section 8.2](#) as a guide to perform specific modal dynamic analysis for the problem at hand.

Procedure:

As briefly discussed above, the general procedure for a modal dynamic analysis involves the following two steps:

1. After establishing a finite element model, perform an eigenvalue analysis to solve for a suitable number of frequencies and mode shapes. If a particular response quantity such as stress is to be requested in the modal dynamic analysis, the corresponding modal response quantity, that is, modal stresses, must be requested in the eigenvalue analysis run. Save NISA files 26 and 27. If a transient dynamic analysis with nonzero initial displacements and velocities is to be performed, then in addition NISA files 30, 32, and 35 are to be saved.

2. Prepare the input data for the desired modal dynamic analysis as described in this chapter and perform the analysis. Give the appropriate executive command to access the NISA files saved from the eigenvalue analysis run.

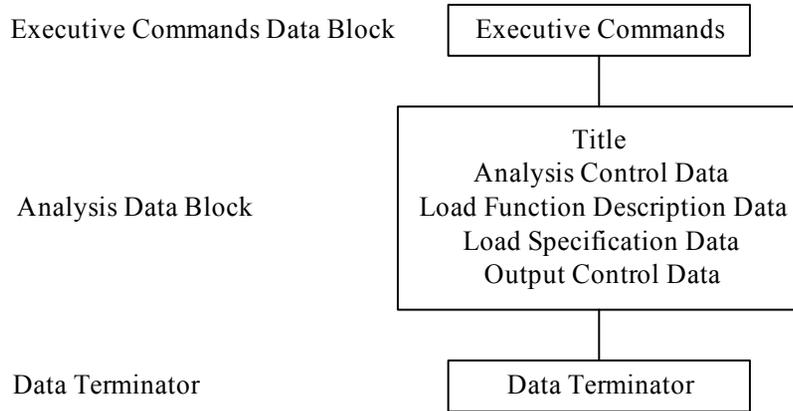


Figure 8.1: Input deck for modal dynamic analyses

Table 8.1: Alphabetical list of executive commands used in modal dynamic analysis

Executive Command Name⁽¹⁾	Description	Applicable Analysis Types⁽²⁾
ACUTOFF	Range of modal coupling	RA
ANALYSIS	Analysis type	ALL
BLANKCOMMON	Blank common storage limit	ALL
CFREQ	Frequency for missing mass correction	SHO
DAMPING	Type of damping	ALL
DELTATIME	Time increment	TR
DIRECTION	Direction superposition procedure	SHO
DPROB	Design probability	RA, SHO
EXECUTION	Execute or check control	ALL
ENDTIME	Ending time	TR

Modal Dynamic Analysis Data

Introduction

Executive Command Name⁽¹⁾	Description	Applicable Analysis Types⁽²⁾
FILENAME	NISA file names to be saved/retrieved	ALL
FLOWER	Lower limit of exciting frequency	RA, FR
FRQRDF	Reduction factor for exciting frequency point generation	RA, FR
FSMALL	Frequency cut-off to identify rigid body modes	ALL
FUPPER	Upper limit of exciting frequency	RA, FR
GENFREQUENCY	Control generation of exciting frequency points	RA, FR
GFACTOR	Acceleration due to gravity	ALL
INPOLATION	Type of interpolation	RA, FR, SHO
INPHASE	Type of interpolation for phase spectra	FR
INTEGRATION	Type of integration procedure	RA
MRESPONSE	Control printout of modal responses	FR
SAVEFILE	To save specific NISA files	ALL
STARTINGTIME	Starting time	TR
TSEISMIC	Time duration for seismic event	SHO, RA
WARNING	Set warning flag	ALL

(1) **Minimum abbreviations are in bold face**

(2) ALL: modal dynamic analyses TR: Transient dynamics RA: Random vibration
FR: Frequency response SHO: Shock spectrum

Table 8.2: List of modal dynamic analysis data groups in a logical sequence

Section No. and Group ID (1)	Description	Applicable Analysis Types(2)
<i>8.4 Title</i>		
8.4.1 *TITLE	Title card	ALL
<i>8.5 Analysis Control Data</i>		
8.5.1 *MODESELECTION	Modal selection data	ALL
8.5.2 *DAMPING	Damping values	ALL
8.5.3 *TSTEP	Time step data	TR
8.5.4 *ADDFREQUENCY	Additional exciting frequencies	RA, FR
8.5.5 *INITIAL	Initial condition data	TR
8.5.6 *MATDAMPING	Material Damping Data	ALL
8.5.7 *FLOORRESPONSE	Floor Response Spectra Data	TR
<i>8.6 Load Function Description Data</i>		
8.6.1 *ARRIVALTIME	Arrival time data	TR
8.6.2 *TIMEFUNCTION	Time function definition	TR
8.6.3 *PSDFUNCTION	Power spectral density function definition	RA
8.6.4 *CORRELATION	Correlation ensemble data	RA
8.6.5 *SPECTRUM	Spectrum definition data	FR, SHO
8.6.6 *MDSPECTRUM	Multiple damping spectrum group data	SHO
8.6.7 *NONSTATIONARY	Non-stationary data	RA
8.6.8 *ENVELOPE	Envelope function data	RA
8.6.9 *PRSPECTRA	Prestored response spectra	SHO
<i>8.7 Load Specification Data</i>		
8.7.2 *GROUND	Ground motion data	ALL
8.7.3 *DCFORCE	Concentrated nodal force data	TR, RA, FR

Modal Dynamic Analysis Data

Introduction

Section No. and Group ID ⁽¹⁾	Description	Applicable Analysis Types ⁽²⁾
8.7.4 *DPRESSURE	Pressure loading data	TR, RA, FR
8.7.5 *DRIVER	Driver node data	FR
8.7.6 *MVELOAD	Moving load data	TR
8.7.7 *MVEPATH	Moving load path description	TR
8.7.8 *MSEXCITATION	Multiple support excitation data	ALL
<i>8.8 Output Control Data</i>		
8.8.1 *RSET	Response set data	TR, RA, FR
8.8.2 *HISTORY	Response time-history request data	TR
8.8.3 *SNAPSHOT	Snapshot response request data	TR
8.8.4 *PSDOUT	Response PSD request data	RA
8.8.5 *RMSOUT	Root mean square of response request data	RA
8.8.6 *SPOUT	Output spectra request data	FR
8.8.7 *RESPONSE	Maximum response request data	SHO
8.8.8 *FRSOUT	Floor Response Spectra Output	TR
<i>8.9 Data Terminator</i>		
8.9.1 *ENDDATA	Input terminator	ALL

(1) **Minimum abbreviations are in bold face**

(2) ALL: Modal dynamic analyses TR: Transient dynamics RA: Random vibration
 FR: Frequency response SHO: Shock spectrum

8.2 Classification of Data by Analysis Types

8.2.1 Overview

This section classifies the input data by analysis type. As mentioned earlier, the four analysis types employ the normal mode method. The natural frequencies, mode shapes, modal stresses, etc. are derived for the different modes from an eigenvalue analysis run. The *MODESELECTION data group may be used to select the significant modes for analysis.

Damping values may be specified via the *DAMPING data group. The DAMP executive command indicates the form of damping (modal-viscous, modal-structural, or proportional-viscous) to be specified. If an undamped response is required, no damping data need to be specified. However, random vibration analysis with zero damping is not allowed.

The applied loads may be a combination of any number of ground excitation components, nodal concentrated forces, and pressure forces specified via the *GROUND (or *MSEXCITATION), *DCFORCE, and *DPRESSURE data groups, respectively. However, only support excitation may be specified for shock spectrum analysis.

The responses to be output may be specified using the *RSET data group as response sets of displacements, velocities, accelerations, stresses, reactions, base shear, stress resultants and beam-end forces. The *RSET data is referenced in the output request data groups (e.g., *HISTORY) used in the different analysis types. The *RSET data group is not employed in shock spectrum analysis.

Additional executive commands may be used as needed. For example, the executive command FSMALL may be used to specify a cut-off tolerance which enables rigid body modes to be identified. Also, the executive command GFACTOR may be used to specify the acceleration due to gravity. The following sections describe the data relevant to each analysis type and list the valid executive commands and data groups.

8.2.2 Transient Dynamic Analysis (TRANSIENT)

Transient dynamic analysis may be used to determine the response of a structure subjected to time-varying loads and support motions. The variation of the loads with respect to time can be defined using the *TIMEFUNCTION and *ARRIVALTIME data groups.

Modal Dynamic Analysis Data

Classification of Data by Analysis Types

The executive commands STARTTIME and ENDTIME may be used respectively, to specify the starting time and ending time for time integration. The default value for STARTTIME = 0.0, but a non-zero positive ENDTIME must be specified for a successful run. DELTATIME may be used to define the time increment for integration. Different time increments may be specified for different time regions in the analysis, via the *TSTEP data group, in order to reduce computational time without loss of accuracy. In the event that different time increments are employed, the DELTATIME will serve as the default time increment in time regions that are not included in the *TSTEP data group. A zero or negative time increment is not allowed in any time region. The default values for the initial nodal displacements and velocities are zero. But non zero values can be specified through *INITIAL data group.

The designated response sets may be output in the form of time-histories or snapshots. The options for output may be specified using the *HISTORY and *SNAPSHOT data groups respectively. The snapshot (or freeze) of the responses may be requested at a given time or at an instant in time when another specified response peaks.

The valid executive commands for transient dynamic analysis are as follows⁽¹⁾:

ANALYSIS	BLANKCOMMON	DAMPING	DELTATIME
ENDTIME	EXECUTION	FILENAME	FSMALL
GFACTOR	SAVEFILE	STARTINGTIME	WARNING

(1) **Minimum abbreviations are in bold face**

Table 8.3: List of valid data groups for transient dynamic analysis ⁽¹⁾:

Group ID Name⁽¹⁾	Description	Section No.
*ARRIVALTIME	Arrival time data	8.6.1
*DAMPING	Damping values	8.5.2
*DCFORCE	Concentrated nodal force data	8.7.3
*DPRESSURE	Pressure loading data	8.7.4
*ENDDATA	Data deck terminator	8.9.1
*FLOORRESPONSE	Floor Response Spectra Data	8.5.7
*FRSOUT	Floor Response Spectra Output	8.8.8
*GROUND	Ground motion data	8.7.2
*HISTORY	Response time history request data	8.8.2
*INITIAL	Initial condition data	8.5.5
*MODESELECTION	Mode selection data	8.5.1
*MVELOAD	Moving load data	8.7.6
*MVEPATH	Moving load path description	8.7.7
*MSEXCITATION	Multiple support excitation data	8.7.8
*RSET	Response set data	8.8.1
*SNAPSHOT	Snapshot response request data	8.8.3
*TIMEFUNCTION	Time function definition	8.6.2
*TITLE	Title card	8.4.1
*TSTEP	Time step data	8.5.3

(1) Minimum abbreviations are in bold face

8.2.3 Random Vibration Analysis (RANDOM)

Random vibration analysis may be used to determine the Root Mean Square (RMS) value and the Power Spectral Density (PSD) of the response of a structure subjected to loads and support motions modelled as stationary or non-stationary random processes. The stationary processes can be described in terms of their auto- and cross-spectral density functions using the *PSDFUNCTION data group. The *CORRELATION data group may be used to specify the excitation as fully or partially correlated processes.

Exciting frequency points may be specified using *ADDFREQUENCY data group and/or automatically generated by setting the executive command GENFREQUENCY = ON. The program automatically generates a higher density of points closer to the natural frequencies corresponding to lightly damped modes as described in [Section 8.2.6](#).

Different analysis options can be specified using the INTEGRATION, and ACUTOFF commands to achieve a trade-off between cost and solution accuracy. Closed form exact integration may be employed if the input consists only of flat (white noise) PSDs. The analysis may be simplified by reducing the cross-modal coupling based on a user specified ACUTOFF value and proximity of modes.

Response PSDs and the RMS values of the designated response sets may be output as per the options specified using the *PSDOUT and *RMSOUT data groups. The valid executive commands for random vibration analysis are as follows⁽¹⁾:

ACUTOFF	ANALYSIS	BLANKCOMMON	DAMPING
DELTATIME	ENDTIME	EXECUTION	FILENAME
FLOWER	FRQRDF	FSMALL	FUPPER
GENFREQUENCY	GFACTOR	INPOLATION	INTEGRATION
SAVEFILE	STARTINGTIME	WARNING	DPROB
TSEISMIC			

(1) Minimum abbreviations are in bold face

Table 8.4: List of valid data groups for random vibration analysis

Group ID Name ⁽¹⁾	Description	Section No.
*ADDFREQUENCY	Additional exciting frequencies	8.5.4
*CORRELATION	Correlation ensemble data	8.6.4
*DAMPING	Damping values	8.5.2
*DCFORCE	Concentrated nodal force data	8.7.3
*DPRESSURE	Pressure loading data	8.7.4
*ENDDATA	Data deck terminator	8.9.1
*ENVELOPE	Envelope function data	8.6.8
*GROUND	Ground Motion data	8.7.2
*MODESELECTION	Mode selection data	8.5.1
*MSEXCITATION	Multiple support excitation data	8.7.8
*NONSTATIONARY	Non-stationary load data	8.6.7
*PSDFUNCTION	PSD function definition	8.6.3
*PSDOUT	PSD response request data	8.8.4
*RMSOUT	RMS response request data	8.8.5
*RSET	Response set data	8.8.1

(1) **Minimum abbreviations are in bold face**

8.2.4 Frequency Response Analysis (FREQUENCY)

Frequency response analysis may be used to determine the steady state response of a structure as related to the frequency of a harmonic exciting function. The harmonic loading and support motions can be described in terms of amplitude and phase spectra using the *SPECTRUM data group.

Modal Dynamic Analysis Data

Classification of Data by Analysis Types

The *ADDFREQUENCY data group allows the specification of exciting frequency points. In addition to these user specified frequencies, extra frequencies, including the natural frequencies, can be automatically generated as described in [Section 8.2.6](#), by setting the command GENFREQUENCY = ON, in order to achieve a smooth plot of response versus frequency curve.

The program also computes the response for a unit harmonic function (either displacement, velocity, acceleration or force) applied at a node. The *DRIVER data group may be employed to derive these unit harmonic point-to-point transfer functions. The *DRIVER group overrides the other forms of load specifications, i.e., *DCFORCE, *DPRESSURE, and *GROUND data groups.

The amplitude and phase angles of the designated response sets may be output using the *SPOUT data group. Also, the real and imaginary parts of the response may be obtained for a given phase shift.

The valid executive commands for frequency response analysis are as follows⁽¹⁾:

ANALYSIS	BLANKCOMMON	DAMPING	EXECUTION
FILENAME	FLOWER	FRQRDF	FSMALL
FUPPER	GENFREQUENCY	GFACTOR	INPHASE
INPOLATION	MRESPONSE	SAVEFILE	WARNING

(1) **Minimum abbreviations are in bold face**

Table 8.5: List of valid data groups for frequency response analysis

Group ID Name⁽¹⁾	Description	Section No.
*ADDFREQUENCY	User specified exciting frequencies	8.5.4
*DAMPING	Damping values	8.5.2
*DCFORCE	Concentrated nodal force data	8.7.3
*DPRESSURE	Pressure loading data	8.7.4
*DRIVER	Driver point data	8.7.5
*ENDDATA	Data deck terminator	8.9.1
*GROUND	Ground Motion data	8.7.2
*MODESELECTION	Mode selection data	8.5.1
*MSEXCITATION	Multiple support excitation data	8.7.8
*RSET	Response set data	8.8.1
*SPECTRUM	Amplitude/phase spectrum input data	8.6.5
*SPOUT	Output spectrum data	8.8.6
*TITLE	Title card	8.4.1

(1) Minimum abbreviations are in bold face

8.2.5 Shock Spectrum Analysis (SHOCK)

Shock spectrum analysis (also known as response spectrum analysis) can be used to estimate the maximum response of a structure subjected to a multi-component support motion modelled as shock spectra.

Displacement, velocity, and acceleration shock spectra may be described using the *SPECTRUM data group. There is also an option to select the 1973 horizontal or vertical design shock spectra recommended by the U.S. Nuclear Regulatory Commission for seismic design of nuclear power plants.

Modal Dynamic Analysis Data

Classification of Data by Analysis Types

Damping values are not directly employed during the analysis. A typical shock spectrum is derived for a specific damping value. However, a family of shock spectra for different values of damping may be specified via the *MDSPECTRUM group. In this event damping values may be specified for the selected modes, and the shock spectra for each modal damping value will be derived by interpolation from the specified family of shock spectra.

The maximum response may be output using the *RESPONSE data group. The output options include the specification of a modal combination rule (ABS, SRSS, NRL, CQC, GRP, TPM and DSM) for superposing the modal maxima. If more than one directional shock spectrum component is employed, the DIRECTION executive command can be included to select the procedure for combination of directional response maxima.

The valid executive commands for shock spectrum analysis are as follow⁽¹⁾:

ANALYSIS	BLANKCOMMON	DAMPING	DIRECTION
EXECUTION	FILENAME	GFACTOR	TSEISMIC
CFREQ	INPOLATION	SAVEFILE	WARNING

(1) **Minimum abbreviations are in bold face**

Table 8.6: List of valid data groups for shock spectrum analysis

Group ID Name⁽¹⁾	Description	Section No.
*DAMPING	Damping values	8.5.2
*ENDDATA	Data deck terminator	8.9.1
*GROUND	Ground Motion data	8.7.2
*MDSPECTRUM	Multiple damping spectrum data	8.6.6
*MODESELECTION	Mode selection data	8.5.1
*MSEXCITATION	Multiple support excitation data	8.7.8
*PRSPECTRA	Prestored response spectra definition	8.6.9
*RESPONSE	Response request data	8.8.7
*SPECTRUM	Response spectrum definition	8.6.5

(1) Minimum abbreviations are in bold face

8.2.6 Frequency Point Selection (for frequency response and random vibration analyses)

A list of exciting frequency points is established by NISA in order to: obtain a smooth plot of the frequency versus response curve in frequency response analysis, or reduce the numerical integration error in random vibration analysis. This list includes the following:

- (i) The lower and upper frequency limits (FLOWER and FUPPER specified in executive commands).
- (ii) All of the natural frequencies of the participating modes (selected in the *MODESELECTION group) that are encompassed between FLOWER and FUPPER.
- (iii) Frequency points representing the frequency shift in modal peak responses due to damping.
- (iv) Additional user exciting frequencies entered in the *ADDFREQUENCY group.
- (v) Intermediate frequency points.

Intermediate frequency points between the modes are automatically selected by the program depending on the value of FRQRDF and when the executive command GENFREQUENCY is set to ON.

The program uses a selection procedure that favors the concentration (or clustering) of intermediate frequency points around each natural frequency, such that the high gradients are properly defined. The number of points $NPOI_i$, and the exponent $NEXP_i$ which determines the nonlinear spacing in the vicinity of the frequency for mode i , are selected from a prestored list (given in [Table 8.7](#)) based on the damping ratio for that mode. The higher the exponent $NEXP_i$ (i.e., lightly damped mode), the higher the concentration of intermediate frequencies around the corresponding natural frequency would be. Note that a value of 1 for $NEXP_i$ would result in constant (i.e., linear) spacing of the points in the vicinity of that mode.

Table 8.7: Values of $NPOI_i$ and $NEXP_i$ selected by the program automatically based on the damping ratio of each mode

Damping Ratio	Number of Frequency Points NPOI	Exponent for Frequency spacing NEXP
$\xi \leq 0.01$	22	11
$0.01 < \xi \leq 0.02$	18	9
$0.02 < \xi \leq 0.04$	14	5
$0.04 < \xi \leq 0.08$	12	3
$0.08 < \xi \leq 0.16$	10	3
$\xi > 0.16$	10	1

Note: The values of $NPOI_i$ are multiplied by FRQRDF specified in the executive commands before being used

8.3 Executive Commands

8.3.1 General Description

The executive commands constitute the first data block in a typical NISA input deck, and they define the overall control parameters for analysis. The syntax for a typical executive command is

command name = option or data

The command name is a character string that may be abbreviated to a minimum of the first *four* unique characters, though a well spelled out form is also acceptable. Minimum abbreviations are shown in bold face letters. The option or data is a character string or data value that will be assigned to the specific command. Various command options are shown between square brackets. Default options are shown between braces. [Section 8.3.2](#) describes the executive commands common to all modal dynamic analyses, whereas [Section 8.3.3](#) describes the executive commands applicable to specific modal dynamic analysis types.

8.3.2 Executive Commands Applicable to *All* Modal Dynamic Analyses

ANALYSIS -Specify analysis type

ANALYSIS = $\left[\begin{array}{l} \{ \text{TRANSIENT} \} \\ \text{RANDOM} \\ \text{FREQUENCY} \\ \text{SHOCK} \end{array} \right]$

where,

TRANSIENT : transient dynamic analysis
RANDOM : random vibration analysis
FREQUENCY : frequency response analysis
SHOCK : shock spectrum analysis

BLANK COMMON -Blank common storage limit

$$\text{BLANK COMMON} = \begin{bmatrix} n \\ \{50000\} \end{bmatrix}$$

where,

n: an integer specifying the blank common storage limit. This defines the available working space for analysis. For most problems the default of 50000 is adequate. However, the limit may need to be increased for analyzing large problems. The maximum allowable blank common size may vary from one machine to another.

DAMPING - Specify type of damping

$$\text{DAMPING} = \begin{bmatrix} \{\text{VISCOUS}\} \\ \text{PROPORTIONAL} \\ \text{STRUCTURAL} \end{bmatrix}$$

where,

- VISCOUS : modal viscous damping
- PROPORTIONAL : proportional viscous damping
- STRUCTURAL : modal structural damping

EXECUTION -Select execution/checking run

$$\text{EXECUTION} = \begin{bmatrix} \{\text{CGO}\} \\ \text{CHECK} \end{bmatrix}$$

where,

- CGO : data checking followed by execution
- CHECK : data checking run only

FILENAME - Specify NISA file prefix

$$\text{FILENAME} = \begin{bmatrix} \text{fname} \\ \{\text{no default}\} \end{bmatrix}$$

where,

fname: an alphanumeric character string that will be used as a file name prefix. Up to 64 characters may be used. This must be the same file name prefix used to save files 26 and 27 in the eigenvalue analysis run. Additional data generated in the modal dynamic analysis run will be appended to files 26 and 27.

FSMALL - Frequency cut-off limit to identify rigid body modes

$$\text{FSMALL} = \begin{bmatrix} f \\ \left\{ 1.0 \times 10^{-8} \right\} \end{bmatrix}$$

where,

f: a frequency cut-off limit below which all the modes are treated as rigid body modes.

GFACTOR -Specify acceleration due to gravity

$$\text{GFACTOR} = \begin{bmatrix} g \\ \{1.0\} \end{bmatrix}$$

where,

g: the acceleration due to gravity

The input ground accelerations will be scaled by GFACTOR. The other input excitations will not be affected. The response acceleration output will be in 'g' units.

MSEXITATION - Selection of method for multiple support excitations

MSEXITATION = [ENVE]

where,

ENVE: Envelope Spectrum method

Note:

1. The default is taken to be the normal response spectrum method, which will be carried out in the absence of this command. Envelope spectrum can only be invoked by providing this executive command. Moreover, if the structure is subjected to any of the pre-stored spectra, then the Envelope Method cannot be invoked.

SAVEFILE - Save specific NISA file

SAVEFILE = $\left[\begin{array}{l} n1, n2... \\ \{ \text{no default} \} \end{array} \right]$

where,

n1, n2,...: NISA file numbers to be saved

This command must be used in conjunction with the FILE NAME command which specifies a character prefix (fname) for files. Refer to [Chapter 3](#) for NISA file numbers. Note that in dynamic analyses NISA files 26 and 27 are automatically saved. This command can be used to save other files.

Note:

1. For certain computer systems, the NISA files to be saved are actually specified by the Job Control Language (JCL) and this command is not needed. Refer to pertinent system manual for details.

WARNING - Set warning flag

$$\text{WARNING} = \begin{bmatrix} \{\text{GO}\} \\ \{\text{STOP}\} \end{bmatrix}$$

where,

GO : directs the program to continue even after a warning messages (s) is produced during execution.

STOP: stops program if a warning messages (s) is produced during execution.

8.3.3 Executive Commands Applicable to *Specific* Modal Dynamic Analysis Types

ACUTOFF - Specify range of modal coupling

$$\text{ACUTOFF} = \begin{bmatrix} r_c \\ \{3.0\} \end{bmatrix} \quad \begin{array}{l} \text{Applicable analysis type:} \\ \text{RANDOM} \end{array}$$

where,

$r_c < 0.0$: then all cross-mode responses will be computed

$0.0 \leq r_c \leq 1.0$: then ACUTOFF = 3.0

$r_c > 1.0$: then ACUTOFF = r_c

This command may be used as a control to increase/reduce the cost of the analysis and the accuracy of the results. A modal covariance matrix of order NMODE is computed in this analysis, where NMODE is the number of the participating modes selected in the *MODESELECTION group. The modal covariance matrix is symmetric, therefore, at the most $(NMODE) \times (NMODE + 1)/2$ number of entries may need to be calculated. An off-diagonal (i, j) entry (with $j > i$) will *only* be computed whenever the ratio of the frequencies of the corresponding modes (f_j/f_i) is less than or equal to ACUTOFF. If modal coupling is to be ignored (i.e., ACUTOFF = 1.0), no cross-mode responses will be computed. The modal covariance matrix will be diagonal and the analysis cost will be reduced. The accuracy of the results, however, might not be acceptable. On the other hand, if modal coupling is to be considered for all modes (i.e., ACUTOFF > the ratio of the highest and lowest natural frequencies of the participating modes, or ACUTOFF < 0.0), a full covariance matrix will be computed with probably many negligible off-diagonal entries for widely separated modes, and the cost of the analysis might be too high. A value of 3.0 - 6.0 for ACUTOFF should produce good results. The modal covariance matrix is treated as a banded matrix and the bandwidth is computed according to the value of ACUTOFF. Only those off-diagonal entries, within the bandwidth satisfying the ratio requirement, are actually computed.

CFREQ – Cut-off frequency for missing mass correction

CFREQ = [f] Applicable analysis type: SHOCK, TRANSIENT, FREQUENCY

where,

f = Cut off frequency in Hz used for deciding number of modes to be included in modal summation and to apply missing mass correction for all other modes. It is applicable for ground excitations only.

Note:

1. For shock spectrum analysis, the response due to shock spectrum is computed using modal summation for natural frequencies below the cut-off frequency. Beyond this frequency, the response due to missing mass in the form of pseudo static solution is added to the previous solution by ABS, SRSS etc. method of summation (as specified by user). If the CFREQ is not in the range of the extracted modes, the acceleration corresponds to the last extracted frequency is taken for pseudo static solution. If CFREQ is not specified, the cut off acceleration in *GROUND data card is used. If *MDSPEC-

TRUM is used, the cut-off acceleration only is to be specified by the user in *GROUND data card.

Similarly, for transient and frequency analyses missing mass correction in the form of pseudo static solution is added to the dynamic solution. It is applicable for multi support excitation and floor response spectrum analyses as well.

2. This CFREQ value (in Hz.) must be same as the CFREQ (in Hz.) of *EIGOUT card in Eigen value analysis NIS file. If there is no entry of CFREQ in *EIGOUT card, user shall take care to see that all the eigenvalues below this CFREQ value are extracted in Eigen value analysis run. Even otherwise, a restart 2 option is available to activate eigen run again.

DELTATIME - Specify time increment

DELTATIME = $\begin{bmatrix} \text{tinc} \\ \text{delt} \end{bmatrix}$ Applicable analysis type:
TRANSIENT, RANDOM

where,

tinc: the time increment for integration in transient analysis. (no default value)
delt: default value computed based on highest frequency ω for random vibration analysis $=2\pi/(10\omega)$

DIRECTION - Specify combination procedure across directions

DIRECTION = $\begin{bmatrix} \{\text{ABS}\} \text{ or } \{\text{PEAK}\} \\ \text{SRSS or RMS} \end{bmatrix}$ Applicable analysis type:
SHOCK

where,

ABS: absolute sum.
SRSS: square root of the sum of the squares.

DPROB- Design probability of the responses

$$\text{DPROB} = \begin{bmatrix} \text{DPROB} \\ \{0.86\} \end{bmatrix} \quad \text{Applicable analysis type:} \\ \text{RANDOM}$$

where,

DPROB : is the design probability of the response quantities required for computing various statistics of the responses like maximal value (EXTDP) and level crossings (LVLCROSS)

ENDTIME -Specify ending time

$$\text{ENDTIME} = \begin{bmatrix} t \\ \text{delt} \end{bmatrix} \quad \text{Applicable analysis type:} \\ \text{TRANSIENT, RANDOM}$$

where,

t : the ending time for time integration in transient analysis. (no default value)

delt : default value computed based on the lowest frequency ω for random vibration analysis = $5 (2\pi / \omega)$

Note:

1. In the case of transient dynamic analysis under moving loads. The value of ENDTIME may get modified

FLOWER -Lower limit of exciting frequencies in cycles/time

$$\text{FLOWER} = \begin{bmatrix} f_0 \\ \{0.3f_1\} \end{bmatrix} \quad \text{Applicable analysis type:} \\ \text{RANDOM, FREQUENCY}$$

FLOWER will be computed based on the specified f_0 and the lowest natural frequency f_1 of the participating modes as follows:

then FLOWER = ABS (f_0) f_1 for $-1.0 \leq f_0 < 0.0$.

$f_0 < 0.0$ or FLOWER = 0.3 f_1 for $-\infty < f_0 < 01.0$.

For example if -1.0 is entered, FLOWER = 0.1 f_1 .

$f_0 = 0.0$ then FLOWER = 0.3 f_1 .

$f_0 > 0.0$ then FLOWER = f_0 .

See notes under FUPPER description.

FRQRDF -Frequency reduction factor

$$\text{FRQRDF} = \begin{bmatrix} r \\ \{ 1.0 \} \end{bmatrix} \quad \begin{array}{l} \text{Applicable analysis type:} \\ \text{RANDOM, FREQUENCY} \end{array}$$

where,

r : a reduction factor for automatically generated intermediate exciting frequency points.

The pre-stored values for the number of frequency points will be multiplied by r before being used. FRQRDF will not be active if GENFREQUENCY = OFF (see [Section 8.2.6](#)).

FUPPER -Upper limit of exciting frequencies in cycles/time

$$\text{FUPPER} = \begin{bmatrix} f_u \\ 3.0f_n \end{bmatrix} \quad \begin{array}{l} \text{Applicable analysis types:} \\ \text{RANDOM, FREQUENCY} \end{array}$$

FUPPER will be computed based on the specified f_u and the highest natural frequency f_n of the participating modes as follows:

$f_u < 0.0$ then FUPPER = ABS (f_u) f_n for $-\infty < f_u \leq -1.0$.

or $FUPPER = 3.0 f_n$ for $-1.0 < f_u < 0.0$.

For example, if -10.0 is entered, $FUPPER = 10.0 f_n$.

$f_u = 0.0$ then $FUPPER = 3.0 f_n$.

$f_u > 0.0$ then $FUPPER = f_u$

FUPPER must be greater than FLOWER when both are entered as positive. At least one mode must be selected inside the region (FLOWER, FUPPER) in the *MODESELECTION data group. Response contributions of any mode selected outside the region (FLOWER, FUPPER) will still be considered at the exciting frequency points within the region. In random vibration analysis FLOWER and FUPPER specify the lower and upper limits used in the numerical integration process and the first and last frequency points at which the response PSD will be computed. In order to minimize integration errors, it is recommended that FLOWER be less than 0.3 times the first selected natural frequency, and that FUPPER be greater than 3.0 times the highest selected natural frequency.

GENFREQUENCY -Control automatic generation of frequencies

$GENFREQUENCY = \begin{bmatrix} \{ON\} \\ OFF \end{bmatrix}$ Applicable analysis type:
RANDOM, FREQUENCY

where,

ON: automatically generate exciting frequency points between modes.

OFF: suppress generation of frequency points.

These points are in addition to the user specified frequencies in the *ADDFREQUENCY data group. If the generation of frequencies is suppressed, at least one exciting frequency must be specified in the (FLOWER, FUPPER) region via the *ADDFREQUENCY data group.

INPHASE - Specify phase angle interpolation scheme

$$\text{INPHASE} = \begin{bmatrix} \{\text{LINEAR}\} \\ \text{SEMILOG} \\ \text{LOGLOG} \end{bmatrix} \quad \begin{array}{l} \text{Applicable analysis type} \\ \text{FREQUENCY} \end{array}$$

where,

- LINEAR: linear along ordinate and abscissa.
- SEMILOG: logarithmic along ordinate, linear along abscissa.
- LOGLOG: logarithmic along ordinate and abscissa.

The interpolation scheme is used to compute the input phase angles at intermediate frequency points.

INPOLATION - Specify interpolation scheme

$$\text{INPOLATION} = \begin{bmatrix} \{\text{LINEAR}\} \\ \text{SEMILOG} \\ \text{LOGLOG} \end{bmatrix} \quad \begin{array}{l} \text{Applicable analysis type:} \\ \text{RANDOM, FREQUENCY, SHOCK} \end{array}$$

where,

- LINEAR : linear along ordinate and abscissa.
- SEMILOG : logarithmic along ordinate, linear along abscissa.
- LOGLOG : logarithmic along ordinate and abscissa.

The interpolation scheme is used to compute the input power spectral density, amplitude spectrum, or shock spectrum at intermediate frequency points.

INTEGRATION -Specify integration procedure

$$\text{INTEGRATION} = \left[\begin{array}{l} \{\text{NUMERICAL, SIMPSON}\} \\ \text{NUMERICAL, TRAPEZOIDAL} \\ \text{EXACT} \end{array} \right] \quad \begin{array}{l} \text{Applicable analysis} \\ \text{type: RANDOM} \end{array}$$

where,

- NUMERICAL, SIMPSON : use the numerical Simpson integration procedure
- NUMERICAL, TRAPEZOIDAL : use the numerical trapezoidal integration procedure
- EXACT : use the closed form exact integration for flat input PSDs.

For most problems, the Simpson numerical integration option gives the best results. However, the trapezoidal numerical integration option is recommended if the selected natural frequencies are far apart (this is likely when rigid body modes are present). The exact integration option requires the input PSD to be flat over the frequency range (i.e., white noise). An average PSD value will be calculated internally for a general shape PSD (i.e., colored noise) input if this option is selected. The exact integration option is not available for structural damping (DAMPING = STRUCTURAL) in this version.

MRESPONSE - Control modal response printout

$$\text{MRESPONSE} = \left[\begin{array}{l} \{\text{ON}\} \\ \text{OFF} \end{array} \right] \quad \begin{array}{l} \text{Applicable analysis type:} \\ \text{FREQUENCY} \end{array}$$

where,

- ON: switch on printout of modal responses for generalized displacement, velocity and acceleration
- OFF: suppress modal response printout.

STARTINGTIME - To specify starting time

$$\text{STARTINGTIME} = \left[\begin{array}{l} t \\ \{\text{0.0}\} \end{array} \right] \quad \begin{array}{l} \text{Applicable analysis type:} \\ \text{TRANSIENT, RANDOM} \end{array}$$

where,

t: the starting time for time integration

TSEISMIC- Time duration of Excitation

TSEISMIC = $\begin{bmatrix} \text{TSEIS} \\ \{30.0\} \end{bmatrix}$ Applicable analysis type:
SHOCK, RANDOM

For random vibration analysis, it is the time duration of the stationary excitations and will be used in calculating PSD Functions of von Mises stresses and statistical properties (Refer *PSDOUT).

8.4 Title

8.4.1 *TITLE Data Group - Problem Title

Applicable analysis types: TRANSIENT, RANDOM, FREQUENCY, SHOCK

This data group is optional and is used to specify a title for the analysis. The title is printed at the top of every page of the output. This data group is the same as in [Section 6.2.1](#) but is repeated here for completeness. It is recommended that this data group be the first group in the analysis data block so that the title of the problem may also be printed in the input echo.

Group ID card:

*TITLE

Title card set:

Entry No: 1
 Variable:

TITLE

 Max char: 80

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	TITLE	alphanumeric title (up to 80 characters).

Note:

- Up to 6 cards (80 characters per card) may be used for the title. The contents of all cards will be printed when this data group is read in. The contents of the first card will be listed as a heading on all subsequent pages of the output file.

8.5 Analysis Control Data

8.5.1 *MODESELECTION Data Group - Mode Selection Data

Applicable analysis types: TRANSIENT, RANDOM, FREQUENCY, SHOCK

This data group is always required and is used to specify the modes to be included in the analysis.

Group ID card:

*MODESELECTION

Mode selection card set:

	1	2	3	\$
Entry No:				
Variable:	MODE1	MODE2	INCR	IDAMP
Max char:	5	5	5	5

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	MODE1	first mode number to be selected for analysis
2-3	MODE2, INCR	automatic generation parameters. Modes from MODE1 through MODE2 in increments of INCR will be selected for analysis (default values are MODE1 and zero, respectively).

--- tab (\$) ---

4	IDAMP	for modal viscous or modal structural damping (see note 1), this is the damping identification number. All modes selected in this card will be assigned a damping value whose ID in DAMPING data group is IDAMP. for proportional viscous damping, this entry is not used.
---	-------	---

Notes:

1. For random vibration analysis, all the selected modes must have non-zero damping IDs. For other analysis types, this ID may be left blank or entered as zero indicating zero damping for the modes selected.
2. Additional cards may be provided, if required, to specify other ranges of modes to be selected for the analysis.

8.5.2 *DAMPING Data Group - Damping Values

Applicable analysis types: TRANSIENT, RANDOM, FREQUENCY, SHOCK

This data group is always required for random vibration analysis and optional for other analysis types. It is used to specify the damping values.

Group ID card:

*DAMPING

Damping Card set:

Entry No:	1	2	3
Variable:	IDAMP	DAMP1	DAMP2
Max Char:	5	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IDAMP	identification number for damping referred in *MODESELECTION data group, see note 2.
2	DAMP1	ratio of damping to critical damping if modal-viscous damping is used (see note 1). structural damping coefficient if modal structural damping is used (see note 1). constant C_1 if proportional viscous damping is used (see note 2).
3	DAMP2	not used for modal-viscous/modal-structural damping. constant C_2 if proportional viscous damping is used (see note 2).

Notes:

1. Damping type is selected in the executive commands. Structural damping is available only in random vibration and frequency response analyses.
2. For proportional viscous damping, the damping matrix is calculated as a linear combination of the stiffness and mass matrices:

$$C = C_1 K + 2 C_2 M$$

Since only one set of C_1 and C_2 values are required, only the first set of values will be used if more than one card is provided, regardless of the damping identification number. However, the ID number (IDAMP) *must* be non-zero.

3. For random vibration analysis, the damping values should be non-zero.
4. Additional cards, if required, may be provided to specify different damping ratios.

8.5.3 *TSTEP Data Group - Time Step Data

Applicable analysis types: TRANSIENT

This data group is optional and may be used to specify variable time step for time integration.

Group ID card:

*TSTEP

Time step card set:

Entry No:	1	2	3			
Variable:	<table border="1"><tr><td>TIME1</td></tr></table>	TIME1	<table border="1"><tr><td>TIME2</td></tr></table>	TIME2	<table border="1"><tr><td>DELT</td></tr></table>	DELT
TIME1						
TIME2						
DELT						
Max char:	10	10	10			

Entry Variable Description

- | | | |
|---|-------|---|
| 1 | TIME1 | starting time for the current region (must be greater than or equal to the starting time for the analysis specified in the executive commands). |
| 2 | TIME2 | ending time for the current region (must be less than or equal to ending time for the analysis specified in the executive commands). |
| 3 | DELT | time step increment to be used for time integration in this region. |

Notes:

1. For time regions not covered in this data group the default time step specified in the executive commands will be used.
2. Additional cards may be provided, if required, to define time increments in other regions.
3. If there is an overlap between two regions, the first definition of time increment will be used in the overlapping region.

8.5.4 *ADDFREQUENCY Data Group - Additional Exciting Frequencies

Applicable analysis types: RANDOM, FREQUENCY

This data group is optional and may be used to specify exciting frequencies in addition to those generated internally by setting GENFREQUENCY = ON in the executive commands.

Group ID card:

*ADDFREQUENCY

Additional frequency card set:

Entry No:	1	2	3
Variable:	FREQ1	FREQ2	NINCR
Max char:	10	10	5

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	FREQ1	value of starting frequency in cycles/time.
2-3	FREQ2, NINCR	automatic data generation parameters - frequencies starting from FREQ1 and ending with FREQ2, and at NINCR equal increments between them will be generated (default values are FREQ1 and zero, respectively).

Notes:

1. In random vibration analysis, additional frequency points specified in this group will be added to the exciting frequency list used in numerical integration and/or response PSD computations.
2. In random vibration analysis, it is recommended that additional frequency points be used in regions of abrupt changes (discontinuities) in the forcing PSD curves in order to minimize the numerical integration errors or if the response PSD is desired at certain specific frequencies.
3. In frequency response analysis, the user can specify any number of exciting frequencies for which the response is required.
4. These are the only frequency points for which the response is computed if the GENFREQUENCY = OFF command is included in the executive commands.
5. Frequencies less than FLOWER or greater than FUPPER will be ignored. FLOWER and FUPPER are the lower and upper cutoff frequencies specified in the executive commands.

8.5.5 *INITIAL Data Group - Initial Condition Data

Applicable analysis types: TRANSIENT

This data group is optional and may be used to specify initial displacements and velocities at specific nodes of the structure.

Group ID card:

*INITIAL

Initial condition card set:

Entry No:	1	2	3	4	5
Variable:	NODE	LABEL	VARIABLE	LAST	INCR
Max char:	8	4	12	8	8

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NODE	node number
2	LABEL	label for initial condition input (see the list of valid labels in Table 8.8)
3	VALUE	value for the initial condition
4-5	LAST, INCR	nodes from 'NODE' through 'LAST' in increments of 'INCR' will be assigned the same value of initial displacement or velocity implied by the 'LABEL'

Notes:

1. Initial conditions specified in this data group will be applied to the nodes of the structure at the starting time for the analysis specified using the executive command 'STARTINGTIME'.
2. Nodes not specified in this data group will be assigned a zero value for the initial displacement or velocity.
3. NISA files 30, 32, and 35 must be saved from NISA eigenvalue analysis whenever the data group is used.
4. If a local displacement coordinate system is defined at a particular node, then the initial conditions must be described in the local coordinate system, otherwise all initial displacements and velocities are in the global coordinate system.

5. Additional cards may be provided, if necessary to specify initial displacements or velocities at other nodes.

Table 8.8: List of valid labels for *INITIAL group

UX UY UZ	Displacement in X direction Displacement in Y direction Displacement in Z direction
ROTX ROTY ROTZ	Rotation about X axis Rotation about Y axis Rotation about Z axis
VX VY VZ	Velocity in X direction Velocity in Y direction Velocity in Z direction
OMGX OMGY OMGZ	Rotation velocity about X axis Rotation velocity about Y axis Rotation velocity about Z axis

8.5.6 *MATDAMPING Data Group - Material Damping Data

Applicable analysis types: TRANSIENT, RANDOM, FREQUENCY, SHOCK

This data group is optional and may be used to specify the damping values depending on material.

Group ID card: ***MATDAMPING**

Entry No:	1	2	3	4	5	6	7	8	9	10
Variable:	MATID	LABEL	NATURE	CONST	DAMP1	DAMP2	DAMP3	DAMP4	DAMP5	DAMP6
Max char:	6	2	2	10	10	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	MATID	Material ID (referred in *MATERIAL data card)
2	LABEL	Energy of the subsystem on which the dissipation id is dependent KE - Kinetic Energy, SE - Strain Energy
3	NATURE	Nature of damping (see note 1 & 2) 1 - Modal damping 2 - Proportional damping
4	CONST	constant of proportionality in all directions (note 3)
5	DAMP1	constant of proportionality in global X-direction (note 4)
6	DAMP2	constant of proportionality in global Y-direction
7	DAMP3	constant of proportionality in global Z-direction
8	DAMP4	constant of proportionality in global ROTX-direction
9	DAMP5	constant of proportionality in global ROTY-direction
10	DAMP6	constant of proportionality in global ROTZ-direction

Notes:

1. If NATURE is proportional damping, the effective damping for each mode is computed according to the sum of the ratios of maximum kinetic/strain energy of each subsystem to the maximum kinetic/strain energy of the complete system (Ref.).
2. If NATURE is modal damping, the effective damping for each mode is computed according to ratio of maximum kinetic or strain energy of each subsystem to the maximum kinetic or strain energy of the complete system (Ref.).
3. If the constant of proportionality is same in all directions, user can provide this one value through the input CONST.
4. The constant of proportionality may be different in different directions. DAMP1 to DAMP6 are to be used in such a case.
5. User can provide proportional damping using KE, or SE or both KE and SE. In the case of both KE and SE to be considered for computation of damping data user has to provide one card with KE data and a second card with SE data. In this case the constant of proportionality is mass proportional constant for KE and stiffness proportional constant for SE.
6. If the NATURE is modal damping only one card either with KE or SE is to be provided. In this case the constant of proportionality is the % critical damping (e.g. for 2% critical damping is to be entered as 0.02).
7. User has to save Element stiffness and mass matrices data files (#23.dat & #24.dat) in eigen value analysis.
8. Material damping can be applied to even those elements which do not require material properties to be specified (for example, spring elements). For such elements, a dummy material property table should be given. The material ID used for the element should be specified in eigenvalue analysis to be used later in the modal dynamic analysis.
9. The effective damping for each mode is based on the sum of the ratios of dissipated kinetic energy/strain energy of each material to the total kinetic/strain energy of that mode.
10. The constant or the constants of proportionality for proportional damping are the coefficients α or β in proportional damping equation $[C] = \alpha [M] + \beta [K]$. The constant for modal damping is the damping ratio ζ .

Reference

ASME, Section III, 'Boiler and Pressure Vessel Code', ASME, SECTION III, DIVISION I -APPENDICES, APPENDIX-N, (N1233.2).

8.5.7 *FLOORRESPONSE Data Group – Floor Response Spectra Data

Applicable analysis type: TRANSIENT

This data group is optional and can be used to request floor response spectra computation for a given set of damping ratios.

Floor Response Spectra Card Set:

Group ID card:

*FLOORRESPONSE

\$

Entry No:	1	2	3	4	5	6	7
Variable:	FREQ1	FREQ2	NINCR	XFACT	YFACT	ZFACT	NDAMP
Max Char:	10	10	5	10	10	10	5

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	FREQ1	value of starting frequency in cycles/time (FREQ1>0)
2-3	FREQ2, NINCR	automatic data generation parameters – frequencies starting from FREQ1 and ending with FREQ2, and at NINCR equal increments between them will be generated (default values are FREQ1 and zero, respectively)
--- tab(\$) ----		
4	XFACT	multiplying factor for the time function in the global X-direction (default = 1.0)
5	YFACT	multiplying factor for the time function in global Y-direction (default=1.0)
6	ZFACT	multiplying factor for the time function in global Z-direction (default=2/3)
7	NDAMP	number of damping ratios for which the floor response spectra is to be computed (default=0) (note 2)

Card Set 2: More than one card if the NDAMP > 0

Entry No:	1	2	3	4-7	8
Variable:	DAMP1	DAMP2	DAMP3	DAMP8
Max Char:	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1-8	DAMP1	Damping ratio (viscous) (note 3)
-		
	DAMP8	

Notes:

1. Floor response spectra are generated at NINCR frequency points in the frequency range between FREQ1 and FREQ2 specified in data card.
2. If NDAMP is not given, the damping ratio is taken as zero for computing the FRS.
3. Acceleration floor response spectra are generated for the NDAMP damping ratios.
4. For the nodes specified in *RSET card and for a specified damping ratio the floor response spectra are generated according to the following steps:
 - Case 1. If A(t), B(t), and C(t) are the three independent input acceleration time histories specified in *TIMEFUNCTION card, six sets of input ground excitation in global X, Y and Z directions are automatically set up in the order (A, B, C), (A, C, B), (B, A, C), (B, C, A), (C, A, B) and (C, B, A). All three time functions (X, Y, Z) should be explicitly specified.
 - Case 2. Acceleration response histories (relative to support) are computed by modal summation procedure at requested set of nodes and directions for each of the above six sets of ground excitation. Missing Mass correction is added to the Acceleration response histories provided the CFREQ (Hz) is specified in the executive data card.
 - Case 3. Absolute responses are computed by addition of the relative response quantities and the input base excitation time histories.
 - Case 4. Average, maximum and average + 1 sigma acceleration response spectra are generated for the nodes specified in *RSET card

8.6 Load Function Description Data

8.6.1 *ARRIVALTIME Data Group - Arrival Time Data

Applicable analysis types: TRANSIENT

This data group is optional and is used to specify several arrival times to be used in ground motion, concentrated forces and/or pressure data.

Group ID card:

*ARRIVALTIME

Arrival time card set:

Entry No:	1	2		
Variable:	<table border="1"><tr><td>IARR</td><td>ARRT</td></tr></table>	IARR	ARRT	
IARR	ARRT			
Max char:	5	10		

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IARR	arrival time identification number. This number is referenced in *GROUND, *DPRESSURE and/or *DCFORCE data groups.
2	ARRT	arrival time value.

Note:

1. Additional cards may be provided, if required, to specify other arrival times. Each arrival time should have a unique ID. If the ID numbers are repeated, then the first definition is used.

8.6.2 *TIMEFUNCTION Data Group - Time Function Definition

Applicable analysis types: TRANSIENT

This data group is always required and is used to specify several time dependent functions to be used in ground motion, concentrated forces and/or pressure data.

Each time function is defined using three sets of cards. The first and the second sets contain one card each and are used to specify the title and control parameters, respectively. The third card set consists of one or more cards and is used to define the time function.

Group ID card:

*TIMEFUNCTION

Card set 1: Time function title card

Entry No: 1
 Variable:

TITLE

 Max char: 80

Entry Variable Description

1 TITLE alphanumeric description of the time function. This title appears in the input echo.

Card set 2: Time function control card

\$

Entry No:	1	2	3	4	5	6
Variable:	ID	NP	NTABLE	SFTR	START	DELT
Max char:	5	5	5	10	10	10

Entry Variable Description

1 ID time function identification number to be referenced in *GROUND, *DPRESSURE and/or *DCFORCE data groups.

2 NP number of points used to define the time function (must be greater than or equal to 2).

Modal Dynamic Analysis Data

Load Function Description Data

- 3 NTABLE table form to be used to input time function in card set 3.
 = 0 - if the time function is defined in terms of abscissa and ordinates
 (T1, F1, T2, F2, etc.).
 = 1 - if the time function is defined at equal intervals.
- tab (\$) ---
- 4 SFTR scale factor used for the time function data (default = 1.0).
- 5 START starting time for the load function if NTABLE = 1 (not used for
 NTABLE = 0).
- 6 DELT time increment used for the time function if NTABLE = 1 (not used for
 NTABLE = 0).

Card set 3: Time function definition cards

For NTABLE = 0

The time value and the function value should be entered in pairs up to a maximum of three pairs per card. The time values should be entered in ascending order and must be positive except at the first point where the time value can be zero. Enter only NP pairs.

Entry No:	1	2	3	4	5	6
Variable:	T1	F1	T2	F2	T3	F3
Max char:	10	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	T1	time at point 1.
2	F1	function value at point 1.
3	T2	time at point 2.
4	F2	function value at point 2.
5	T3	time at point 3.
6	F3	function value at point 3, if needed.

For NTABLE = 1

The time function values at equal intervals are entered (6 points per card). Enter as many cards as required to define NP number of points.

Entry No:	1	2	3	4	5	6
Variable:	F1	F2	F3	F4	F5	F6
Max char:	10	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	F1	function value at point 1 (at time = START).
2	F2	function value at point 2 (at time = START + DELT).
3	F3	function value at point 3 (at time = START + 2.0 * DELT), if needed.
4	F4	function value at point 4 (at time = START + 3.0 * DELT), if needed.
5	F5	function value at point 5 (at time = START + 4.0 * DELT), if needed.
6	F6	function value at point 6 (at time = START + 5.0 * DELT), if needed.

Notes:

1. Any number of time functions can be defined by repeating card sets 1 to 3. Each time function should have a unique ID.
2. Amplitudes corresponding to time values outside the specified range will be assumed zero, whereas amplitudes within the specified time range will be linearly interpolated.

8.6.3 *PSDFUNCTION Data Group - Power Spectral Density (PSD)

Function Definition

Applicable analysis types: RANDOM

This data group is always required. The one-sided auto PSD function and the real part of the one sided cross PSD function may be defined using this data group.

Each PSD function is defined using three sets of cards. The first and the second sets contain one card each and are used to specify the title and control parameters, respectively. The third card set is required if the PSD is not defined as a constant value, and it may consist of one or more cards.

Group ID card:

*PSDFUNCTION

Card set 1: PSD function title card

Entry No: 1
 Variable:

TITLE

 Max char: 80

Entry Variable Description

1 TITLE alphanumeric description of PSD function. This title appears in the echo of the input.

Card set 2: PSD function control card

\$

Entry No:	1	2	3	4	5	6
Variable:	ID	NP	NTABLE	SFTR	DATA1	DATA2
Max char:	5	5	5	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	ID	PSD function identification number to be referenced in *GROUND, *CORRELATION, *DPRESSURE and/or *DCFORCE data groups.
2	NP	number of points used to define the function (≥ 2). Enter zero for constant PSD (NTABLE = 2).
3	NTABLE	table form to be used to input the PSD forcing function. = 0 - frequency versus real PSD value. = 1 - center frequency of percentage octave band versus the power level for each band in decibels. = 2 - constant value of PSD is given, i.e., the PSD forcing function has a constant value over the frequency range of interest (does not require card set 3). = 3 - frequency versus real and imaginary components of a complex PSD function
--- tab (\$) ---		
4	SFTR	scale factor used for the PSD function data (default = 1.0).
5	DATA1	this variable is used to complete the definition of the PSD function in card set 3. It is not used for NTABLE = 0 or 3. The variable is: <u>if NTABLE = 1</u> PERCNT - the percentage octave band. Default value = 0.333 (if zero or a blank is entered, one third octave band is assumed). <u>if NTABLE = 2</u> P ₀ - the constant value of the forcing PSD.
6	DATA2	this variable is used to complete the definition of the PSD function in card set 3. It is used only for NTABLE = 1. The variable is: <u>if NTABLE = 1</u> REFVAL - the reference value used in expressing the power level in decibels. This could be a reference value for pressure, force, or acceleration level, but is usually for acoustic pressure. If zero or a blank is entered, a reference pressure value of 2.9×10^{-9} lb/in ² is assumed.

Modal Dynamic Analysis Data

Load Function Description Data

Card set 3: PSD function definition cards

This card set is required for $NTABLE = 0, 1$ or 3 . The abscissa (frequency or center frequency) and the ordinates (PSD or power level) should be entered in sets *up to* a maximum of three sets (for $NTABLE=0$ or 1) or two sets (for $NTABLE=3$) per card. Enter only NP sets.

For $NTABLE = 0$ or 1

Entry No:	1	2	3	4	5	6
Variable:	F1	P1	F2	P2	F3	P3
Max char:	10	10	10	10	10	10

Entry Variable Description

- | | | |
|---|----|--|
| 1 | F1 | - frequency in cycles/time at point 1 (for $NTABLE = 0$).
- center frequency in cycles/time for point 1 (for $NTABLE = 1$). |
| 2 | P1 | - PSD function in (unit of excitation) ² /(cycles/time) at point 1 (for $NTABLE = 0$).
- power level in decibels for point 1 (for $NTABLE = 1$). |
| 3 | F2 | - frequency in cycles/time at point 2 (for $NTABLE = 0$).
- center frequency in cycles/time for point 2 (for $NTABLE = 1$). |
| 4 | P2 | - PSD function in (unit of excitation) ² /(cycles/time) at point 2 (for $NTABLE = 0$).
- power level in decibels for point 2 (for $NTABLE = 1$). |
| 5 | F3 | - frequency in cycles/time at point 3 (for $NTABLE = 0$).
- center frequency in cycles/time for point 3 (for $NTABLE = 1$). |
| 6 | P3 | - PSD function in (unit of excitation) ² /(cycles/time) at point 3 (for $NTABLE = 0$).
- power level in decibels for point 3 (for $NTABLE = 1$). |

For NTABLE = 3

Entry No:	1	2	3	4	5	6
Variable:	F1	R1	I1	F2	R2	I2
Max char:	10	10	10	10	10	10

Entry	Variable	Description
1	F1	- frequency in cycles/time at point 1
2	R1	- real component of PSD function at point 1 in (unit of excitation) ² /(cycles/time)
3	I1	- imaginary component of PSD function at point 1 in (unit of excitation) ² /(cycles/time)
4	F2	- frequency in cycles/time at point 2
5	R2	- real component of PSD function at point 2 in (unit of excitation) ² /(cycles/time)
6	I2	- imaginary component of PSD function at point 2 in (unit of excitation) ² /(cycles/time)

Notes:

1. The frequency values should be entered in *ascending* order. They must be positive, except for the first point which can be zero. For NTABLE = 1, the center frequency cannot be specified as zero.
2. Any number of PSD functions may be defined by repeating card sets 1 to 3. Each PSD function should have a unique ID.

8.6.4 *CORRELATION Data Group - Correlation Ensemble Data

Applicable analysis types: RANDOM

This data is required if two or more input processes are correlated with each other.

Group ID card:

*CORRELATION

Correlation card set:

	\$				
Entry No:	1	2	3	4-13	14
Variable	ID	NPRCS	NFN1	...	NFN12
Max char:	5	5	5	...	5

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	ID	correlation ensemble identification number to be referenced in *GROUND, *DPRESSURE and/or *DCFORCE groups.
2	NPRCS	total number of processes in this correlated ensemble.
---tab(\$)--		
3-14	NFN1 to NFN12	entries of the cross spectral density matrix for this ensemble. Each entry is to be referenced to a PSD function ID in the *PSD-FUNCTION group (see note 1).

Notes:

1. This data group specifies one or more ensembles of correlated input sources. The processes in each ensemble will be considered fully or partially correlated with each other but uncorrelated with any other ensemble. The correlated ensemble will be represented by a cross spectral density matrix. An entry (i, j) in the matrix points to the cross PSD between process i and j, and a diagonal (i, i) entry points to the auto spectral density for process i. The entries represent the upper triangle of the symmetric cross-PSD matrix assigned row-wise, moving away from the diagonal. A total of NPRCS * (NPRCS + 1) / 2 number of entries must be defined. An off-diagonal entry (i, j) may be zero if processes i and j are not correlated, but the diagonal entries must be non-zero and must ref-

reference a *PSD function ID number defined in *PSDFUNCTION group. See Example 8.5.1.

2. If more than 12 entries of the cross spectral density matrix are needed, continue on additional cards starting with a tab (\$) character, specifying 12 entries per card.
3. Additional cards, if required, may be provided to specify other ensembles of fully or partially correlated processes. Each ensemble must have a unique correlation ensemble ID.

Example 8.5.1:

A simple problem is posed to illustrate if, and how, this data group may be employed. Suppose five input processes are to be defined in terms of their auto- and cross-spectral density functions given by a cross spectral density matrix as follows:

Process	A	B	C	D	E
A	$\begin{bmatrix} 1001 & 0 & 0 & 0 & 0 \\ 0 & 1002 & 1002 & 0 & 0 \\ 0 & 1002 & 1002 & 0 & 0 \\ 0 & 0 & 0 & 1003 & 1005 \\ 0 & 0 & 0 & 1005 & 1004 \end{bmatrix}$				
B					
C					
D					
E					

in which 1001, 1002, 1003, 1004 and 1005 are IDs of PSD functions defined in the *PSDFUNCTION data group.

Method 1

For NPRCS = 5, the upper triangle of the PSD matrix is input (row-wise) as follows:

1st card: 2001, 5, 1001, 0, 0, 0, 0, 1002, 1002, 0, 0, 1002, 0, 0

2nd card: \$1003, 1005, 1004

A single correlation ensemble (ID = 2001) with fifteen ($5 [5 + 1] / 2 = 15$) entries has been defined. However, it may be seen from the PSD matrix that process A is not correlated to any of the other processes. Thus, PSD function 1001 may be directly referenced in

Modal Dynamic Analysis Data

Load Function Description Data

*GROUND, *DCFORCE and/or *DPRESSURE groups, and not be included in this data group.

Method 2

Hence, the definition of correlation may be reduced to four processes (NPRCS = 4) given by a PSD matrix as follows:

Process	B	C	D	E
B	$\begin{bmatrix} 1002 & 1002 & 0 & 0 \\ 1002 & 1002 & 0 & 0 \\ 0 & 0 & 1003 & 1005 \\ 0 & 0 & 1005 & 1004 \end{bmatrix}$			
C				
D				
E				
E				

for which the input may be given as follows:

1st card: 2002, 4, 1002, 1002, 0, 0, 1002, 0, 0, 1003, 1005, 1004

The PSD function 1001 will be referenced directly in *GROUND, *DCFORCE and/or *DPRESSURE card.

Here we have a single correlation ensemble (ID = 2001) with ten entries. Again we may see that processes B and C are not correlated to processes D and E although they are correlated to each other (In fact, processes B and C are fully correlated). The user may thus define two ensembles each consisting of two processes.

Method 3

The resulting two ensembles are:

$$\begin{array}{cc} \text{Process B} & \text{C} \\ \text{B} & \begin{bmatrix} 1002 & 1002 \\ 1002 & 1002 \end{bmatrix} \\ \text{C} & \end{array} \quad \text{and} \quad \begin{array}{cc} \text{Process D} & \text{E} \\ \text{D} & \begin{bmatrix} 1003 & 1005 \\ 1005 & 1004 \end{bmatrix} \\ \text{E} & \end{array}$$

for which the input may be given in terms of two ensembles, each consisting of two processes as follows:

1st card: 2003, 2, 1002, 1002, 1002
2nd card: 2004, 2, 1003, 1005, 1004

The PSD function 1001 will be referenced directly in *GROUND, *DCFORCE and/or *DPRESSURE groups.

Any of the above three methods are acceptable forms of input. Method 1 will result in a single uncorrelated group during modal response calculations, whereas, Method 3 will result in three uncorrelated groups. However, the size of the PSD matrix in Method 1 (5 x 5) is greater than that in Method 2 (2 x 2). Proper judgement should be exercised to determine whether decreasing the number of uncorrelated groups or minimizing the size of the PSD matrix will reduce the cost for the problem at hand.

8.6.5 *SPECTRUM Data Group - Spectrum Definition Data

Applicable analysis types: FREQUENCY, SHOCK

This data group is always required and may be used to define amplitude and/or phase spectra in frequency response analysis, and response spectra in the case of shock spectrum analysis. Each spectrum is defined using three sets of cards. The first and the second sets contain one card each and are used to specify the title and control parameters, respectively. The third card set is required if the spectrum is not defined as a constant value, and it may consist of one or more cards.

Group ID card:

*SPECTRUM

Card set 1: Spectrum title card

Entry No: 1
 Variable:

TITLE

 Max char: 80

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
--------------	-----------------	--------------------

1	TITLE	alphanumeric description of the spectrum. This title appears in the echo of the input.
---	-------	--

Card set 2: Spectrum control card

Entry No:	1	2	3	4	5
Variable:	ID	NP	NTABLE	SFTR	CONST
Max char:	5	5	5	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
--------------	-----------------	--------------------

1	ID	spectrum identification number to be referenced in *GROUND, *DCFORCE, *DPRESSURE and/or *MDSPECTRUM data groups.
2	NP	number of points used to define the spectrum(≥ 2). For constant spectrum (NTABLE = 3), this entry is ignored.

- 3 NTABLE table form to be used in input spectrum.
 = 0 -spectrum is a function of frequency (cycles/time).
 = 1 -spectrum is a function of frequency (radians/time).
 = 2 -spectrum is a function of period.
 = 3 -spectrum is constant.
- 4 SFTR scale factor used for the spectrum data (default value = 1.0).
- 5 CONST spectrum value (used only if NTABLE = 3).

Card set 3: Spectrum definition cards

Entry No:	1	2	3	4	5	6
Variable:	F1	S1	F2	S2	F3	S3
Max char:	10	10	10	10	10	10

This card set is required if NTABLE = 0, 1 or 2. The abscissa (frequency or period) and the ordinates (spectrum values) should be specified in pairs *up to* a maximum of 3 pairs per card. Enter only NP pairs.

Entry Variable Description

- 1 F1 - frequency (cycles/time) at point 1 (for NTABLE = 0).
 - frequency (radians/time) at point 1 (for NTABLE = 1).
 - period (time) at point 1 (for NTABLE = 2).
- 2 S1 - spectrum value at point 1
- 3 F2 - frequency (cycles/time) at point 2 (for NTABLE = 0).
 - frequency (radians/time) at point 2 (for NTABLE = 1).
 - period (time) at point 2 (for NTABLE = 2).
- 4 S2 - spectrum value at point 2.
- 5 F3 - frequency (cycles/time) at point 3 (for NTABLE = 0).
 - frequency (radians/time) at point 3 (for NTABLE = 1).
 - period (time) at point 3 (for NTABLE = 2).
- 6 S3 - spectrum value at point 3.

Notes:

1. For shock spectrum analysis, it must be ensured that the lowest and highest natural frequencies of the modes selected in the analysis lie within the specified frequency range of the spectrum.
2. The abscissa values (period/frequency) must be specified in *ascending* order. They must be positive values, except for the first point which can be zero.
3. Any number of spectra can be specified by repeating card sets 1 to 3. Each spectrum must have a unique ID.

8.6.6 *MDSPECTRUM Data Group - Multiple Damping Spectrum Data

Applicable analysis types: SHOCK

This data group is required only when the shock spectrum is dependent on damping. Groups of damping dependent response spectra can be specified using this data group.

Group ID card:

*MDSPECTRUM

Multiple damping spectrum card set:

Entry No:	1	2	3
Variable:	IDMDS	ISPEC	DAMP
Max char:	5	5	10

Entry Variable Description

- | | | |
|---|-------|--|
| 1 | IDMDS | multiple damping spectrum identification number. This number is referenced in the *GROUND data. |
| 2 | ISPEC | spectrum identification defined in *SPECTRUM data. The spectrum curve with the ID number ISPEC is included in the multiple damping spectrum. |
| 3 | DAMP | ratio of damping to critical damping assigned to the spectrum curve ISPEC. |

Notes:

1. Each multiple damping spectrum ID refers to a group of spectra, which may have any number of spectra assigned to specific damping values. All groups of multiple damping spectra referenced in *GROUND data should cover the entire range of modal damping values used in the analysis. See Example 8.5.2.
2. Any number of groups of multiple damping spectra can be specified using additional cards each group having a unique ID.

Example 8.5.2

Two groups of spectra are used in this example to specify the vertical (Y-direction) and horizontal (X-direction) components of ground motion. The vertical component is independent of damping and has the response spectra defined with an ID 1001. The horizontal component is dependent on damping and the spectra for damping ratios of 0.01, 0.02 and 0.05 are defined with IDs 2001, 2002 and 2003, respectively. The *SPECTRUM, *MDSPECTRUM and *GROUND data groups for the above problem can be entered as given below.

```
...
*SPECTRUM
RESPONSE SPECTRA FOR VERTICAL COMPONENT
1001, 10, 0, 1.0
...
...
...
...
RESPONSE SPECTRA FOR HORIZONTAL COMPT - DAMPING = 0.01
2001, 6, 2, 18.1
...
...
RESPONSE SPECTRA FOR HORIZONTAL COMPT - DAMPING = 0.02
2002, 6, 2, 12.3
...
...
RESPONSE SPECTRA FOR HORIZONTAL COMPT - DAMPING = 0.05
2003, 6, 2, 9.5
...
...
*MDSPECTRUM
** HORIZONTAL SPECTRA ARE COUPLED TOGETHER
** UNDER MULTIPLE DAMPING SPECTRUM ID NO. 1002
1002, 2001, 0.01
1002, 2002, 0.02
1002, 2003, 0.05
*GROUND
UX, 1002, 0, 1.0
```

UY, 0, 1001, 1.0

...

Here the ground motion card for horizontal direction (X-direction) refers to the multiple damping spectrum group ID 1002 defined in *MDSPECTRUM data group. But the card for the vertical direction (Y-direction) refers directly to the spectrum ID 1001 defined in *SPECTRUM data group. (See [Section 8.7.1](#) and *GROUND data group in [Section 8.7.2](#) for a more detailed description). In this problem, it needs to be ensured that the modal damping ratios are between 0.01 and 0.05.

8.6.7 *NONSTATIONARY Data Group - PSD and the Corresponding Envelope Function Definition

Application analysis types: RANDOM

This data group is optional and is used to specify the PSD and the corresponding envelope function IDs for nonstationary input.

Group ID card:

*NONSTATIONARY

Nonstationary card set:

Entry No:	1	2
Variable:	IDPSD	IDENV
Max char:	5	5

Entry Variable Description

- | | | |
|---|-------|--|
| 1 | IDPSD | PSD function identification number to be referenced in *PSDFUNCTION data group |
| 2 | IDENV | envelope function identification number to be referenced in *ENVELOPE data group |

Note:

1. Additional cards may be provided, if required, to specify other sets of PSD and corresponding envelope functions.

8.6.8 *ENVELOPE Data Group - Envelope Function Definition

Application analysis types: RANDOM

This data group is optional and may be used to specify the nonstationary input loads.

Each envelope function is defined using three sets of cards. The first and the second sets contain one card each and are used to specify the title and control parameters, respectively. The third card set is required if the envelope is not defined as a constant value, and it may consist of one or more cards.

Group ID card:

*ENVELOPE

Card set 1: envelope function title card

Entry No: 1
 Variable:

TITLE

 Max char: 80

Entry Variable Description

1 TITLE alphanumeric description of the time function. This title appears in the input echo.

Card set 2: envelope function control card

Entry No: 1 2 3 4 5 6
 Variable:

ID	NP	NTABLE	SFTR	DATA1	DATA2
----	----	--------	------	-------	-------

 Max char: 5 5 5 10 10 10

Entry Variable Description

1 ID PSD function identification number to be referenced in *GROUND, *CORRELATION, *DPRESSURE and/or *DFORCE data groups
 2 NP number of points used to define the function (for NTABLE = 2). Enter zero for NTABLE = 0 or 1.

- 3 NTABLE table form to be used to input envelope function
 = 0 constant envelope function (does not require card set 3).
 = 1 to specify the envelope function using alpha and beta
 = 2 if the envelope function is defined in terms of abscissa and ordi-
 nates (T1, E1, T2, E2, etc.)
 = 3 if the envelope function is defined in equal intervals (E1, E2, E3,
 E4, etc.).
- 4 SFTR scale factor for the envelope function data (default = 1.0)
- 5 DATA1 this variable is used to complete the definition of the envelope function
 in card set 3.
 It is not used for NTABLE = 2. The variable is:
 if NTABLE = 0
 E0 - the constant value of envelope function
 if NTABLE = 1
 ALPHA - this variable is used to define the envelope function
 if NTABLE = 3
 START - starting time for the envelope function
- 6 DATA2 this variable is used to complete the definition of the envelope function
 in card set 3.
 It is not used for NTABLE = 0 or 2. The variable is:
 if NTABLE = 1
 BETA - this variable is used to define the envelope function
 if NTABLE = 3
 DELT - time increment used for the envelope function

Modal Dynamic Analysis Data

Load Function Description Data

Card set 3: envelope function definition cards

For NTABLE = 2

The magnitude of the function should be entered in pairs up to maximum of three pairs per card. The time values should be entered in ascending order and must be positive except the first point where the time value can be zero. Enter only NP pairs.

Entry No:	1	2	3	4	5	6
Variable:	T1	E1	T2	E2	T3	E3
Max char:	10	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	T1	time at point 1
2	E1	function at point 1
3	T2	time at point 2
4	E2	function at point 2
5	T3	time at point 3
6	E3	function at point 3

For NTABLE = 3

The envelope function at equal intervals are entered (6 points per card). Enter as many as required to define NP number of points.

Entry No:	1	2	3	4	5	6
Variable:	E1	E2	E3	E4	E5	E6
Max char:	10	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	E1	function value at point 1 (at time = START)
2	E2	function value at point 2 (at time = START + DELT)
3	E3	function value at point 3 (st time = START + 2.0 *DELT), if needed
4	E4	function value at point 4 (at time = START + 3.0 *DELT), if needed
5	E5	function value at point 5 (at time = START + 4.0 *DELT), if needed
6	E6	function value at point 6 (at time = START + 5.0 *DELT), if needed

Notes:

1. Any number of envelope functions can be defined by repeating sets 1 to 3. Each envelope function should have a unique ID.
2. Amplitudes corresponding to time values outside the specified range will be assumed zero, whereas amplitudes within the specified range will be linearly interpolated.

8.6.9 *PRSPECTRA data group -Prestored Response Spectra definition data

Application analysis types: SHOCK

This data group is optional and may be used to specify the parameters describing the prestored response spectrum. Each prestored response spectrum is defined using two sets of cards. Each set contains one card. First card is used to specify the standard spectrum identification and second card control parameters.

Card set 1: Prestored Spectrum Identification Card

Entry No:	1	2
Variable:	ID	LABEL
Max char:	5	5

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
--------------	-----------------	--------------------

1	ID	Prestored spectrum identification number defined in *GROUND data group
2	LABEL	Prestored spectra label recommended by regulatory agencies EURO8 - Eurocode8 Spectra INDIA - Indian Spectra JAPAN - Japanese Spectra USNRC - USNRC Spectra

Card set 2: Prestored spectrum control parameters card

This card set is required to define the control parameters for different international standards spectra.

For LABEL = EURO8:

Entry No:	1	2	3	4
Variable:	DIR	SFTR	GA	ISOIL
Max char:	4	10	10	1

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	DIR	Horizontal or vertical excitation ('HORI' or 'VERT')
2	SFTR	Scale factor (default value = 1.0)
3	GA	Design ground acceleration
4	ISOIL	Soil class parameter (1, 2 or 3)

DIR indicates the direction of the seismic event - horizontal or vertical. SFTR is a scale factor that can be used for scaling the amplitude of the response spectrum curve if necessary (default value = 1.0). GA is the magnitude of the design ground acceleration and ISOIL, the soil class parameter. The ground acceleration is the effective peak ground acceleration in firm soil and is to be defined in 'g' units. The soil class parameter is to be defined in terms of numerals 1,2 or 3. From these two specified parameters GA and ISOIL, the elastic response spectrum S_e for a reference return period of 475 years is obtained as per note 1 below and utilized in the shock spectrum analysis as per the details in note 2.

For vertical component of the seismic action (DIR = 'VERT'), the ordinates of the spectrum curve are reduced according to the procedure detailed in note 3.

For LABEL = INDIA:

Entry No:	1	2	3	4	5	6	7	8
Variable:	DIR	SFTR	ZFACT	ISOIL	SIFACT	RRFACT	USFACT	IYEAR
Max char:	4	10	10	1	10	10	10	4

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	DIR	Horizontal or vertical excitation ('HORI' or 'VERT')
2	SFTR	Scale factor (default value = 1.0)
3	ZFACT	Zone factor (default zone II; Seismic Intensity = 0.1)
4	ISOIL	Soil class parameter (default=1)
5	SIFACT	Structural Importance factor (default =1.0)
6	RRFACT	Response Reduction Factor (default =3.0 for ordinary RC moment resisting frame)
7	USFACT	Under Ground Structure Factor (Default Factor = 0.0 i.e. Depth of structure below ground level is 0.0 meters)
8	IYEAR	Earthquake resistant design Code - Year (default: 1984)

Modal Dynamic Analysis Data

Load Function Description Data

Both the acceleration spectra specified in IS - 1893 (1984) and IS - 1893 (2002) are available as prestored data. [Figure 3.9](#) and [Figure 3.10](#) (Section 3.5.6 of *NISA User's Manual*) show the average acceleration spectra in 'g' units.

The parameters pertinent to IS - 1893(1984) are DIR and SFTR only and for IS - 1893(2002) are DIR, SFTR, ZFACT, ISOIL, SIFACT, RRFAC, USFACT, and IYEAR (see Note 4).

For LABEL = JAPAN:

Entry No:	1	2	3	4	5	6	7
Variable:	DIR	SFTR	DUM	ISOIL	FA	FV	BND
Max char:	4	10	10	1	10	10	10

Entry No:	8	9	10	11
Variable:	NRP	IFLAG	ACCL/ERMAG	EPI
Max char:	4	1	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	DIR	Horizontal or vertical excitation ('HORI' or 'VERT')
2	SFTR	Scale factor (default value = 1.0)
3	DUM	Not used
4	ISOIL	Soil class parameter (1, 2 or 3)
5	FA	Response spectrum shape parameter (default value = 2.5 - see note 3)
6	FV	Response spectrum shape parameter (default value = 2.0 - see note 3)
7	BND	Response spectrum shape parameter (default value = 0.5 - see note 3)
8	NRP	Return period in years (applicable range 20 - 500 years)
9	IFLAG	0 or 1
10	ACCL/ERMAG	Basic peak acceleration (in 'g' units) if IFLAG = 0 Earthquake magnitude if IFLAG = 1
11	EPI	Epicentral distance in km (if IFLAG = 1)

[Figure 8.3](#) shows the prestored acceleration spectra and is used according to the procedure detailed in Note 5.

For LABEL = USNRC:

The two sets of response spectra supplied in *NISA User's Manual* and made available in NISA/DA module refer to U.S. Nuclear Regulatory Commission (NRC) velocity spectra. [Figure 3.7](#) (Section 3.5.6 of *NISA User's Manual*) is for vertical seismic event and [Figure 3.8](#) for the horizontal seismic event.

Entry No: 1
Variable:

DIR

Max char: 4

Entry Variable Description

1 DIR Horizontal or vertical excitation ('HORI' or 'VERT')

Notes:

- For EUROCODE8, the elastic response spectrum S_e for a given damping ratio is defined in the following form (also see [Figure 8.2](#)):

$$S_e = \left[\begin{array}{ll} (GA)S \left\{ 1 + \frac{T}{T_B}(F_d\beta_0 - 1) \right\} & , 0 \leq T \leq T_B \\ (GA)SF_d\beta_0 & ,(T_B \leq T \leq T_C) \\ (GA)SF_d\beta_0 \left\{ \frac{T_C}{T} \right\}^{k_1} & , T_C \leq T \leq T_D \\ (GA)SF_d\beta_0 \left\{ \frac{T_C}{T_D} \right\}^{k_1} \left\{ \frac{T_D}{T} \right\}^{k_2} & , T \geq T_D \end{array} \right]$$

- The parameter GA is defined in card set 2. S is a soil parameter. The parameter β_0 is the spectral acceleration amplification factor for 5% viscous damping. T_B and T_C are the limits of the constant acceleration branch 'BC' in the [Figure 8.2](#). T_D is the value defining the start of constant displacement range of the response spectrum (see [Figure 8.2](#)). F_d is the damping correction factor and is determined from the expression $F_d = 7.0 / (2.0 + 100\eta)$. η is the value of the viscous damping ratio specified in *DAMPING data group. k_1 and k_2 are exponents influencing the spectrum shape. The values of β_0 , T_B , T_C , T_D , k_1 , k_2 and S are dependent on the soil class parameter ISOIL as shown in [Table 8.9](#).

Table 8.9: Values of the parameters describing the elastic response spectrum
 (LABEL = EURO8)

Soil class (ISOIL)	S	β_0	k_1	k_2	T_B	T_C	T_D
1	1.0	2.5	1.0	2.0	0.1	0.4	3.0
2	1.0	2.5	1.0	2.0	0.15	0.6	3.0
3	0.9	2.5	1.0	2.0	0.2	0.8	3.0

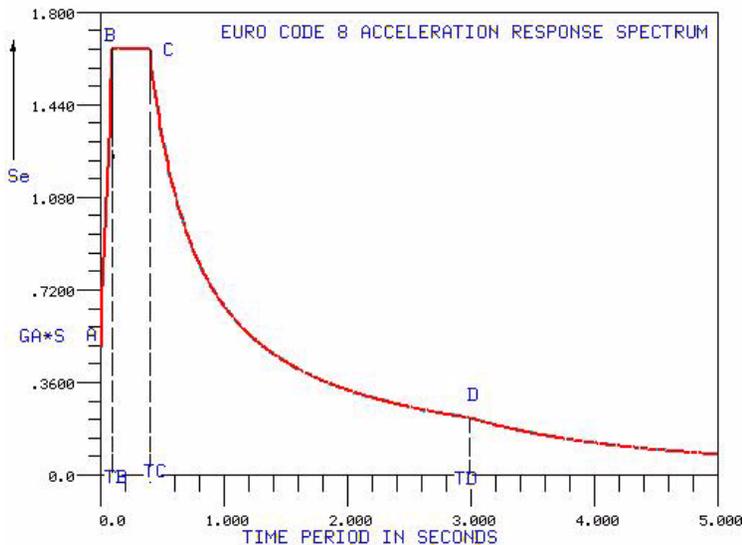


Figure 8.2: Acceleration response spectrum for LABEL = EURO8

- For LABEL = EURO8, if variable DIR = 'VERT', it is considered as a seismic action in vertical direction. In this case the spectral amplitudes are reduced as follows: For vibration periods T , smaller than 0.15 sec, the ordinates are multiplied by 0.7. For vibration periods T , greater than 0.5 sec., the ordinates are multiplied by 0.5. For vibration periods T , between 0.15 sec. and 0.5 sec., the ordinates are multiplied by a factor interpolated linearly within the range of 0.7 - 0.5.
- If LABEL = INDIA and IYEAR = 1984, the acceleration spectra definition requires only two parameters DIR and SFTR.

Direction (DIR): DIR indicates the direction of the seismic event

Scale Factor (SFTR) for IYEAR 2002:

For vertical seismic event the magnitude of the acceleration spectrum can be reduced with the help of the variable SFTR

For horizontal acceleration Spectrum, SFTR = 1.0

For vertical acceleration Spectrum, SFTR = 2/3

Scale Factor (SFTR) for IYEAR 1984:

This scale factor shall be inclusive of coefficient depending upon soil-foundation system, importance factor, seismic zone factor, and direction scale factor. These factors must be taken from IS 1893-1984 CODE.

SFTR = 1.0 (Default)

If LABEL = INDIA and IYEAR = 2002, the additional parameters are defined below.

Zone Factor (ZFACTOR):

Table Zone Factor (ZFACT)	
SEISMIC ZONE	SEISMIC INTENSITY
Zone II (Low)	0.10
Zone III (Moderate)	0.16
Zone IV (Severe)	0.24
Zone V (Very Severe)	0.36

Soil class Parameter (ISOIL):

For Rocky or Hard Soil sites i.e. ISOIL = 1

S_a/g	1+ 15 T	$0.00 \leq T \leq 0.10$
	2.50	$0.10 \leq T \leq 0.40$
	1.00/T	$0.40 \leq T \leq 4.00$

Modal Dynamic Analysis Data

Load Function Description Data

For Medium Soil sites i.e. ISOIL = 2

S _a /g	1+ 15 T	0.00 ≤ T ≤ 0.10
	2.50	0.10 ≤ T ≤ 0.55
	1.36/T	0.55 ≤ T ≤ 4.00

For Soft Soil sites i.e. ISOIL = 3

S _a /g	1+ 15 T	0.00 ≤ T ≤ 0.10
	2.50	0.10 ≤ T ≤ 0.67
	1.67/T	0.67 ≤ T ≤ 4.00

Multiplying Factors for Obtaining Values for Other Damping									
Damping%	0	2	5	7	10	15	20	25	30
Factors	3.2	1.40	1.00	0.90	0.80	0.70	0.60	0.55	0.50

Structure Importance Factor (SIFACT):

Important Structure = 1.5 (default value)

Other building = 1.0 (default value)

Response Reduction Factor (RRFACT):

Sl. No.	Lateral Load Resisting System	R
	Building Frame Systems	
i	Ordinary RC moment resisting frame (OMRF)	3.0
ii	Special RC moment resisting frame (SMRF)	5.0
iii	Steel frame with	
	a) Concentric braces	4.0
	b) Eccentric braces	5.0
iv	Steel moment resisting frame designed as per SP 6(6)	5.0
	Building With Shear Walls	
v	Load bearing masonry wall buildings	
	a) Un-reinforced	1.5
	b) Reinforced with horizontal RC bands	2.5
	c) Reinforced with horizontal RC bands & vertical bars at corners of rooms and jambs of openings	3.0
vi	Ordinary reinforced concrete shear walls	3.0
vii	Ductile Shear walls	4.0
	Building with dual systems	
viii	Ordinary Shear wall with OMRF	3.0
ix	Ordinary Shear wall with SMRF	4.0
x	Ductile Shear walls with OMRF	4.5
xi	Ductile Shear walls with SMRF	5.0

Factor for underground structures and foundations (USFACT):

USFACT is used for calculating the factor for underground structures and foundations. It is 1.0 at the surface and 0.5 for depths of 30 meters and below. For the structures and

foundations placed between the ground level and 30 m depth, the factor is obtained by interpolating between 1 and 0.5 for the given depth.

Earthquake resistant Design Code Year (IYEAR):

- IS Criteria for Earthquake Resistant Design of Structures (fifth Revision) = 2002
 - IS Criteria for Earthquake Resistant Design of Structures (fourth Revision) = 1984
5. IF LABEL = JAPAN, DIR and SFTR are the same parameters to define the direction of seismic event and the scale factor.

The ground acceleration response spectrum amplitude S_a is dependent on soil class (ISOIL) parameter, spectral shape parameters (F_A , F_V , and BND), return period (NRP), basic peak acceleration (ACCL) and basic peak velocity V_0 , of earthquake ground motion (see [Figure 8.3](#)).

The acceleration spectrum amplitude S_a is defined for a specified damping η and in terms of the natural period T (sec.) as,

$$S_a = \begin{cases} \left(1 + \frac{(FA-1) T}{BND T_C}\right) F_d G_A R_A * ACCL, & 0 < T < BND * T_C \\ F_d * FA * G_A R_A * ACCL, & BND * T_C < T < T_C \\ \frac{2\pi F_d * FV * G_V R_V * VEL}{T}, & T > T_C \end{cases}$$

ISOIL with 1 or 2 or 3 fixes the values of F_A , F_V , and BND.

F_A , F_V are soil type modification factors for the peak ground acceleration and velocity. These are dependent on the soil class parameter, ISOIL. The recommended values of F_A and F_V are given in [Table 8.10](#).

BND is upper bound period of the range where, S_a is constant (see [Figure 8.3](#)). The recommended values of BND for a given soil class (ISOIL) parameter is given in [Table 8.10](#)

R_A and R_V in the equation are the return period conversion factors respectively for peak acceleration and velocity of the earthquake ground motion and are related to the return period NRP by

$$R_A = R_V = \{NRP/100\}^{0.54}$$

The above relationship for R_A and R_V applies for a return period, NRP between 20 and 500 years.

F_d : damping modification factor = $1.5 / (1 + 10\eta)$, where, η is the damping ratio specified in *DAMPING data group.

ACCL and VEL are the base peak acceleration and velocity of the ground motion and are related by $ACCL = 15 * VEL$. These parameters pertain to a reference soil of firm ground and a return period of 100 years. User can specify ACCL only with IFLAG =0. Alternatively with IFLAG =1, user can specify earthquake magnitude ERMAG instead of ACCL and epicentral distance EPI. These two parameters first yield the basic peak velocity, VEL from the relationship.

$$VEL = 2[10^{[0.61(ERMAG) - (1.66 + 3.60/(EPI))\log(EPI) - (0.0631 + 1.83/(EPI))]}]$$

with the value of VEL, ACCL follows from $ACCL = 15 * VEL$.

Table 8.10: Recommended F_A , F_V and BND values for soil types

Soil Type	FA	FV	BND
1	1.0	1.0	0.33
2	1.2	2.0	0.56
3	1.2	3.0	0.84

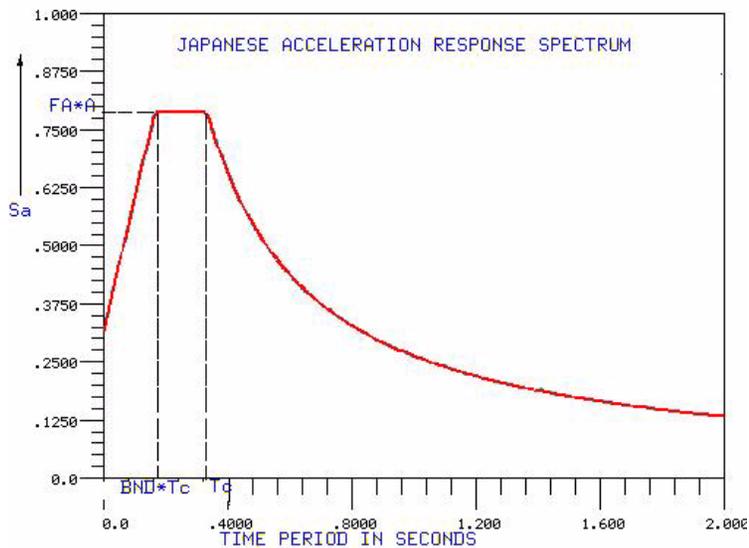


Figure 8.3: Acceleration response spectrum for LABEL = JAPAN

Table 8.11: List of valid labels for regulatory agencies

LABEL	Regulatory agencies	Response Spectrum
EURO8	European Committee for Standardization,1994	Acceleration response spectra
INDIA	Indian Standards Institution IS 1893-1984 & IS 1893-2002	Acceleration response spectra
JAPAN	Architectural Institute of Japan 1996	Acceleration response spectra
USNRC	US Nuclear regulatory commission 1973	Vertical & horizontal velocity spectra

8.7 Load Specification Data

8.7.1 Load Specification Parameters

The data groups included in this section are *GROUND, *DCFORCE, *DPRESSURE, and *DRIVER. They are common to all analysis types except *DRIVER data group, which is used only in frequency response analysis. The common data groups have two entries ID1 and ID2, which are interpreted differently for different analysis types as described below.

(a) TRANSIENT dynamic analysis

ID1 (IFN): Time function ID defined in *TIMEFUNCTION data group.

ID2 (IAT): Arrival time ID defined in *ARRIVALTIME data group. This entry can be specified as zero, if *ARRIVALTIME data group is not provided, indicating a zero arrival time.

(b) RANDOM vibration analysis

ID1 (NCR): Correlation ensemble ID defined in *CORRELATION data group. Enter zero if this input is not correlated with any other process.

ID2 (NPR): Process number for this input in the correlation ensemble if ID1 > 0.
PSD function ID defined in *PSDFUNCTION data group describing the auto PSD of this input if ID1 = 0.

(c) FREQUENCY response analysis

ID1 (IASP): Amplitude spectrum ID defined in *SPECTRUM data group.

ID2 (IPSP): Phase spectrum ID defined in *SPECTRUM data group.

(d) SHOCK spectrum analysis

ID1 (IMDS): Multiple damping spectrum group ID.

- > 0 multiple damping spectrum group ID defined in the *MDSPECTRUM data group.

- = 0 the spectrum is independent of damping and is defined using ID2.

- < 0 prestored spectrum values are used (see note 1).

ID2 (ISPE): Response spectrum ID defined in *SPECTRUM data group which describes the damping-independent spectrum if ID1 = 0.

Not used if ID1 ≠ 0.

The above information is summarized in [Table 8.12](#).

Note:

1. In shock spectrum analysis, a negative number may be specified for ID1 to use pre-stored groups of spectra. The prestored spectrum groups currently available are listed below.

ID1 = -1: Nuclear Regulatory Commission (NRC) horizontal velocity spectra (velocity in inch/sec. vs. frequency in Hz) defined for damping ratios between 0.005 and 0.10.

ID1 = -2: Nuclear Regulatory Commission (NRC) vertical velocity spectra (velocity in inch/sec. vs. frequency in Hz) defined for damping ratios between 0.005 and 0.10.

Table 8.12: Interpretation of load specification parameters ID1 and ID2

Analysis type	ID1	ID2	
		For ID1 > 0	For ID1 = 0
TRANSIENT dynamics	Time function ID number	Arrival time ID number (can be zero)	
RANDOM vibration	Correlation ensemble ID number (can be zero)	Process number in the correlation group	Auto PSD ID number
FREQUENCY response	Amplitude spectrum ID number	Phase spectrum ID number	
SHOCK spectrum	Multiple damping spectrum group ID number (can be zero or negative)	Not used	Damping independent spectrum ID number

8.7.2 *GROUND Data Group - Ground Motion Data

Applicable analysis types: TRANSIENT, RANDOM, FREQUENCY, SHOCK

This data group is always required for shock spectrum analysis and optional for other analysis types. Ground motion in all the six global directions and the global coordinates of the point about which there is rotational ground motion can be specified using this data group.

Group ID card:

*GROUND

Ground motion card set:

	\$						
Entry No:	1	2	3	4	5	6	7
Variable:	LABEL	ID1	ID2	SFTR/XC	YC	ZC	ZPA
Max char:	4	5	5	10	10	10	10

Entry Variable Description

- 1 LABEL label for ground motion direction. See [Table 8.13](#) for list of allowable labels.
- 2 ID1 analysis dependent parameter, see [Section 8.7.1](#) (Enter zero or blank if label is 'CORD'.)
- 3 ID2 analysis dependent parameter, see [Section 8.7.1](#). (Enter zero or blank if label is 'CORD'.)

--- tab(\$) --

- 4 SFTR scale factor used to multiply the function value if LABEL is not 'CORD' or
 XC global X coordinate of the point about which there is rotational ground motion if LABEL is 'CORD' (default value = 0.0)
- 5 YC not used if LABEL is not 'CORD'.
 Global Y coordinate of the point about which there is rotational ground motion if LABEL is 'CORD' (default value = 0.0).

- | | | |
|---|-----|--|
| 6 | ZC | not used if LABEL is not 'CORD'.
Global Z coordinate of the point about which there is rotational ground motion if LABEL is 'CORD' (default value = 0.0). |
| 7 | ZPA | Cutoff acceleration associated with the zero period acceleration in the corresponding direction as specified by LABEL |

Notes:

1. If rotational ground motion is not used, coordinates for the point of excitation need not be specified.
2. In frequency response analysis, any ground motion specified in this data group will be ignored if the *DRIVER group has been specified for transfer function computation.
3. The scale factor is used to scale the function values, i.e., time function ordinates in transient dynamic analysis, cross spectral density matrix in random vibration analysis, amplitude spectra in frequency response analysis and response spectra in shock spectrum analysis.
4. Additional cards may be provided if necessary to specify ground motion in other global directions. If ground motion in one direction is defined more than once, only the first definition is used.

Table 8.13: List of valid labels for ground motion data

UX UY UZ	Displacement in global X direction† Displacement in global Y direction† Displacement in global Z direction†
ROTX ROTY ROTZ	Rotation about global X axis† Rotation about global Y axis† Rotation about global Z axis†
VX VY VZ	Velocity in global X direction† Velocity in global Y direction† Velocity in global Z direction†
OMGX OMGY OMGZ	Rotational velocity about global X axis† Rotational velocity about global Y axis† Rotational velocity about global Z axis†
ACCX ACCY ACCZ	Acceleration in global X direction Acceleration in global Y direction Acceleration in global Z direction
ALPX ALPY ALPZ	Rotational acceleration about global X direction Rotational acceleration about global Y direction Rotational acceleration about global Z direction
CORD	Coordinates of the point about which there is rotational excitation

†Not available in TRANSIENT dynamic and RANDOM vibration analyses.

8.7.3 *DCFORCE Data Group - Concentrated Nodal Force Data

Applicable analysis types: TRANSIENT, RANDOM, FREQUENCY

This data group is optional and may be used to specify concentrated nodal forces.

Group ID card:

*DCFORCE

Concentrated nodal force card set:

	\$						
Entry No:	1	2	3	4	5	6	7
Variable:	NODE	LABEL	ID1	ID2	LAST	INCR	FORCE
Max char:	8	4	5	5	8	8	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NODE	node number.
2	LABEL	label for force or moment component. FX, FY, FZ - forces in the X, Y and Z directions, respectively. MX, MY, MZ - moments about the X, Y and Z axes, respectively.
3	ID1	analysis dependent parameter, see Section 8.7.1 .
4	ID2	analysis dependent parameter, see Section 8.7.1 .
5-6	LAST, INCR	nodes from 'NODE' through 'LAST' in increments of 'INCR' will be assigned the same value of FORCE and the same ID1 and ID2 (default values are NODE and zero, respectively).

--- tab (\$) ---

7	FORCE	multiplier for the load function.
---	-------	-----------------------------------

Notes:

1. If a local displacement coordinate system is defined at a particular node, then the forces applied at this node must be described in the local displacement coordinate system. Otherwise, all nodal loads are in the global coordinate system.
2. In frequency response analysis, all the loads specified in this data group will be ignored if *DRIVER group has been specified.
3. Additional cards may be provided, if necessary, to specify other concentrated nodal forces.

8.7.4 *DPRESSURE Data Group - Pressure Loading Data

Applicable analysis types: TRANSIENT, RANDOM, FREQUENCY

This data group is optional and is used to specify pressure loading on the elements. Refer to NISA element library for elements on which this loading may be applied.

Each pressure data input is defined by using two sets of cards. The first set consists of one card and is always required. The second set may have one or more cards and is required if the pressure is nonuniform.

Group ID card:

*DPRESSURE

Card set 1:

	\$									
Entry No:	1	2	3	4	5	6	7	8	9	10
Variable:	KELL	ID1	ID2	KFC	KY	LAST	INCR	PFU	CDLOW	CDUPP
Max char:	8	5	5	5	5	8	8	5	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	KELL	element ID number.
2	ID1	analysis dependent parameter, see Section 8.7.1 .
3	ID2	analysis dependent parameter, see Section 8.7.1 .
4	KFC	face number on which the pressure acts - see NISA element library for the face numbering convention.
5	KY	number of additional cards required to complete the pressure loading data.
6-7	LAST, INCR	automatic generation parameters. Elements from 'KELL' through 'LAST' in increments of 'INCR' will be assigned the same pressure load and the same ID1 and ID2 (default values are KELL and zero, respectively).

--- tab (\$) ---

8	PFU	uniform pressure load intensity - used only if KY is specified as zero.
9	CDLOW	variable not used in this version.

Modal Dynamic Analysis Data

Load Specification Data

10 CDUPP variable not used in this version.

Card set 2: Pressure load intensities

This card set is not required if a uniform pressure is to be specified ($KY = 0$). The uniform pressure load intensity can be specified as PFU if $KY = 0$.

Entry No:	1	2	3	4	5	6	7	8
Variable:	P1	P2	P3	P4	P5	P6	P7	P8
Max char:	10	10	10	10	10	10	10	10

Entry Variable Description

1-6 P1-P8 pressure load intensities at the nodal points of the element. See NISA element library for the node numbering convention on the face.

Notes:

1. Additional cards may be provided, if required, to specify other pressure loads.
2. Pressure acts normal to the face. A positive value indicates pressure acting towards the face.

8.7.5 *DRIVER Data Group - Driver Node Data

Applicable analysis types: FREQUENCY

This data group is optional and may be used to specify unit harmonic excitations (displacements, velocities, accelerations or forces) that may be applied to a specific node on the structure for transfer function calculations.

Group ID card:

*DRIVER

Driver node card set:

Entry No:	1	2	3
Variable:	NODE	LABEL	ID
Max char:	8	4	5

Entry Variable Description

- | | | |
|---|-------|---|
| 1 | NODE | node number. |
| 2 | LABEL | label for the transfer functions input (see the list of valid labels in Table 8.14). |
| 3 | ID | the phase spectrum identification number defined in *SPECTRUM group. |

Notes:

1. The phase spectrum ID referred to here should be defined in the *SPECTRUM group.
2. Additional cards may be provided, if needed, to specify other driver nodes and types of excitation. The NODE and the LABEL in each card correspond to one driver point. If more than one driver node is present, the program will excite the structure at each of the driver points one by one. All of the requested response output will be printed separately for each driver point.
3. Whenever this data group is present, all other specified loading types i.e., *GROUND, *DPRESSURE and *DCFORCE are ignored by the program, and only transfer function computations are carried out.
4. If a local displacement coordinate system is specified at a particular node, then the excitation (represented by LABEL) applied at this node must be described in the local displacement coordinate system. Otherwise, the excitation is in global coordinate system.

Table 8.14: List of valid labels for *DRIVER group

UX UY UZ	Displacement in X direction Displacement in Y direction Displacement in Z direction
ROTX ROTY ROTZ	Rotation about X axis Rotation about Y axis Rotation about Z axis
VX VY VZ	Velocity in X direction Velocity in Y direction Velocity in Z direction
OMGX OMGY OMGZ	Rotational velocity about X axis Rotational velocity about Y axis Rotational velocity about Z axis
ACCX ACCY ACCZ	Acceleration in X direction Acceleration in Y direction Acceleration in Z direction
ALPX ALPY ALPZ	Rotational acceleration about X direction Rotational acceleration about Y direction Rotational acceleration about Z direction
FX FY FZ	Force in X direction Force in Y direction Force in Z direction
MX MY MZ	Moment about X direction Moment about Y direction Moment about Z direction

Modal Dynamic Analysis Data

Load Specification Data

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IDPATH	moving load path identification number defined in *MVEPATH data group
2	LABEL	label for moving force or moment component. FX, FY, FZ - forces in X, Y and Z axes, respectively. MX, MY, MZ - moments about X, Y and Z axes respectively.
3	NF	number of forces moving along the path defined in IDPATH
4	NTABLE	table form to be used in input load in card set 3. = 0 position of the load in terms of the distance from the first load = 1 position of the load in terms of its arrival time
5	VEL	velocity of the moving load
6	ARRT	arrival time value (Used for NTABLE = 0 only) This corresponds to arrival time of the first load)
7	ACCEL	acceleration of the moving load
8	IDTMF	time function id. The additional moving forcing function to be specified in *TIMEFUNCTION card.

Card set 3: Moving load definition cards

For NTABLE = 0

The magnitude of the load and position of the load in terms of the distance from the first load should be entered in pairs up to maximum of three pairs per card. The position of the load should be entered in ascending order and must be positive. The first position is zero. Enter only NF pairs.

Entry No:	1	2	3	4	5	6
Variable:	P1	F1	P2	F2	P3	F3
Max char:	10	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	P1	0
2	F1	magnitude of first load
3	P2	distance between the first and second load

- 4 F2 magnitude of second load
- 5 P3 distance between the first and third load
- 6 F3 magnitude of third load

For NTABLE = 1

The magnitude of the load and position of the load in terms of the arrival time of each load should be entered in pairs up to maximum of three pairs per card. The arrival time of the load should be entered in ascending order and must be positive except at the first position where the position value can be zero. Enter only NF pairs.

Entry No:	1	2	3	4	5	6
Variable:	T1	F1	T2	F2	T3	F3
Max char:	10	10	10	10	10	10

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	T1	arrival time of the first load
2	F1	magnitude of first load
3	T2	arrival time of the second load
4	F2	magnitude of second load
5	T3	arrival time of the third load
6	F3	magnitude of third load

Note:

1. Any number of moving loads can be defined by repeating sets 1 to 3.

8.7.7 *MVEPATH Data Group - Moving Load Path Definition

Applicable to: TRANSIENT

Set of integer numbers (nodes IDs) defined in this group for subsequent reference in the *MVELOAD data group, to specify the path of moving load.

Group ID card:

*MVEPATH

Moving load path card set:

Entry No:	1	2	3	4	5	6	7	8	9	10	11
Variable:	IDPATH	LABEL	I1	J1	K1	I2	J2	K2	I3	J3	K3
Max char:	6	1	8	8	8	8	8	8	8	8	8

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IDPATH	path identification number, integer > 0.
2	LABEL	card identification label (to interpret the remaining entries on this card). Allowable labels are: S = entries on this card are single numbers R = entries on this card define range of numbers E = entries on this card are single numbers to be excluded from any ranges of numbers defined in this path
3-11	I1, J1, K1...K3	interpretation of these entries depend on the identification LABEL such that I1, J1, K1,...K3 are single node numbers when LABEL is 'S'. (I1, J1, K1), (I2, J2, K2), (I3, J3, K3) define three ranges of numbers beginning with I _i , ending with J _i and in increment of K _i , for i = 1, 2, 3, respectively, when LABEL is set to 'R'. I1, J1, K1, ... K3 are single numbers to be excluded from any ranges of numbers defined in this path, when LABEL is set to 'E'.

Notes:

1. The path of the moving load should follow along the element edges. Whatever nodes are present along the path should be specified in this card. Not a single node should be left out even if some boundary condition is specified on the node.
2. LABEL must not be blank for the first card used to define the path.
3. If LABEL is left blank for any subsequent card (defining the same path), the LABEL of the previous card is used.
4. Define as many paths as desired, each path must start with a unique non-zero 'IDPATH'.
5. If any path requires more than one card, continue with additional cards. Additional cards must start with a tab character \$. If any additional card starts with two consecutive tab characters \$\$, then the label of the previous card is used.
6. Once a path is defined, a non-zero value of 'IDPATH', on any subsequent card will be the definition of a new path, if any.

8.7.8 *MSEXCITATION data group – Multiple support excitation data group

Applicable analysis types: SHOCK, FREQUENCY, TRANSIENT, RANDOM

This data group is always required for multi support excitation analysis. Ground motion in all the six global directions can be specified at nodes using this data group

Group ID card: *MSEXCITATION

Ms excitation card set:

Entry No:	1	2	3	4	5	6	7	7	8
Variable	NODE	LABEL	ID1	ID2	ID3	ZFD	ZPA	LASTND	NODINC
Max char	8	4	6	6	6	12	12	8	8

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	NODE	Node Number
2	LABEL	label for ground motion direction see Table 8.16 for list of allowable labels
3	ID1	analysis dependent parameter, see Table 8.15
4	ID2	analysis dependent parameter, see Table 8.15
5	ID3	analysis dependent parameter, see Table 8.15
--- tab(\$) ---		
6	ZFD	Zero frequency displacement (used only for shock spectrum analysis)
7	ZPA	Zero period acceleration in the direction as specified in LABEL, used for missing mass correction.
8	LASTND	Last node of a range of nodes with same specified spectra
9	NODINC	Positive Increment for the range of nodes

Notes:

1. ZFD is the Zero Frequency Displacement vector given by user to compute quasi-static responses. If not given by the user, it is taken to be the displacements corresponding to the lowest frequency of each of the input spectra.
2. ZPA is the Zero Period Acceleration vector given by user in case of missing mass correction. If ZPA is not given, the cut-off frequency using the CFREQ data in EXECUTIVE command is used to find ZPA. In case both are not given, missing mass correction is not computed.
3. This card is to be used only for differential support motions (non-uniform multi support excitation only). This card can also be used in the case of some of the supports being at the same level. In such a case user shall define spectra at the same level with identical ID (This is applicable for both ID1 and ID2).

Table 8.15: Interpretation of load specification parameters ID1, ID2 and ID3

Analysis Type	ID1	ID2 (for ID1=0)	ID3
SHOCK Spectrum Analysis	Multiple damping spectrum group ID number (can be zero or negative)	Damping independent spectrum ID number (Not used for ID1 > 0)	Not Used
Transient	Time function ID defined in *TIMEFUNCTION data group.	Arrival time ID defined in *ARRIVALTIME data group. This entry can be specified as zero, if *ARRIVALTIME data group is not provided, indicating a zero arrival time.	- Displacement time history ID defined in *TIMEFUNCTION data group - '0' to integrate time history specified in ID1
FREQUENCY response analysis	Amplitude spectrum ID defined in *SPECTRUM data group.	Phase spectrum ID number defined in *SPECTRUM data group.	Not Used
Random Vibration Analysis	Correlation ensemble ID defined in *CORRELATION data group. Enter zero if this input is not correlated with any other process.	- Process number for this input in the correlation ensemble if ID1 > 0. - PSD function ID defined in *PSDFUNCTION data group describing the auto PSD of this input if ID1 = 0.	Not Used

Table 8.16: List of valid labels for MSEXCITATION on data

UX UY UZ	Displacement in global X direction [†] Displacement in global Y direction [†] Displacement in global Z direction [†]
ROTX ROTY ROTZ	Rotation about global X axis [†] Rotation about global Y axis [†] Rotation about global Z axis [†]
VX VY VZ	Velocity in global X direction [†] Velocity in global Y direction [†] Velocity in global Z direction [†]
OMGX OMGY OMGZ	Rotational velocity about global X axis [†] Rotational velocity about global Y axis [†] Rotational velocity about global Z axis [†]
ACCX ACCY ACCZ	Acceleration in global X direction Acceleration in global Y direction Acceleration in global Z direction
ALPX ALPY ALPZ	Rotational acceleration about global X direction Rotational acceleration about global Y direction Rotational acceleration about global Z direction

[†]Not available in TRANSIENT dynamic and RANDOM vibration analyses.

8.8 Output Control Data

8.8.1 *RSET Data Group - Response Set Data

Applicable analysis types: TRANSIENT, RANDOM, FREQUENCY

This data group is always required. It defines the nodal/element response quantities that are to be output or stored in file for postprocessing. These responses are grouped together in sets, each set having a unique ID.

Group ID card:

*RSET

Response card set:

	\$					
Entry No:	1	2	3	4	5	6
Variable:	ID	NDEL	LABEL	LAST	INCR	LAYR
Max char:	5	8	4	8	8	5

Entry Variable Description

- | | | |
|-------------|---------------|---|
| 1 | ID | response set identification number to be referenced in other data groups in this section. |
| 2 | NDEL | node number/element number. |
| 3 | LABEL | label for the nodal or element response quantity required, see the list of valid labels in Table 8.17 . |
| 4-5 | LAST,
INCR | nodes or elements from 'NDEL' through 'LAST' in increments of INCR will be assigned the same label and set ID (default values are NDEL and zero, respectively). |
| --tab(\$)-- | | |
| 6 | LAYR | layer number if 'LABEL' indicates layer stress for composite elements. Not used for other labels. |

Notes:

1. Only the nodal or element responses specified in this group will be computed.
2. Additional cards may be used, if necessary, to specify responses to be included in the same or different response set.
3. Beam stresses can be computed in DISPLAY-POST for Transient Dynamics, Frequency Response, and Shock Spectrum analysis provided all components of beam end forces are computed during modal dynamic analysis.

Table 8.17: List of valid labels for response sets

UX UY UZ ROTX ROTY ROTZ	Nodal displacement components
DISP	All nodal displacement components
VX VY VZ OMGX OMGY OMGZ	Nodal velocity components
VELO	All nodal velocity components
ACCX ACCY ACCZ ALPX ALPY ALPZ	Nodal Acceleration components
ACCL	All nodal acceleration components
SXX SXY SZX SYY SYZ SZZ	Nodal stress components for continuum elements or mid-surface stress components for shell elements

STRS	All nodal stress components
SXXT SXYT SZXT SYYT SYZT SZZT	Nodal stress components at the top surface of shell elements
STRT	All nodal top surface stress components
SXXB SXYB SZXB SYYB SYZB SZZB	Nodal stress components at the bottom surface of shell elements
STRB	All nodal bottom surface stress components
NXX NYY NXY MXX MYX MXY QX QY	Nodal resultant stress components for composite shell elements
SRLT	All nodal resultant stress components
SXXL SXYL SYYL SZXL SYZL	Nodal layer stress components for composite shell elements
STRL	All nodal layer stress components
FX FY FZ MX MY MZ	Nodal reaction components
RCTN	All nodal reaction components

Modal Dynamic Analysis Data

Output Control Data

BFX BSY BSZ BMX BMY BMZ	Beam element end forces
BFOR	All beam element end forces (see Note 3)
NXXC NYYC NXYC MXXC MYYC MXYC QXC QYC	Centroidal stress resultant of shell elements
SRTC	All Centroidal Stress resultants (see Note 3)
CXX CXY CZX CYY CYZ CZZ	Centroidal stress
CSRS	All Centroidal stress (see Note 3)
CXXB CXYB CZXB CYYB CYZB CZZB	Centroidal stress at Bottom surface
CSRB	All centroidal stress at Bottom surface
CXXT CXYT CZXT CYYT CYZT CZZT	Centroidal stress at Top surface
CSRT	All centroidal stress at Top surface

CXXL CXYL CZXL CYYL CYZL CZZL	Centroidal layer stresses
CSRL	All Centroidal layer stress components
BASF	Base Shear -FX, FY, FZ (see Note 3)
ELFO	Elemental Internal Forces
VMS	Nodal von Mises stress

8.8.2 *HISTORY Data Group - Response Time History Request Data

Applicable analysis types: TRANSIENT.

This data group is optional and can be used to request response history computation for printout and/or postprocessing.

Group ID card:

*HISTORY

History card set:

	\$						
Entry No:	1	2	3	4	5	6-13	14
Variable:	TIME1	TIME2	NOPI	IOPT	ID1	...	ID10
Max char:	10	10	5	5	5	...	5

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	TIME1	starting time for history output (default value is the starting time for analysis specified in executive commands).
2	TIME2	ending time for history output (default value is the ending time for analysis specified in executive commands).
3	NOPI	history calculation increment. The response history will be calculated at intervals of NOPI time steps (default value = 1).
4	IOPT	printout option = 0 - response history is calculated but not printed. = 1 - time history, maximum values and time of occurrence of maximum values are printed. = 2 - only maximum values and times of occurrence of maximum values are printed.
-- tab(\$) --		
5-14	ID1 to ID10	the response set identification numbers for which the time history response is required. A maximum of ten IDs can be specified per card.

Notes:

1. The response set IDs referred to here should be defined in the *RSET data group.
2. If a response is included in more than one response set ID, the evaluation of the response will be repeated.
3. Additional cards may be provided, if needed, to request other responses in the same or different time ranges.

8.8.3 *SNAPSHOT Data Group - Snap Shot

Applicable analysis types: TRANSIENT

This data group is optional and may be used to request a computation of the snapshot (freeze) response, i.e., responses at a specified time, for printout and/or postprocessing.

Group ID card:

*SNAPSHOT

Snapshot card set:

	\$					
Entry No:	1	2	3	4	5-14	15
Variable:	TIME	IDPEAK	IOPT	ID1	...	ID12
Max char:	10	5	5	5	...	5

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	TIME	time at which the snapshot response is required (see note 1).
2	IDPEAK	response set ID whose time of maxima is used as the time at which the snapshot is required (see note 1).
3	IOPT	printout option = 0 - responses are calculated but not printed. = 1 - responses are calculated and printed = 2- responses are calculated and printed. In addition, displacement and velocity responses are written on file 18 (see note 7).
--tab(\$) --		
4-15	ID1 to ID12	response set identification numbers for which the snapshot response is required. A maximum of twelve IDs can be specified per card.

Notes:

1. Either TIME or IDPEAK can be specified to determine the time at which a snapshot response is required. If IDPEAK is non-zero, then TIME is ignored.
2. The response set IDs referred to here should be defined in the *RSET data group.
3. Usually IDPEAK has only one nodal response in *RSET data group whose time of maxima is used to calculate the responses.
4. ID1 through ID12 may typically cover all the nodes of the structure in terms of displacements, velocities, accelerations, stresses, etc. to get the total picture at the specified time step. This is required if contour plotting is needed during postprocessing.
5. If a response is included in more than one response set ID, the evaluation of the response will be repeated.
6. Additional cards may be provided, if needed, to request responses at the same or different time steps.
7. If IOPT = 2, nodal displacements and/or velocities implied by ID1 through ID12 are written into file 18 in the initial condition data format so that it can be used as part of the input to restart the analysis at this time. This option cannot be used if IDPEAK is nonzero. File 18, if needed, has to be saved using SAVEFILE command in executive commands.

8.8.4 *PSDOUT Data Group - Response PSD Request Data

Applicable analysis types: RANDOM

This data group is optional and can be used to request response PSD computation for printout and/or postprocessing

Group ID card:

*PSDOUT

Response PSD card set:

	\$					
Entry No:	1	2	3	4	5-14	15
Variable:	IDNO	NOPI	IOPT	ID1	...	ID12
Max char:	5	5	5	5	...	5

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IDNO	output identification number to be referenced during postprocessing.
2	NOPI	printout increment. The response PSD will be output at every NOPI of the frequency points employed for integration (default value = 1). The PSD computation is done however at all the frequency points.
3	IOPT	PSD printout option = 0 - response auto PSDs are calculated but not printed. = 1 - response auto PSDs and statistical properties are printed (See note 1). = 2 - response auto PSDs are printed (see note 2). = 3 - response auto PSDs' maxima and statistical properties are printed. = 4 - PSD matrices (Auto & Cross PSD) are computed and printed in file 18 (see note 3) for all RSET IDs referred in this card.
--tab(\$)--		
4-15	ID1 to ID12	the response set identification numbers for which the PSDs are required. A maximum of twelve IDs can be specified per card.

Notes:

1. Statistical properties obtained from PSD functions are
 - (a) Root Mean square value (RMS): It is equal to standard deviation for zero processes like displacements, velocities, accelerations and stress components.
 - (b) Zero Mean Crossing Rate (MCR): The mean crossing rate is the expected value of the number of zero crossings with positive slope per unit time (apparent frequency).
 - (c) Extreme value of the response quantity (EXTDP) with probability equal to DPROB in the time interval TSEISMIC.
 - (d) Shape Factor (SHAPE): It is a measure of the bandwidth of a random process. It is defined as

$$k = \sqrt{2\pi(1 - \lambda_1^2 / (\lambda_0\lambda_2))}$$

where, λ_1 is the spectral moment given by

$$\lambda_j = \int_0^{\infty} \omega^j S_X(\omega) d\omega$$

- (e) Expected value of the number of crossings (LVLCROSS) of a level defined by EXTPD with a positive slope in the time interval TSEISMIC.
2. Maxima of the PSDs and their frequency at which maxima occur are also printed.
 3. Cross PSDs are calculated within the groups of same kind of response quantities and provided in file 18.

For example, if RSET and PSDOUT cards read as follows:

```
*RSET
1001, 1,UX,4,1
1002,1,UY,5,1
1003,1,UZ,3,1
*PSDOUT
2001,1,4,1001,1002,1003
```

PSD matrix of size 12 x 12 between UX, UY and UZ is computed and printed in File 18.

It is to be noted that cross PSD functions for stresses are not made available in file 18.

4. The response set IDs referred to here should be defined in the *RSET data.
5. If a response is included in more than one response set ID, the evaluation of the response will be repeated. Additional cards may be provided, if needed, to request PSD output for other responses.

8.8.5 *RMSOUT Data Group - Root Mean Square of Response Request Data

Applicable analysis types: RANDOM

This data group is optional and can be used to request computation of RMS values (standard deviations) of responses for printing and/or postprocessing.

Group ID card:

*RMSOUT

RMS response card set:

Entry No:	1	2 \$ 3	4-13	14	
Variable:	IDNO	IOPT	ID1	...	ID12
Max char:	5	5	5	...	5

Entry Variable Description

- | | | |
|---|------|--|
| 1 | IDNO | output identification number to be referenced during postprocessing. |
| 2 | | printout option. |
| | IOPT | = 0 - RMS responses are calculated but not printed.
= 1 - RMS responses are calculated and printed. |

--tab(\$)--

- | | | |
|------|----------------|---|
| 3-14 | ID1 to
ID12 | the response set identification numbers for which the RMS values are required. A maximum of twelve IDs can be specified per card. |
|------|----------------|---|

Notes:

1. The response set IDs referred to here should be defined in the *RSET data.
2. If a response is included in more than one response set, the evaluation of the response will be repeated.
3. To get a contour plot of RMS responses during post-processing, all the nodes should be included in the set for the corresponding response quantity.
4. Additional cards may be provided, if needed, to request computation of other RMS responses.

8.8.6 *SPOUT Data Group - Output Spectra Request Data

Applicable analysis types: FREQUENCY

This data group is always required and is used to request a computation of output responses at all the exciting frequencies for printout and/or postprocessing.

Group ID card:

*SPOUT

Output spectra request card set:

	\$					
Entry No:	1	2	3	4	5 -12	13
Variable:	IDNO	OMGT	IOPT	ID1	...	ID10
Max char:	5	10	5	5	...	5

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IDNO	response output identification number to be referenced during postprocessing.
2	OMGT	phase shift angle Ωt (see note 2).
3	IOPT	printout option (see note 2). = 0 output spectra are calculated but only the response maxima and the frequencies at which they occur are printed. = 1 output spectra, response maxima and the frequencies at which they occur are printed. = 2 the response is printed at the user specified phase shift OMGT.
--tab(\$)--		
4-13	ID1to ID10	the response set identification numbers for which the output is required. A maximum of ten IDs can be specified.

Notes:

1. The response set IDs referred to here should be defined in the *RSET data.
2. For IOPT = 0 and IOPT = 1, the responses are computed and/or printed in amplitude-phase format and OMGT is ignored.

For IOPT =2, the responses are stored in amplitude-phase format but printed in real imaginary format for the given value of OMGT as follows. Any response can be given as

$$a(t) = a_o e^{i(\Omega t + \beta)}$$

where, a_o is amplitude and β is the phase. For IOPT = 2, a_o and β are stored in the file for postprocessing but the real and imaginary components of $a(t)$ for the given value of Ωt (OMGT) are printed.

3. If a response is included in more than one response set, the evaluation of the response will be repeated.
4. Additional cards may be provided, if required, to request responses with the same or different values of OMGT and IOPT.

8.8.7 *RESPONSE Data Group - Maximum Response Request Data

Applicable analysis types: SHOCK

This data group is always required and is used to request a computation of estimates of maximum responses using any of the modal combination methods for printout and/or postprocessing.

Group ID card:

*RESPONSE

Response request card set:

	\$			
Entry No:	1	2	3	4
Variable:	METHOD	IOPT	LABEL	LAYR
Max char:	5	10	5	5

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	METHOD	label for modal combination method to calculate response quantities. Allowable labels are: ABS or PEAK - absolute sum of the modal responses. SRSS or RMS - square root of the sum of the squares of the modal responses. NRL or PRMS - the absolute maximum of the modal responses added to the RMS of the remaining modal responses. CQC - complete quadratic combination. GRP - grouping method TPM - ten percent method DSM - double sum method
2	IOPT	printout option = 0 - responses are calculated but not printed. = 1 - responses are calculated and printed.
--tab(\$)--		
3	LABEL	see Section 8.7.1 for valid labels.

- 4 LAYR layer number if LABEL indicates layer stresses for composite elements, otherwise enter zero or blank.

Notes:

1. NISA allows one or more of the modal combination methods to be used for combining modal response maxima. The method for combination of maxima across the directions is chosen, however, by using the DIRECTION card in the executive commands.
2. The CQC combination method employs modal damping values specified in the *DAMPING group. For zero damping in all modes, the CQC method reduces to the SRSS/RMS approach.
3. All element/nodal responses indicated by 'LABEL' are calculated and/or printed.
4. Additional cards may be provided, if needed, to request other responses with the same or different modal combination method.
5. NISA/SHOCK analysis can compute estimate of maximum base shear force (refer [Section 8.7.1](#) for valid labels)

8.8.8 *FRSOUT Data Group – Floor Response Spectra Output Request Data Group

Applicable analysis type: TRANSIENT

This data group is optional and can be used to request floor response spectra computation for printout and/or post processing.

Floor Response Spectra Card Set:

Group ID card:

FRSOUT

Entry No:	1	2	3	4	\$	5	6	7
Variable:	IDNO	NTABLE	IOPT	LABEL	IDMDS	ISPEC	MSEXT	
Max Char:	5	5	5	4	5	5	2	

<u>Entry</u>	<u>Variable</u>	<u>Description</u>
1	IDNO	identification number to be referenced during post-processing
2	NTABLE	table form to be used to output the floor response spectrum. = 0 – spectrum is a function of frequency (cycles/time) = 1 – spectrum is a function of frequency (radians/time) = 2 – spectrum is a function of period
3	IOPT	Printout option = 0 – response FRSs are calculated but not printed = 1 – response FRSs are calculated and printed
4	LABEL	AVER – Average response spectrum MAXI – Maximum response spectrum AP1S - (Average + One Sigma) response spectrum
--- tab(\$) ----		
5	IDMDS	multiple damping spectrum identification number. This number is referenced in the *GROUND data (Default IDMDS = 2001) (See Note 3)
6	ISPEC	spectrum identification defined in *SPECTRUM data. The spectrum curve with the ID number ISPEC is included in the multiple damping spectrum. (Default ISPEC= 1001) (see note 3)
7	MSEXT	Output option (FILE 18)

- = 0 - FRS output in *GROUND format
- = 1 - FRS output in *MSEXCITATION format

Notes:

1. All responses will be in the global coordinate system.
2. Additional cards may be provided, if needed, to request FRS output for other responses.
3. The FRS outputs can be utilized as inputs to shock response analysis of NISA/DA, if so required by the user. To meet this requirement, an additional input NISA file is generated with FRS outputs included in *SPECTRUM card. For more than one damping value, multiple damping card *MDSPECTRUM is also included in the input NISA file.
4. If more than one response is requested, then the Ids IDMDS and ISPEC are incremented by one.

Example:

The average response acceleration spectra in X, Y and Z directions generated at a given node is printed as per NISA input format. For example if the user requests the FLOOR RESPONSE SPECTRA in X, Y and Z directions at node 100, for 3 damping values 0.01, 0.02, 0.05 with the spectrum ID as ISPEC 1001, and with multiple spectra ID as IDMDS=2001, then the following spectra are written in the following NISA format.

```
*SPECTRUM
** AVERAGE RESPONSE SPECTRA IN X,Y AND Z DIRECTIONS AT NODE = 100
AVERAGE RESPONSE SPECTRA FOR ACCX COMPONENT – DAMPING 0.01
1001,10,0,1.0
...
...
...
AVERAGE RESPONSE SPECTRA FOR ACCX COMPONENT – DAMPING = 0.02
1002,10,0,1.0
...
...
...
AVERAGE RESPONSE SPECTRA FOR ACCX COMPONENT – DAMPING = 0.05
1003,10,0,1.0
...
...
...
AVERAGE RESPONSE SPECTRA FOR ACCY COMPONENT – DAMPING 0.01
1004,10,0,1.0
...
...
...
AVERAGE RESPONSE SPECTRA FOR ACCY COMPONENT – DAMPING = 0.02
1005,10,0,1.0
...
...
...
AVERAGE RESPONSE SPECTRA FOR ACCY COMPONENT – DAMPING = 0.05
1006,10,0,1.0
...
...
```

...

AVERAGE RESPONSE SPECTRA FOR ACCZ COMPONENT – DAMPING 0.01
1007,10,0,1.0

...

...

...

AVERAGE RESPONSE SPECTRA FOR ACCZ COMPONENT – DAMPING = 0.02
1008,10,0,1.0

...

...

...

AVERAGE RESPONSE SPECTRA FOR ACCZ COMPONENT – DAMPING = 0.05
1009,10,0,1.0

...

...

...

*MDSPECTRUM

**ACCELERATION SEPCTRA ARE COUPLED TOGHETER

**UNDER EACH MULTIPLE DAMPING SPECTRUM ID NOs. 2001,2002, 2003

2001,1001,0.01

2001,1002,0.02

2001,1003,0.05

2002,1004,0.01

2002,1005,0.02

2002,1006,0.05

2003,1007,0.01

2003,1008,0.02

2003,1009,0.05

*GROUND

ACCX,2001,0,1.0

ACCY,2002,0,1.0

ACCZ,2003,0,1.0

8.9 Data Deck Terminator

8.9.1 *ENDDATA Data Group - Input Data Deck Terminator

Applicable analysis types: TRANSIENT, RANDOM, FREQUENCY, SHOCK

This data group is the same as in [Section 7.6.1](#) but is repeated here for completeness. The group ID must be the last entry in the data deck. Any data following this group ID card will be ignored.

Group ID card:

*ENDDATA

A

Modeling Hints

Finite element modeling is an art that may not be governed by strict rules. Hence, the guidelines given herein are not rigid rules, since for some of them there may be an exception.

It is important to note that the results that one obtains using the finite element method are approximate to the physical system which is represented by a discrete mathematical model. The physical system has an infinite number of degrees of freedom, whereas the discrete model (the finite element model) has finite number of degrees of freedoms. The degree of the approximation in the calculated results depends on how closely the physical system is modeled. The modeling process involves many possibilities: The element types and shapes, the number of elements and the mesh grading, the material and geometric properties and the applied loads and boundary conditions. Each of these factors may cause the misrepresentation of the actual physical system.

The following items are general guidelines for modeling:

- NISA includes a large number of different structural and heat transfer elements which are developed for specific applications.

The selection of elements for an application should be based on its capabilities, its cost (stiffness matrix generation, decomposition and stress calculation, etc.) and the desired accuracy in the results. For example, the discrete Kirchhoff thin shell element (NKTP = 40) should not be used to model thick shells for which the transverse shear deformation influence may not be negligible. [Section 4.1.1](#) includes general guidelines for the element selection.

- ❑ A major cause of ill-conditioning of the system equations (Ref. A.1), which may affect the accuracy of the results significantly, is a region of high stiffness connected to a region of low stiffness (i.e., stiff region connected to a flexible region). Make use of coupling, multipoint constraints and rigid elements to model stiff or rigid regions.
- ❑ Gap elements (NKTP = 42, 43) should not be applied to edges or faces of elements with midside nodes. Use the first order elements in regions connected to gap elements.
- ❑ Elements should be of regular shapes as much as possible. This can be achieved when the element aspect ratio is close to unity. The best shape of a quadrilateral is a square, and that of hexahedron is a cube. The distortion index, calculated by the pre-processing program DISPLAY, is the indicator of how well the element maps to the ideal or best shape of element. The most desirable distortion index is 1.0, which indicates that the element maps perfectly to the ideal shape. Values of 0.6 to 1.0 are acceptable; elements with distortion index less than 0.4 in the high stress gradient areas should be avoided.

The best shape for the triangular element is an equilateral triangle. The included angles for triangular elements should be kept more than 45°.

Included angles for elements of less than 45° between the sides of the quadrilateral or the faces of the hexahedron should be avoided, specially in the area of interest. The included angles can be calculated by the pre-processing program DISPLAY.

A warping angle for linear quadrilateral shell elements should be kept less than 10 degrees. It is an indicator of the deviation of the element from its mean plane and is calculated as shown in [Figure A.1](#). Highly warped element in less important areas can be divided into two triangular elements if remodeling is not possible. Linear triangular elements cannot warp, because three nodes define a plane. The pre-processing program DISPLAY can be used to check warping angle for the elements and split them into two triangular elements if necessary.

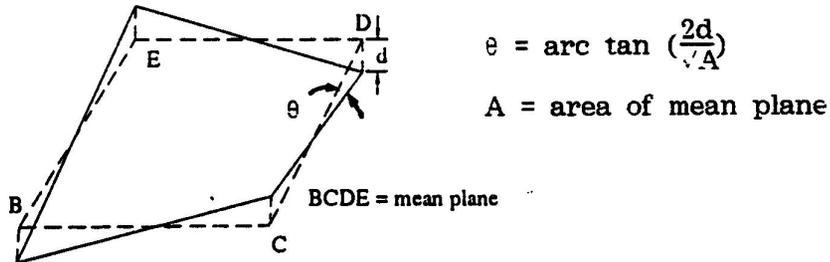


Figure A.1: Warping angle calculation

Very high aspect ratio (length/width) should be avoided. The aspect ratio of 7 or less for the displacement solution and 3 or less for the stress solution should generally produce acceptable results.

Examples of acceptable and unacceptable shapes are shown in [Figure A.2](#).

Higher order elements (parabolic, etc.) are less sensitive to element distortion and aspect ratio than the first order elements (linear).

Avoid the usage of triangles and wedges in high stress gradient areas.

- Among the entire family of isoparametric elements, the parabolic elements perform the best and are cost-effective for all types of analyses. It is our recommendation that parabolic elements be used as far as possible. Quadrilaterals and hexahedrons are more cost effective than triangles and wedge elements.

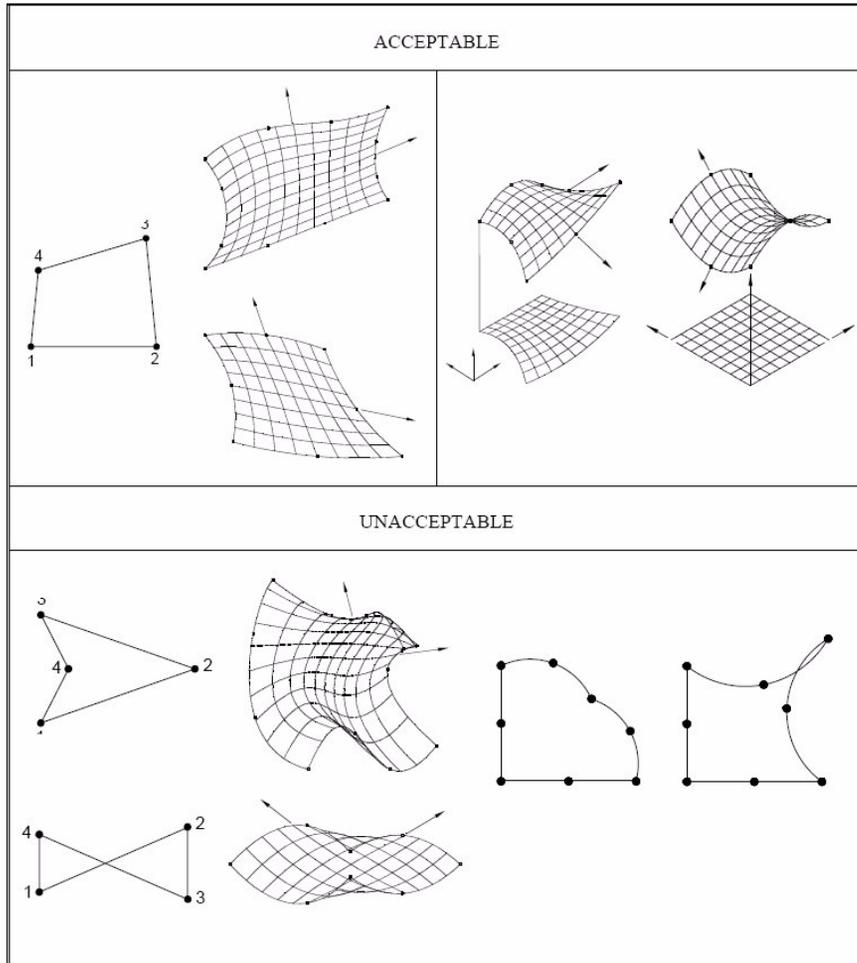


Figure A.2: Examples of Acceptable and Unacceptable Distortion in 2-D and 3-D Isoparametric Elements

- ❑ For parabolic elements the midside node should be placed as close to the midpoint of the side as possible. Similarly, for cubic elements, the side nodes should be placed as the one-third and two-third point along the sides.
- ❑ Care should be taken when using isoparametric elements of different orders in the same model. For example, a parabolic element should not be connected to a linear element without special treatment. This is due to the fact that, along the common

edge, the displacement varies parabolically for the parabolic element and linearly for the linear element. This produces incompatibility. A remedy of this situation would be to use a transition element to match the displacement variation along the common edge, or to restrict the displacement variation along the common edge to be linear using multipoint constraint equations.

The midside node of element should not be used as the corner node of another element. This happens during the transition from the fine mesh to the coarse mesh. The recommended procedures to achieve these are shown in [Figure A.3](#).

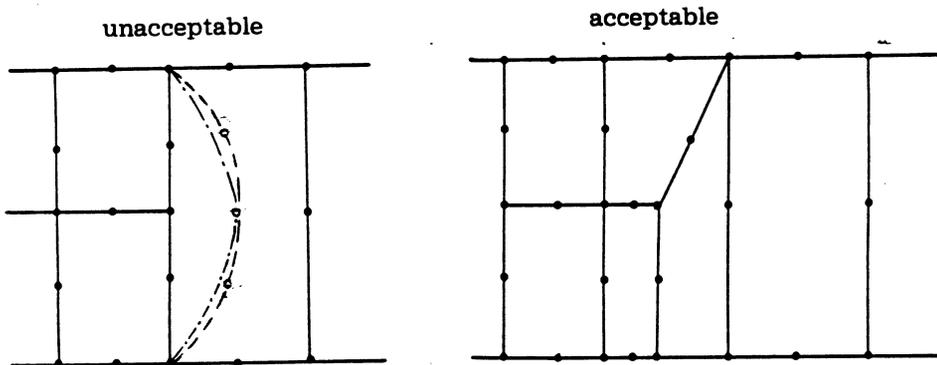


Figure A.3: Examples of Acceptable and Unacceptable Techniques for Mesh Refinements using Isoparametric Finite Elements

A triangular region and circular region can be modeled as shown in [Figure A.4](#) and [Figure A.5](#), respectively, with the quadrilateral or hexahedron elements.

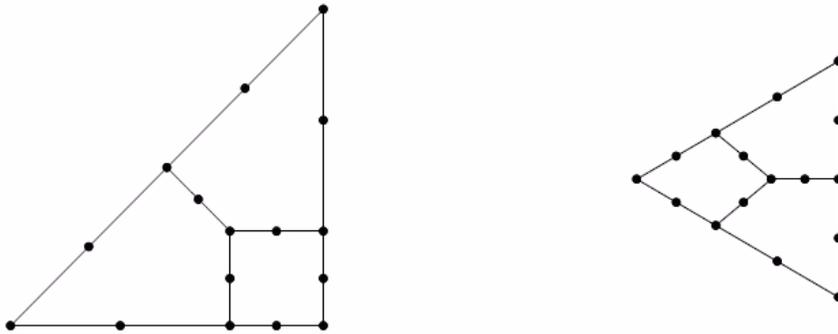


Figure A.4: Alternative Element Patterns for Triangular Regions

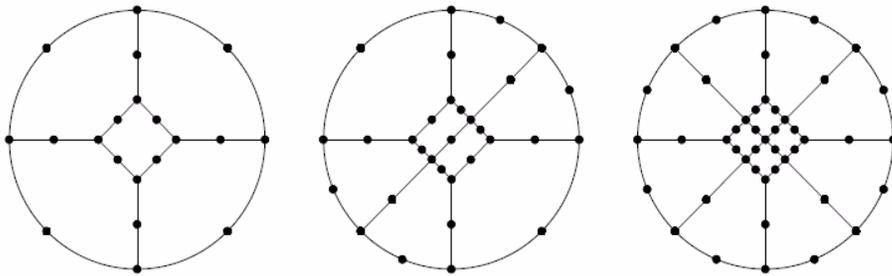


Figure A.5: Alternative Element Patterns for Circular Regions

- Finer mesh should be used in regions of particular interest, e.g., high stress gradient areas. Use of triangles and wedges in these areas is not recommended. In these areas if expected stress gradients are high, then a 90° arc should be divided into 3 elements if higher order elements are used. Mesh grading should be done in such a way that abrupt changes in element size are minimized. Several schemes of mesh grading are shown in [Figure A.6](#).

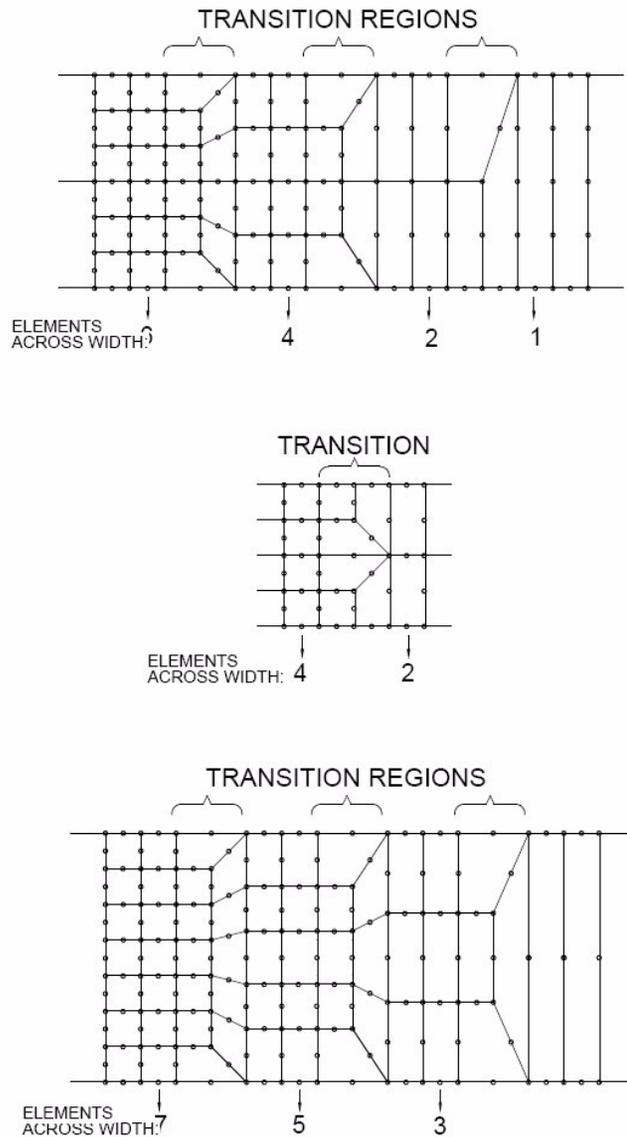


Figure A.6: Examples of Mesh Refinements and Grading for Isoparametric Elements in NISA Library

- The arrangement of a given number of elements in the mesh may influence the results, most noticeably if the mesh is coarse. For example, consider a square clamped plate under uniform pressure loading (Ref. 1) modeled with triangular elements as shown in Figure A.7. Best results are obtained using an element pattern having alternating diagonals (Figure A.7a). The second element pattern is inadequate since all the elements passing through the plate edges are totally constrained (Figure A.7b). The third element pattern shown in Figure A.7c is not a good choice either, since it is not symmetric with respect to the axis of symmetry of the plate.

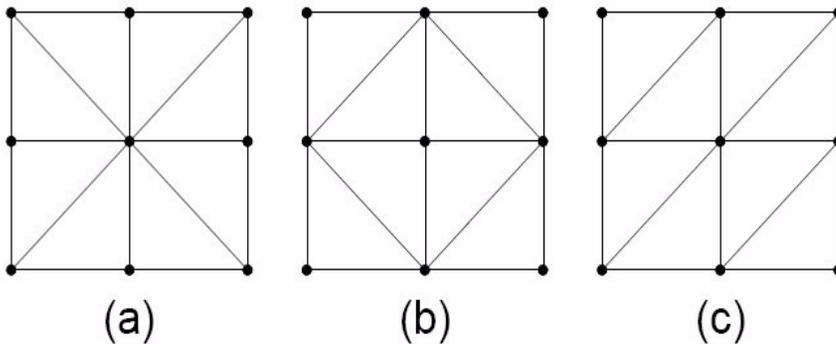


Figure A.7: A Clamped Square Plate under Uniform Pressure Modeled by using Eight Triangular Elements. Elements with dashed edges are Inactive because of the Boundary Conditions.

- Use the executive command ‘AUTO CONSTRAINT’ to suppress spurious normal rotation (about the normal to the shell middle surface) in the model with shell elements (NKTP = 20, 32, 33, 37, 40, 41).
- When the geometry, material and support conditions are symmetrical with respect to a plane, then the model is considered symmetric about that plane. Use of symmetry will allow the user to analyze only one half of the structure, thus saving computer resources. On smaller computers with limited memory, it will allow greater modeling details in the symmetric half.

The symmetrical structure with unsymmetric loading can be analyzed by decomposing the load into symmetrical and antisymmetrical components with respect to the plane of symmetry as shown in Figure A.8 (for linear analysis only). Two load cases should be performed, one for the symmetric component and the other for the anti-

symmetric component. Results under the actual loading may then be obtained using the load combination capability in NISA.

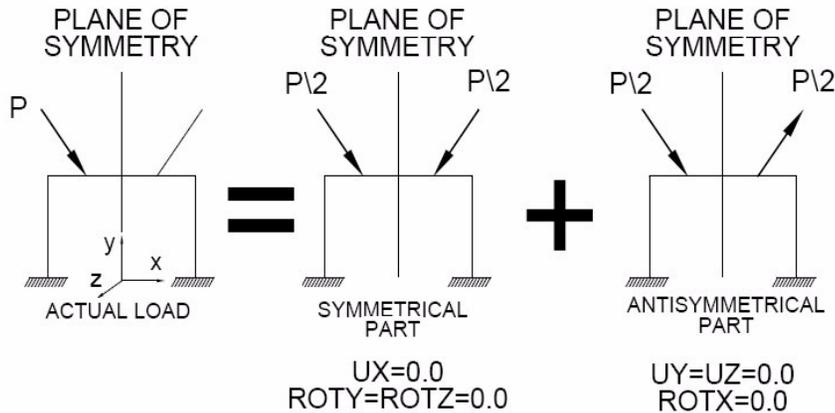


Figure A.8: Example of Decomposition of Unsymmetric Load applied to a Symmetrical Structure into Symmetrical and Antisymmetrical Components

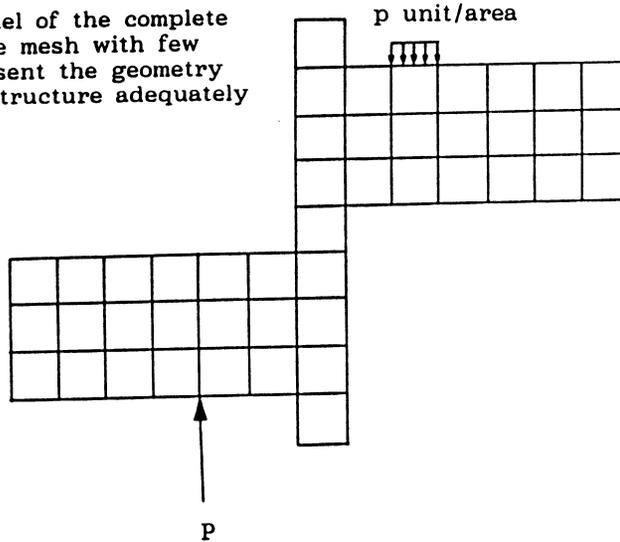
- ❑ The submodeling concept as shown in [Figure A.9](#) can be applied to a coarser model to predict the stresses in the areas of interest more accurately. This will help cut computer cost and allow users of smaller systems to predict stresses by creating relatively coarser mesh for the initial analysis.
- ❑ The output results should be compared against available standards, for example, experimental results, previous similar analysis, results from simplified models, and of course, engineering judgment. The stress results should not be taken seriously unless the displacement results are judged acceptable. Note however that a mesh that gives accurate displacements may be coarse for accurate stress prediction.

Reference

1. R.D. Cook, *Concepts and applications of Finite Element Analysis*, 2nd Edition, John Wiley & Sons, New York, 1981.

step 1 analysis:

finite element model of the complete structure - coarse mesh with few elements to represent the geometry and stiffness of structure adequately



step 2 analysis:

finite element model of the critical region of interest - fine mesh with more elements

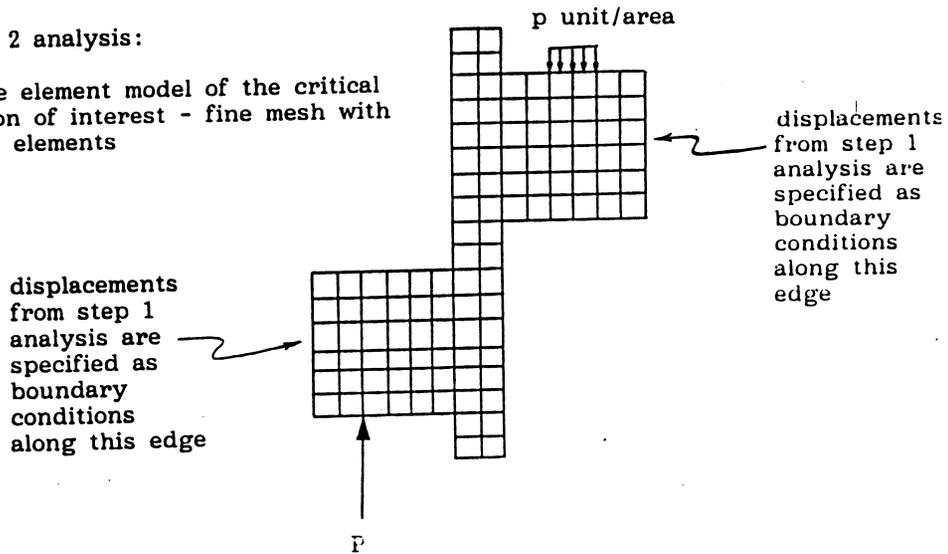


Figure A.9: Submodeling Concept

B

NISA Static Analysis Example

Title:

A hook under concentrated force and pressure loading.

Element Type:

2-D, 8 node plane stress element (NKTP = 1, NORDR = 2)

Problem:

To illustrate the use of NISA, an example of a hook under two different loading conditions is considered. A hook as shown in [Figure B.1](#) is subjected to a concentrated force and distributed pressure loadings, separately, as shown in [Figure B.2](#).

The resultant deflections and the stresses are to be computed.

Properties:

Material:

EX = 30×10^6 lb/in² (Modulus of elasticity)

NUXY = 0.3 (Poisson's ratio)

Geometry:

t = 0.125 in (Plate thickness)

Finite Element Model:

Since the geometry and the loading conditions of the hook are symmetric about the Y-axis, only one half of the structure is modeled as shown in [Figure B.1](#). The structure is analyzed for two different load cases, namely, a concentrated nodal force $FX = 450$ lb at node 31, and distributed pressure loading $P = 1000$ psi on face 4 of elements 3 and 5, as shown in [Figure B.2](#). The symmetry boundary condition is specified for the nodes lying on the global Y-axis (nodes 1 to 5). To suppress the rigid body translation in the global Y-direction, a vertical constraint is applied to node 5.

Output:

- Displacements for nodes 11 to 33
- Reactions
- Gauss point and node point global stresses for element 1, load case 1
- Average nodal stresses at nodes 1, 6 and 11
- Nodal principal stresses (with direction cosines), maximum shear stress, von Mises equivalent stress and octahedral shear stress at nodes 1, 6 and 11.

Postprocessing:

NISA files 26 and 27 are saved for the purpose of post-processing. These files contain all the information necessary to obtain deformed geometry and stress distribution plots using the DISPLAY-POST program. [Figure B.3](#) shows the deformed geometry plot of the hook for the first load case. [Figure B.4](#) shows the stress distribution in the hook for the first load case. [Figure B.5](#) shows the deformed geometry of the hook for the second load case, and [Figure B.6](#) shows the stress distribution in the hook for the second load case.

Listings:

The listing of the following files is attached herewith

1. Input file
2. Output file

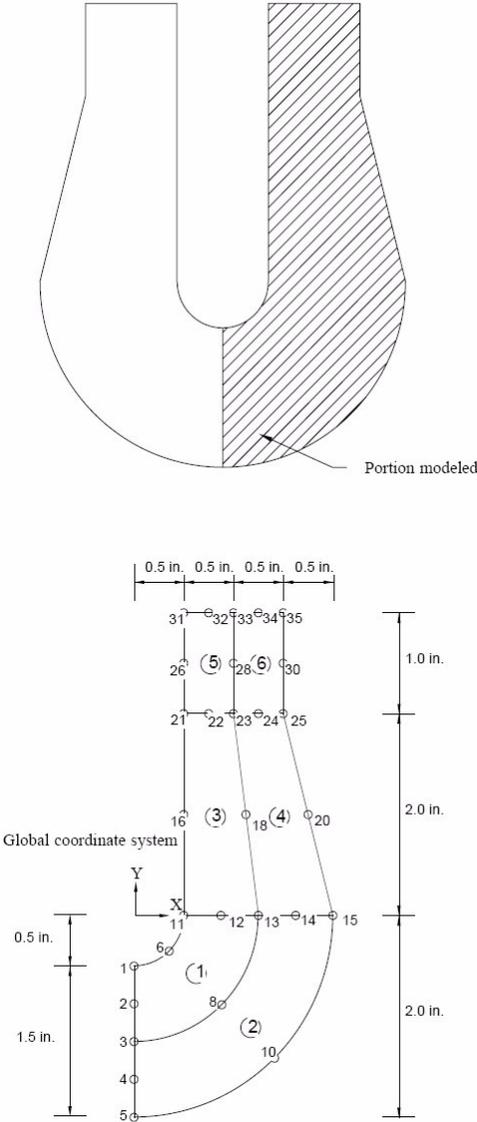


Figure B.1: Geometry and finite element mesh for the hook

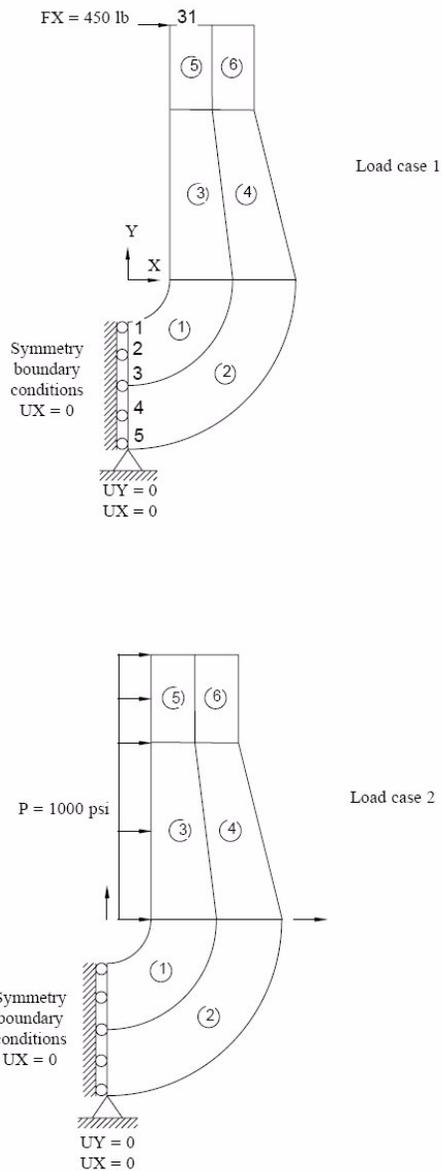


Figure B.2: Loading conditions for the hook

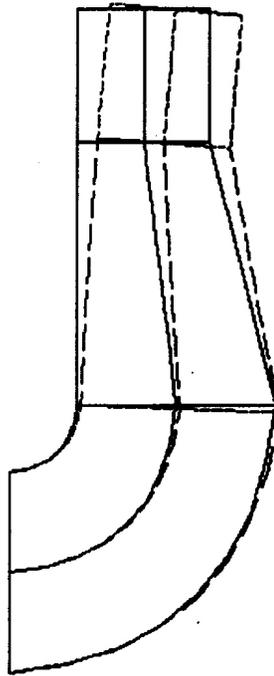


Figure B.3: Deformed geometry plot for the first load case

DISPLAY III - GEOMETRY MODELING SYSTEM (15.1.1.0) PRE/POST MODULE

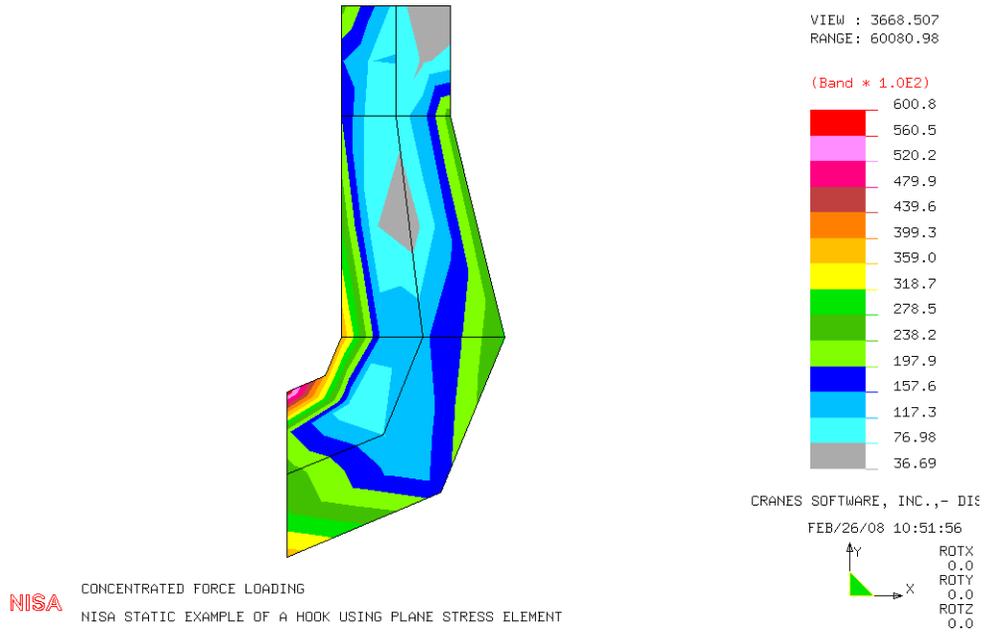


Figure B.4: Stress distribution in hook for the first load case

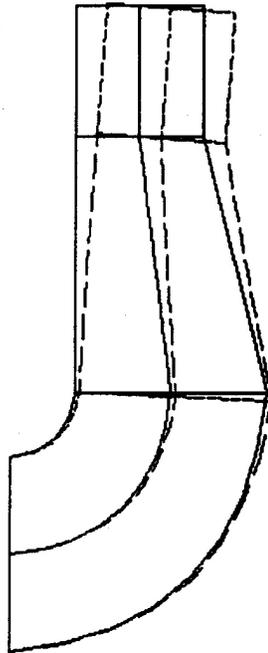


Figure B.5: Deformed geometry plot for the second load case

DISPLAY III - GEOMETRY MODELING SYSTEM (15.1.0) PRE/POST MODULE

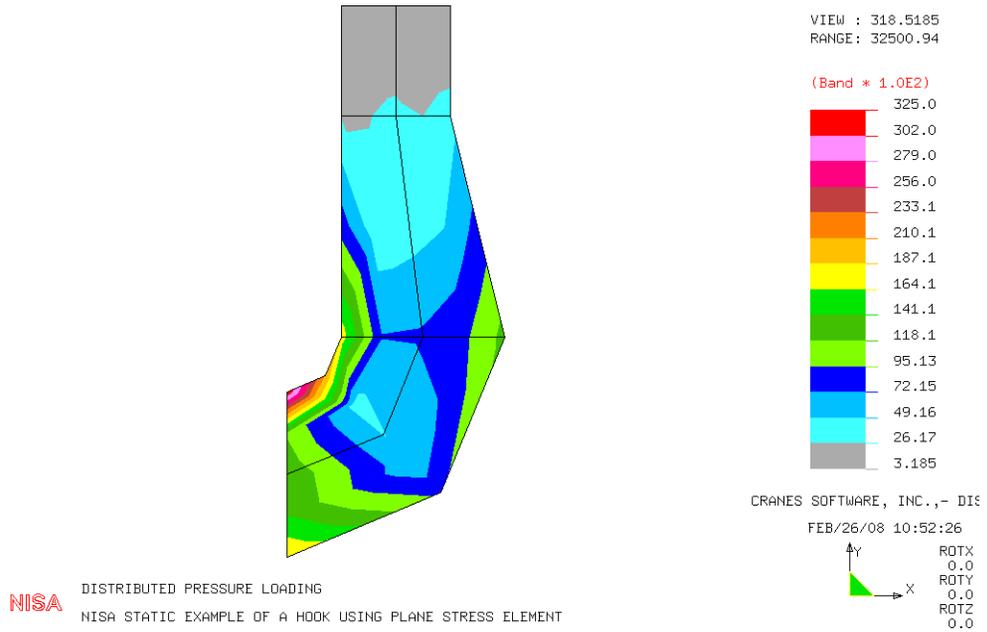


Figure B.6: Stress distribution in hook for the second load case

Input Data Listing

```

**-----EXECUTIVE COMMANDS-----**
ANALYSIS = STATIC
RESEQ = OFF
FILE = HOOK
SAVE = 26,27
**-----MODEL DATA BLOCK-----** *TITLE
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
*ELTYPE
** Select 2-D 8-noded plane stress elements (NKTP=1,NORDR=2) **
1,1,2
*RCTABLE
1,8
0.125////////
*ELEMENTS
** Define element connectivity using second level generation **
$-3,10,2,2
1,1,1,1,0, 2,2,1,1
1,2,3,8,13,12,11,6
*NODES
** Define node coordinates using first and second level generation **
-10, 1, 5, 2,  0.0, 90.0, 0
1, 1, 0, 0,  0.5,-90.0, 0
5, 1, 1, 0,  2.0,-90.0, 0
6, 1, 0, 0,  0.5,-45.0, 0
10, 1, 2, 0,  2.0,-45.0, 0
16, 0, 0, 0,  0.5, 1.0, 0
20, 0, 2, 0,  1.75, 1.0, 0
-10, 0, 5, 2,  0.0, 1.0, 0
21, 0, 0, 0,  0.5, 2.0, 0
25, 0, 1, 0,  1.5, 2.0, 0
26, 0, 0, 0,  0.5, 2.5, 0
30, 0, 2, 0,  1.5, 2.5, 0
*MATERIAL
EX, 1, 0, 30.0E6
NUXY, 1, 0, 0.3
*SETS

```

NISA Static Analysis Example

** Define sets for the selective printout purpose **

101,S,1

102,S,1,6,11

103,R,11,35,1

-----ANALYSIS DATA BLOCK-----

** Beginning of load case number 1 **

*LDCASE,ID =1

0,1,3,0, 1,1,0,0.0,0.0

*LCTITLE

CONCENTRATED FORCE LOADING

*CFORCE

31, FX, 450.0

*SPDISP

1, UX, 0.0, 4, 1

5, UX, 0.0, 0, 0, UY

*PRINTCNTL

ELST,101

AVND,102

DISP,103

** Beginning of load case number 2 **

*LDCASE,ID =2

0,1,3,0, 1,1,0,0.0,0.0

*LCTITLE

DISTRIBUTED PRESSURE LOADING

*PRESSURE

3,5,2,4,0,0, 1000.0

*PRINTCNTL

AVND,102

DISP,103

-----DATA DECK TERMINATOR-----

*ENDDATA

1

I
CRANES SOFTWARE, INC. MAKES NO WARRANTY OR REPRESENTATIONS IN CONNECTION WITH NISA II, DISPLAY III, DISPLAY IV, NISA-COMPOSITE, ENDURE, NISAOPT, FEAP, NISA/3D-FLUID, DYMES OR ANY OTHER SOFTWARE PROGRAM OF CRANES SOFTWARE, INC. (HEREINAFTER REFERRED TO AS "NISA/DISPLAY SOFTWARE"), EXPRESS OR IMPLIED, INCLUDING ANY IMPLIED WARRANTY OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. CRANES SOFTWARE, INC. MAKES NO WARRANTY AND ASSUMES NO LIABILITY FOR THE NISA/DISPLAY SOFTWARE.

CRANES SOFTWARE, INC. ASSUMES NO RESPONSIBILITY FOR THE USE OF THE PROGRAMS OR FOR THE ACCURACY OR VALIDITY OF ANY RESULTS OBTAINED FROM THE NISA/DISPLAY SOFTWARE. CRANES SOFTWARE, INC. SHALL NOT BE LIABLE FOR LOSS OF PROFIT, LOSS OF BUSINESS, OR OTHER FINANCIAL LOSS WHICH MAY BE CAUSED, DIRECTLY OR INDIRECTLY BY THE NISA/DISPLAY SOFTWARE, WHEN USED FOR ANY PURPOSE OR USE, OR DUE TO ANY DEFECT OR DEFICIENCY THEREIN.

EXCLUSION OF INCIDENTAL, CONSEQUENTIAL AND CERTAIN OTHER DAMAGES. TO THE MAXIMUM EXTENT PERMITTED BY APPLICABLE LAW, IN NO EVENT SHALL CSI OR ITS SUPPLIERS BE LIABLE FOR ANY SPECIAL, INCIDENTAL, PUNITIVE, INDIRECT, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING, BUT NOT LIMITED TO, DAMAGES FOR LOSS OF PROFITS OR CONFIDENTIAL OR OTHER INFORMATION, FOR BUSINESS INTERRUPTION, FOR PERSONAL INJURY, FOR LOSS OF PRIVACY, FOR FAILURE TO MEET ANY DUTY INCLUDING OF GOOD FAITH OR OF REASONABLE CARE, FOR NEGLIGENCE, AND FOR ANY OTHER PECUNIARY OR OTHER LOSS WHATSOEVER) ARISING OUT OF OR IN ANY WAY RELATED TO THE USE OF OR INABILITY TO USE THE SOFTWARE, THE PROVISION OF OR FAILURE TO PROVIDE SUPPORT OR OTHER SERVICES, INFORMATION, SOFTWARE, AND RELATED CONTENT THROUGH THE SOFTWARE OR OTHERWISE ARISING OUT OF THE USE OF THE SOFTWARE, OR OTHERWISE EVEN IN THE EVENT OF THE FAULT, TORT (INCLUDING NEGLIGENCE), MISREPRESENTATION, STRICT LIABILITY, BREACH OF CONTRACT OR BREACH OF WARRANTY OF CSI OR ANY SUPPLIER, AND EVEN IF CSI OR ANY SUPPLIER HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

Any questions relating to the use or interpretation of the SOFTWARE or their operation should be directed to:

CRANES SOFTWARE, INC.
1607, East Big Beaver Road, Suite 250, Troy, MI 48063, U.S.A.
P.O. Box 696, Troy, MI 48099 U.S.A.
Tel: 248/689-0077. Fax: 248/689-7479. WEB: www.nisasoftware.com.

NOTICE FOR U.S. GOVERNMENT USERS ONLY: RESTRICTED RIGHTS LEGEND.

Use, duplication or disclosure by the Government is subject to restrictions as set forth in subparagraph (c)(1)(ii) of the Rights in Technical Data and Computer Software Clause at DFARS 252.227-7013.

1

LISTING OF EXECUTIVE COMMANDS

```
LINE 1 **** NISA file: DISPLAY VERSION - 15.0 PRODUCTION: ****
LINE 2 ** This NISA file is written out by DISPLAY-III FEA program
LINE 3 ** All ** lines are comment cards except lines with **DISP3:
LINE 4 ** labels which have special meanings and retained in NISA file
LINE 5 ** for compatibility with DISP3 database. Pls do not modify them
LINE 6 ****
LINE 7 **EXECUTIVE data deck
LINE 8 ANALYSIS = STATIC
LINE 9 SOLV = FRON
```

```

LINE 11 RESEQUENCE = OFF
LINE 12 SAVE = 26,27
LINE 13 TITLE
*** CRANES SOFTWARE INC. NISA2 *** -- Version 16.0 (01/15/08-ALLMEG) FEB/26/2008 10: 1:47

```

NISA COMPUTER PROGRAM RELEASE NO. 16.0

STATIC ANALYSIS

```

PROBLEM TITLE (*TITLE DATA GROUP)
-----
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
*** CRANES SOFTWARE INC. NISA2 *** -- Version 16.0 (01/15/08-ALLMEG) FEB/26/2008 10: 1:48
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT

```

SELECTION OF ELEMENT TYPES FROM THE NISA ELEMENT LIBRARY (*ELTYPE DATA GROUP)

NSRL	NKTP	NORDR	NODES/EL	DOF/NODE
1	1	2	8	2

```

*** CRANES SOFTWARE INC. NISA2 *** -- Version 16.0 (01/15/08-ALLMEG) FEB/26/2008 10: 1:48
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT

```

TABLE OF REAL CONSTANTS (*RCTABLE DATA GROUP)

INDEXRC /----- VALUES OF REAL CONSTANTS -----/

```

1 1.250000E-01 1.250000E-01 1.250000E-01 1.250000E-01 1.250000E-01 1.250000E-01 1.250000E-01
*** CRANES SOFTWARE INC. NISA2 *** -- Version 16.0 (01/15/08-ALLMEG) FEB/26/2008 10: 1:48
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT

```

--- NODAL COORDINATES (*NODES DATA GROUP) ---

NODE NO.	COORD SYS. ID.	COORDINATES	ROTATED SYS. ID.		
			1	2	3
1	1	5.000000E-01	-9.000000E+01	0.000000E+00	0
2	1	8.750000E-01	-9.000000E+01	0.000000E+00	0
3	1	1.250000E+00	-9.000000E+01	0.000000E+00	0
4	1	1.625000E+00	-9.000000E+01	0.000000E+00	0

```

LINE 11 RESEQUENCE = OFF
LINE 12 SAVE = 26,27
LINE 13 *TITLE
*** CRANES SOFTWARE INC. NISA2 *** -- Version 16.0 (01/15/08-ALLMEG) FEB/26/2008 10: 1:48

```

NISA COMPUTER PROGRAM RELEASE NO. 16.0

STATIC ANALYSIS

PROBLEM TITLE (*TITLE DATA GROUP)

```

NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
*** CRANES SOFTWARE INC. NISA2 *** -- Version 16.0 (01/15/08-ALLMEG) FEB/26/2008 10: 1:48
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT

```

SELECTION OF ELEMENT TYPES FROM THE NISA ELEMENT LIBRARY (*ELTYPE DATA GROUP)

MSRL	NKTP	NORDR	NODES/EL	DOF/NODE
1	1	2	8	2

```

*** CRANES SOFTWARE INC. NISA2 *** -- Version 16.0 (01/15/08-ALLMEG) FEB/26/2008 10: 1:48
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT

```

TABLE OF REAL CONSTANTS (*RCTABLE DATA GROUP)

INDEXRC /----- VALUES OF REAL CONSTANTS -----/

```

1 1.250000E-01 1.250000E-01 1.250000E-01 1.250000E-01 1.250000E-01 1.250000E-01 1.250000E-01
*** CRANES SOFTWARE INC. NISA2 *** -- Version 16.0 (01/15/08-ALLMEG) FEB/26/2008 10: 1:48
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT

```

-- NODAL COORDINATES (*NODES DATA GROUP) --

NODE NO.	COORD SYS. ID.	COORDINATES	ROTATED SYS. ID.	
			1	2
1	1	-9.00000E+01	0.00000E+00	0
2	1	8.75000E-01	0.00000E+00	0
3	1	1.25000E+00	0.00000E+00	0
4	1	1.62500E+00	0.00000E+00	0

103 R 11, 35, 1 --- Version 16.0 (01/15/08-ALLMEG) LOAD CASE ID NO. 1 FEB/26/2008 10: 1:48
 *** CRANES SOFTWARE INC. NISA ***
 NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT

OUTPUT CONTROL FOR LOAD CASE ID NO. 1

INTERNAL FORCE AND STRAIN ENERGY KEY ... (KELFR) = 0
 REACTION FORCE KEY ... (KRCTN) = 1
 STRESS COMPUTATION KEY ... (KSTR) = 3
 STRAIN COMPUTATION KEY ... (KSTN) = 0
 ELEMENT STRESS/STRAIN OUTPUT OPTIONS ... (LQ1) = 1
 NODAL STRESSES OUTPUT OPTIONS ... (LQ2) = 1
 DISPLACEMENT OUTPUT OPTIONS ... (LQ7) = 0
 STRESS FREE TEMPERATURE ... (TSFRE) = 0.00000E+00
 TOLERANCE FOR NODE POINT FORCE BALANCE (TOL) = 0.00000E+00
 *** CRANES SOFTWARE INC. NISA *** -- Version 16.0 (01/15/08-ALLMEG)
 NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT

LOAD CASE TITLE (*LCITITLE DATA GROUP) - LOAD CASE ID. 1

CONCENTRATED FORCE LOADING
 *** CRANES SOFTWARE INC. NISA *** -- Version 16.0 (01/15/08-ALLMEG) LOAD CASE ID NO. 1 FEB/26/2008 10: 1:48
 NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
 CONCENTRATED FORCE LOADING

SPECIFIED DISPLACEMENT DATA (*SPOISP DATA GROUP)

NODE NO.	LABEL	DISPLACEMENT VALUE	LAST NODE	INC	LABELS
1	UX	0.00000E+00	1	1	
2	UX	0.00000E+00	2	1	
3	UX	0.00000E+00	3	1	
4	UX	0.00000E+00	4	1	
5	UX	0.00000E+00	5	1	
5	UY	0.00000E+00	5	1	

*** CRANES SOFTWARE INC. NISA *** -- Version 16.0 (01/15/08-ALLMEG) LOAD CASE ID NO. 1 FEB/26/2008 10: 1:48
 NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
 CONCENTRATED FORCE LOADING

CONCENTRATED NODAL FORCE AND MOMENT DATA (*CFORCE DATA GROUP)

NODE NO.	LABEL	FORCE VALUE	LASTNOD	INC	LFN
31	FX	4.50000E+02	31	1	0

*** CRANES SOFTWARE INC. NISA *** -- Version 16.0 (01/15/08-ALLMEG) LOAD CASE ID NO. 1 FEB/26/2008 10: 1:48
 NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
 CONCENTRATED FORCE LOADING

SELECTIVE PRINTOUT CONTROL PARAMETERS (*PRINTCNTL DATA GROUP)

OUTPUT TYPE --- SET NUMBERS (NEGATIVE MEANS NONE, ZERO MEANS ALL)

LOAD VECTOR

```

ELEMENT INTERNAL FORCES      -1
ELEMENT STRAIN ENERGY      -1
RIGID LINK FORCES          -1
REACTIONS                   0
DISPLACEMENTS              103
ELEMENT STRESSES            101
AVERAGED NODAL STRESSES    102
VELOCITIES                  0
ACCELERATIONS               0
SUMMARY OF LINE FORCES
*** CRANES SOFTWARE INC. NISA ***  -- Version 16.0 (01/15/08--ALLMEG)
*** NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
LOAD CASE ID NO.          2      FEB/26/2008  10: 1:48

-----
OUTPUT CONTROL FOR LOAD CASE ID NO.      2
-----

INTERNAL FORCE AND STRAIN ENERGY KEY ... (KELFR)= 0
REGION FOR KEY ..... (KSTR)= 1
STRESS COMPUTATION KEY ..... (KSTRN)= 1
STRAIN COMPUTATION KEY ..... (KSTN)= 0
ELEMENT STRESS/STRAIN OUTPUT OPTIONS ... (LO1)= 1
NODAL STRESSES/OUTPUT OPTIONS ..... (LQ2)= 1
DISPLACEMENT OUTPUT OPTIONS ..... (LQ7)= 0
STRESS FREE TEMPERATURE ..... (TSFRE)= 0.00000E+00
TOLERANCE FOR NODE POINT FORCE BALANCE (TOL)= 0.00000E+00
*** CRANES SOFTWARE INC. NISA ***  -- Version 16.0 (01/15/08--ALLMEG)
*** NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
LOAD CASE ID NO.          2      FEB/26/2008  10: 1:48

-----
LOAD CASE TITLE (*LCITITLE DATA GROUP) - LOAD CASE ID.      2
-----
DISTRIBUTED PRESSURE LOADING
*** CRANES SOFTWARE INC. NISA ***  -- Version 16.0 (01/15/08--ALLMEG)
*** NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
DISTRIBUTED PRESSURE LOADING

DISTRIBUTED ELEMENT PRESSURE DATA (*PRESSURE DATA GROUP)
-----
ELE, NO.=      3  LFN=      0  FACE=      4  ITCRV=      0  IVECT=      0
ELE 1.00000E+03 (UNIFORM)
ELE 1.00000E+03 (UNIFORM)
*** CRANES SOFTWARE INC. NISA ***  -- Version 16.0 (01/15/08--ALLMEG)
*** NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
DISTRIBUTED PRESSURE LOADING

SELECTIVE PRINTOUT CONTROL PARAMETERS (*PRINTCNTL DATA GROUP)
-----
OUTPUT TYPE      -- SET NUMBERS (NEGATIVE MEANS NONE, ZERO MEANS ALL)
LOAD VECTOR      -1
ELEMENT INTERNAL FORCES      -1
ELEMENT STRAIN ENERGY      -1

```

```

REACTIONS
DISPLACEMENTS      0
ELEMENT STRESSES    103
AVERAGED NODAL STRESSES  -1
VELOCITIES          102
ACCELERATIONS       0
SUMMARY OF LINE FORCES
*** CRANES SOFTWARE INC. NISA2 *** -- Version 16.0 (01/15/08-ALLMEG)
*** NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
FEB/26/2008 10: 1:48
    
```

PROCESS NODAL COORDINATES DATA

```

PROCESS ELEMENT CONNECTIVITY DATA
*** CRANES SOFTWARE INC. NISA2 *** -- Version 16.0 (01/15/08-ALLMEG)
*** NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
FEB/26/2008 10: 1:48
    
```

SUMMARY OF ELEMENT TYPES USED

NKTP	NORDR	NO. OF ELEMENTS
1	2	6

```

TOTAL NUMBER OF ELEMENTS ..... = 6
TOTAL NUMBER OF NODES ..... = 29
TOTAL NUMBER OF ACTIVE NODES ..... = 29
LARGEST NODE NUMBER ..... = 35
    
```

```

MINIMUM X-COORD = 0.13242E-13 MAXIMUM X-COORD = 0.20000E+01
MINIMUM Y-COORD = -0.20000E+01 MAXIMUM Y-COORD = 0.30000E+01
MINIMUM Z-COORD = 0.00000E+00 MAXIMUM Z-COORD = 0.00000E+00
    
```

```

*** CRANES SOFTWARE INC. NISA *** -- Version 16.0 (01/15/08-ALLMEG)
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
DISTRIBUTED PRESSURE LOADING
LOAD CASE ID NO. 2 FEB/26/2008 10: 1:48
    
```

GEOMETRIC PROPERTIES OF THE MODEL

```

TOTAL VOLUME = 8.01317E-01
TOTAL MASS = 0.00000E+00
X COORDINATE OF C.G. = 0.00000E+00
Y COORDINATE OF C.G. = 0.00000E+00
Z COORDINATE OF C.G. = 0.00000E+00
MASS MOMENT OF INERTIA WITH RESPECT TO GLOBAL AXES AT GLOBAL ORIGIN
IXX = 0.00000E+00
IYY = 0.00000E+00
IZZ = 0.00000E+00
IYZ = 0.00000E+00
MASS MOMENT OF INERTIA WITH RESPECT TO CARTESIAN AXES AT C.G.
    
```

```

IYY = 0.000000E+00      IYZ = 0.000000E+00
IZZ = 0.000000E+00      IXZ = 0.000000E+00

PROCESS *SPDISP (SPECIFIED DISPLACEMENT) DATA FOR LOAD CASE ID NO.      1

TOTAL NUMBER OF VALID DOFS IN MODEL .....= 58
TOTAL NUMBER OF UNCONSTRAINED DOFS .....= 52
TOTAL NUMBER OF CONSTRAINED DOFS .....= 6
TOTAL NUMBER OF SLAVES IN MPC EQS .....= 0
*** CRANES SOFTWARE INC. NISA *** -- VERSTION 16.0 (01/15/08--ALLMEG)
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
CONCENTRATED FORCE LOADING
LOAD CASE ID NO.      1      FEB/26/2008      10: 11:48
    
```

```

*** WAVE FRONT SOLUTION PARAMETERS ***
MAXIMUM WAVEFRONT (MAXPA)      = 20
R.M.S. WAVEFRONT              = 18
AVERAGE WAVEFRONT            = 18
LARGEST ELEMENT MATRIX RANK USED (LWMAX) = 16
TOTAL NUMBER OF DEGREES OF FREEDOM = 52
ESTIMATED NUMBER OF BUFFERS ON FILE 30 = 1
MAXIMUM BUFFER SIZE          = 65536
PHYSICAL RECORD LENGTH ON FILE 30 = 2048
EQUATION NUMBER SET OF WORDS ON FILE 30 = 2048
NUMBER OF GROUPS OF ELIMINATED EQNS. IN EACH BLOCK:
--Comp1-- 0  --Comp2-- 2  --Comp3-- 2  --Comp4-- 2  --Comp5-- 0  --Comp6-- 3  --Comp7-- 0  --Comp8-- 2
    
```

```

DECOMPOSITION COMPLETED IN      0.000 SECONDS
MAXIMUM PIVOT                    = 0.3256302E+08
MINIMUM PIVOT                    = 0.5341853E+05
TOTAL NUMBER OF BUFFERS USED ON FILE 30 = 1
TOTAL NO. OF PHYSICAL RECORDS WRITTEN = 1
SIZE OF FILE 30 IN WORDS        = 2048
*** CRANES SOFTWARE INC. NISA *** -- VERSTION 16.0 (01/15/08--ALLMEG)
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
CONCENTRATED FORCE LOADING
LOAD CASE ID NO.      1      FEB/26/2008      10: 11:48
    
```

***** REACTION FORCES AND MOMENTS AT NODES *****

LOAD CASE ID NO.	1	FX	FY	FZ	MX	MY	MZ
1	-8.56992E+02	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
2	-1.39843E+03	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
3	3.15872E+02	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
4	9.05824E+02	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
5	5.40879E+02	-4.72937E-11	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00

1. 2 5.272118E+03 1.087556E+04 -5.167264E+03 3.8137E-01 -1.0115E+00
 3 1.657596E+04 2.676767E+04 1.068441E+04 6.1026E-01 -2.3008E-01
 4 9.4652801E+03 2.693456E+02 -1.312854E+03 1.0115E+00 -3.8137E-01

ELEMENT 1 -----GLOBAL STRESSES-----
 NODE POINT 1 SX SY SZ SXY SZX SYZ SZY SZX SYZ SZY SZX SYZ SZX SYZ
 1 5.8212194E+04 6.907601E+03 1.396820E+04 1.396820E+04
 2 1.359395E+04 1.188437E+04 3.42833E+02 3.42833E+02
 3 1.012676E+04 1.994020E+04 -8.09137E+03 -8.09137E+03
 4 1.136276E+04 -1.647433E+04 -7.994833E+03 -7.994833E+03
 11 1.030478E+04 1.395638E+04 5.688815E+03 5.688815E+03
 12 8.656800E+03 4.438708E+04 1.537248E+04 1.537248E+04
 6 3.398937E+04 2.564734E+04 1.467033E+04 1.467033E+04
 *** CRANES SOFTWARE INC. NISA *** -- Version 16.0 (01/15/08-ALLMEG) LOAD CASE ID NO. 1 FEB/26/2008 10: 1:48
 NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
 CONCENTRATED FORCE LOADING

**** AVERAGE NODAL STRESSES - LOAD CASE ID NO. 1 ****
 NODE 1 SX SY SZ SXY SZX SYZ SZY SZX SYZ SZY SZX SYZ SZX SYZ
 1 5.81219E+04 6.90760E+03 1.38683E+04 1.38683E+04 0.00000E+00 0.00000E+00 0.00000E+00
 2 3.39894E+04 2.56473E+04 1.46703E+04 1.46703E+04 0.00000E+00 0.00000E+00 0.00000E+00
 11 3.30352E+03 3.95511E+04 0.00000E+00 6.91866E+03 0.00000E+00 0.00000E+00 0.00000E+00
 LARGEST MAGNITUDES OF STRESS COMPONENTS
 AT NODES 1 11
 5.81219E+04 3.92511E+04 0.00000E+00 1.46703E+04 0.00000E+00 0.00000E+00
 *** CRANES SOFTWARE INC. NISA *** -- Version 16.0 (01/15/08-ALLMEG) 6 LOAD CASE ID NO. 1 FEB/26/2008 10: 1:48
 NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
 CONCENTRATED FORCE LOADING

**** NODAL PRINCIPAL STRESSES - LOAD CASE ID NO. 1 ****
 NODE 1 SG1 SG2 SG3 MAX. SHEAR EQUIV. STRESS VON MISES OCTAHEDRAL SHEAR
 1 6.16839E+04 3.34565E+03 0.00000E+00 3.08419E+04 6.00810E+04 2.83224E+04
 6 4.46908E+04 -0.24702E+00 0.00000E+00 2.23454E+04 3.95214E+04 1.86306E+04
 11 0.79206E+00 -0.61007E+00 0.00000E+00 2.02171E+04 3.94847E+04 1.86133E+04
 4.05434E+04 2.21021E+03 0.00000E+00 2.02171E+04 3.94847E+04 1.86133E+04
 0.18409E+00 0.98301E+00 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00

LARGEST MAGNITUDES OF STRESS FUNCTIONS
 6.16839E+04 1.43459E+04 -3.39340E+04 3.08419E+04 6.00810E+04 2.83224E+04
 *** CRANES SOFTWARE INC. NISA *** -- Version 16.0 (01/15/08-ALLMEG) 5 LOAD CASE ID NO. 1 FEB/26/2008 10: 1:48
 NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
 CONCENTRATED FORCE LOADING

```

T I M E L O G I N S E C O N D S
LOAD CASE ID NO. 1
INPUT ( READ, GENERATE ) .....= 0.063
DATASORTING AND CHECKING .....= 0.000
REORDERING OF ELEMENTS .....= 0.000
FORM ELEMENT MATRICES .....= 0.000
FORM GLOBAL LOAD VECTOR .....= 0.000
MATRIX TRANSFORMATION DUE TO MPC .....= 0.000
PRE-FRONT .....= 0.000
SOLUTION OF SYSTEM EQUATIONS .....= 0.000
INTERNAL FORCES AND REACTIONS .....= 0.000
STRESS CALCULATION .....= 0.016
TOTAL CPU .....= 0.078

*** CRANES SOFTWARE INC. NISA *** -- Version 16.0 (01/15/08-ALLMEG) LOAD CASE ID NO. 2 FEB/26/2008 10: 1:48
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
DISTRIBUTED PRESSURE LOADING
***** REACTION FORCES AND MOMENTS AT NODES *****
LOAD CASE ID NO. 2

```

NODE	FX	FY	FZ	MX	MY	MZ
1	-4.80241E+02	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
2	-7.62262E+02	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
3	1.50484E+02	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
4	4.06785E+02	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
5	3.10235E+02	-2.22826E-11	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00

SUMMATION OF REACTION FORCES IN GLOBAL DIRECTIONS

```

-----
FX FY FZ
*** CRANES SOFTWARE INC. NISA *** -- Version 16.0 (01/15/08-ALLMEG) LOAD CASE ID NO. 2 FEB/26/2008 10: 1:48
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
DISTRIBUTED PRESSURE LOADING
***** DISPLACEMENT SOLUTION *****
LOAD CASE ID NO. 2

```

NODE	UX	UY	UZ	ROTX	ROTY	ROTZ
11	1.25692E-03	1.87116E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
12	1.21901E-03	-4.10094E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
13	1.23840E-03	-9.49152E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
14	1.27477E-03	-1.53590E-03	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
15	1.30631E-03	-2.12483E-03	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
16	3.23676E-03	5.46057E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
18	3.18458E-03	-7.64853E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
20	3.19304E-03	-1.98239E-03	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
21	5.52706E-03	7.03375E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
22	5.52486E-03	1.21367E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00
23	5.51347E-03	-4.156448E-04	0.00000E+00	0.00000E+00	0.00000E+00	0.00000E+00

```

24 5.50482E-03 -1.04875E-03 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
25 5.51267E-03 -1.64430E-03 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
26 6.71099E-03 7.40408E-04 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
28 6.73672E-03 -4.58257E-04 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
30 6.73660E-03 -1.66940E-03 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
31 7.197190E-03 7.52102E-04 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
32 7.196088E-03 1.44442E-04 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
33 7.195960E-03 -4.57869E-04 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
34 7.195983E-03 -1.06283E-03 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
35 7.195499E-03 -1.67283E-03 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00

```

LARGEST MAGNITUDES OF DISPLACEMENT VECTOR =

```

7.97190E-03 -2.12483E-03 0.00000E+00 0.00000E+00 0.00000E+00 0.00000E+00
AT NODE 31
*** CRANES SOFTWARE INC. NISA *** -- Version 16.0 (01/15/08-ALLMIEG) LOAD CASE ID NO. 2 1 FEB/26/2008 10: 1:48
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
DISTRIBUTED PRESSURE LOADING

```

**** AVERAGE NODAL STRESSES - LOAD CASE ID NO. 2 ****

NODE	SX	SY	SZ	SXY	SYZ	SZX
1	3.17680E+04	4.17071E+03	0.00000E+00	7.35308E+03	0.00000E+00	0.00000E+00
6	1.83519E+04	1.22523E+04	0.00000E+00	7.61411E+03	0.00000E+00	0.00000E+00
11	1.14570E+03	1.67399E+04	0.00000E+00	4.24891E+03	0.00000E+00	0.00000E+00

LARGEST MAGNITUDES OF STRESS COMPONENTS

```

AT NODES 1, 11
3.17680E+04 1.67399E+04 0.00000E+00 7.61411E+03 0.00000E+00 0.00000E+00
*** CRANES SOFTWARE INC. NISA *** -- Version 16.0 (01/15/08-ALLMIEG) 6 LOAD CASE ID NO. 2 1 FEB/26/2008 10: 1:48
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
DISTRIBUTED PRESSURE LOADING

```

**** NODAL PRINCIPAL STRESSES - LOAD CASE ID NO. 2 ****

NODE	SG1	SG2	SG3	MAX. SHEAR	EQUITY STRESS VON MISES	OCTAHEDRAL SHEAR
1	3.36049E+04	2.33381E+03	0.00000E+00	1.68025E+04	3.25009E+04	1.53211E+04
6	0.970 0.242 0.000	-0.242 0.970 0.000	0.000 0.000 1.000	1.17585E+04	2.08861E+04	9.84582E+03
11	2.35169E+04	7.12735E+03	0.00000E+00	8.91122E+03	1.77909E+04	8.38673E+03
	0.828 0.561 0.000	-0.561 0.828 0.000	0.000 0.000 1.000	0.00000E+00	0.00000E+00	0.00000E+00
	1.78224E+04	6.31610E+01	0.00000E+00	0.000 0.000 1.000		
	0.247 0.969 0.000	0.969 -0.247 0.000	0.000 0.000 1.000			

LARGEST MAGNITUDES OF STRESS FUNCTIONS

```

AT NODES 1, 6
3.36049E+04 7.12735E+03 -1.68885E+04
*** CRANES SOFTWARE INC. NISA *** -- Version 16.0 (01/15/08-ALLMIEG) 5 LOAD CASE ID NO. 2 1 FEB/26/2008 10: 1:48
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
DISTRIBUTED PRESSURE LOADING

```

```

I T I M E L O G I N S E C O N D S
LOAD CASE ID NO. 2
INPUT ( READ,GENERATE ) .....= 0.000
DATA SORTING AND CHECKING .....= 0.000
REORDERING OF ELEMENTS .....= 0.000
FORM ELEMENT MATRICES .....= 0.000
FORM GLOBAL LOAD VECTOR .....= 0.000
MATRIX TRANSFORMATION DUE TO MPC .....= 0.000
PRE-FRONT .....= 0.000
SOLUTION OF SYSTEM EQUATIONS .....= 0.000
INTERNAL FORCES AND REACTIONS .....= 0.000
STRESS CALCULATION .....= 0.000
TOTAL CPU .....= 0.000

*** CRANES SOFTWARE INC. NISA2 *** -- Version 16.0 (01/15/08-ALLMEG)
NISA STATIC EXAMPLE OF A HOOK USING PLANE STRESS ELEMENT
FEB/26/2008 10: 1:48

O V E R A L L T I M E L O G I N S E C O N D S

INPUT ( READ,GENERATE ) .....= 0.063
DATA SORTING AND CHECKING .....= 0.000
REORDERING OF ELEMENTS .....= 0.000
FORM ELEMENT MATRICES .....= 0.000
FORM GLOBAL LOAD VECTOR .....= 0.000
MATRIX TRANSFORMATION DUE TO MPC .....= 0.000
PRE-FRONT .....= 0.000
SOLUTION OF SYSTEM EQUATIONS .....= 0.000
INTERNAL FORCES AND REACTIONS .....= 0.000
STRESS CALCULATION .....= 0.016
LOAD COMBINATION .....= 0.000
TOTAL CPU .....= 0.078

TOTAL ELAPSED TIME IS .....= 1.000

NO OF FILES USED FOR DECOMPOSITION= 1
FILE UNIT = 30

** NOTE ** - RUN IS COMPLETED SUCCESSFULLY
TOTAL NO. OF COMPLETED CASES= 2
LAST COMPLETED CASE-ID = 2
FILE 26 IS SAVED, NAME=HOOK26.dat, STATUS=NEW
FILE 27 IS SAVED, NAME=HOOK27.dat, STATUS=NEW

** DISK SPACE USED = 238.18 KBYTES

```

C***Quick Reference List*****C.1 Alphabetical Quick Reference List of all Executive Commands**

ACUTOFF
(8.2.3) = $\left[\begin{array}{c} r_c \\ \{3.0\} \end{array} \right]$

ANALYSIS
(5.3.2) = $\left[\begin{array}{c} \{STATIC\} \\ BUCKLING \\ EIGENVALUE \\ NLSTATIC \\ LTRANSIENT \\ NLTRANSIENT \\ SHEAT \\ THEAT \end{array} \right]$

ANALYSIS
(8.2.2) = $\left[\begin{array}{c} \{TRANSIENT\} \\ RANDOM \\ FREQUENCY \\ SHOCK \end{array} \right]$

Quick Reference List

Alphabetical Quick Reference List of all Executive Commands

AUTOCONSTRAINT (5.3.3)	=	[{ON} OFF]
CTAL (5.3.3)	=	[{PENA} AUGM LAGR]
BLANK COMMON (5.3.2), (8.2.2)	=	[n {50000}]
CFREQ (8.2.3)	=	[cfreq {0.0}]
DAMPING (8.2.2)	=	[{VISCOUS} PROPORTIONAL STRUCTURAL]
DELTATIME (8.2.3)	=	[tinc {no default}]
DIRECTION (8.2.3)	=	[{ABS} or {PEAK} SRSS or RMS]
DPROB (8.3.2)	=	[DPROB {0.86}]
EIGEN EXTRACTION (5.3.3)	=	[{SUBSPACE, ACCELERATED} SUBSPACE, CONVENTIONAL INVERSE LANCZOS]

ELEMENT ECHO (5.3.2)	=	$\left[\begin{array}{l} \{ON\} \\ \{OFF\} \end{array} \right]$
ENDTIME (8.2.3)	=	$\left[\begin{array}{l} t \\ \{no\ default\} \end{array} \right]$
EXECUTION (5.3.2), (8.2.2)	=	$\left[\begin{array}{l} \{GO\} \\ CHECK \\ CGO \end{array} \right]$
ERRTOLERANCES (7.1.3)	=	$\left[\begin{array}{l} ERRTOL \\ \{1E-10\} \end{array} \right]$
FILE NAME (5.3.2), (8.2.2)	=	$\left[\begin{array}{l} fname \\ \{no\ default\} \end{array} \right]$
FLOWER (8.2.3)	=	$\left[\begin{array}{l} f_o \\ \{0.3f_1\} \end{array} \right]$
FRQRDF (8.2.3)	=	$\left[\begin{array}{l} r \\ \{1.0\} \end{array} \right]$
FSMALL (8.2.3)	=	$\left[\begin{array}{l} f \\ \{1.0E-8\} \end{array} \right]$
FUPPER (8.2.3)	=	$\left[\begin{array}{l} f_u \\ \{3.0f_n\} \end{array} \right]$

Quick Reference List

Alphabetical Quick Reference List of all Executive Commands

GENFREQUENCY (8.2.3)	=	$\left[\begin{array}{l} \{\text{ON}\} \\ \text{OFF} \end{array} \right]$
GEOM PROPERTIES (5.3.2)	=	$\left[\begin{array}{l} \{\text{ON, LIST}\} \\ \text{ON, NOLIST} \\ \text{OFF} \end{array} \right]$
GFACTOR (8.2.2)	=	$\left[\begin{array}{l} g \\ \{1.0\} \end{array} \right]$
IDNUMBER (5.3.2)	=	$\left[\begin{array}{l} \{\text{REGULAR}\} \\ \text{LARGE} \end{array} \right]$
INITIAL TEMPERATURE (5.3.3)	=	$\left[\begin{array}{l} \text{Temp} \\ \{0.0\} \end{array} \right]$
INPHASE (8.2.3)	=	$\left[\begin{array}{l} \{\text{LINEAR}\} \\ \text{SEMILOG} \\ \text{LOGLOG} \end{array} \right]$
INPOLATION (8.2.3)	=	$\left[\begin{array}{l} \{\text{LINEAR}\} \\ \text{SEMILOG} \\ \text{LOGLOG} \end{array} \right]$
INTEGRATION (8.2.3)	=	$\left[\begin{array}{l} \{\text{NUMERICAL, SIMPSON}\} \\ \text{NUMERICAL, TRAPEZOIDAL} \\ \text{EXACT} \end{array} \right]$

INCREMENTS (7.1.3)	=	$\left[\begin{array}{l} \text{EQUAL, n} \\ \text{USER, n} \\ \text{AUTO, n} \\ \text{ARCL, n} \\ \text{ADAPTIVE, n} \\ \{\text{EQUAL, 10}\} \end{array} \right]$
MASS FORMULATION (5.3.3)	=	$\left[\begin{array}{l} \{\text{CONSISTENT}\} \\ \text{LUMPED} \end{array} \right]$
MAXCPU TIME (5.3.3)	=	$\left[\begin{array}{l} \text{n, m} \\ \{0, 0\} \end{array} \right]$
MRESPONSE (8.2.3)	=	$\left[\begin{array}{l} \{\text{ON}\} \\ \text{OFF} \end{array} \right]$
MSEXCITATION (8.3.2)	=	$\left[\text{ENVE} \right]$
NLTYPE (5.3.3)	=	$\left[\begin{array}{l} \text{MATGEOM} \\ \text{MATERIAL} \\ \text{GEOMETRY} \\ \{\text{LINEAR}\} \end{array} \right]$
NODE ECHO (5.3.2)	=	$\left[\begin{array}{l} \{\text{ON}\} \\ \text{OFF} \end{array} \right]$
ORTHOTROPIC (5.3.3)	=	$\left[\begin{array}{l} \text{n} \\ \{0\} \end{array} \right]$
REFCONFIG (5.3.3)	=	$\left[\begin{array}{l} \{\text{TOTAL}\} \\ \text{UPDATED} \end{array} \right]$

Quick Reference List

Alphabetical Quick Reference List of all Executive Commands

RESEQUENCE (5.3.2)	=	$\left[\begin{array}{l} \{\text{ON}\}, \text{NOKC}, \text{MAX}, \text{LIST} \\ \text{OFF} \end{array} \right]$
RESTART (5.3.2)	=	$\left[\begin{array}{l} n, \text{NOLIST}, \text{NEW30}, \text{USE30} \\ \{0\}, \{\text{LIST}\}, \{\text{OLD30}\} \end{array} \right]$
SAVE FILE (5.3.2), (8.2.2)	=	$\left[\begin{array}{l} n1, n2, \dots \\ \{\text{no default}\} \end{array} \right]$
SIGMA (5.3.3)	=	$\left[\begin{array}{l} \sigma \\ \{\text{no default}\} \end{array} \right]$
SORT STRESS (5.3.3)	=	$\left[\begin{array}{l} n \\ \{n = 25\} \end{array} \right]$
SOLVER (5.3.2)	=	$\left[\begin{array}{l} \{\text{FRON}\} \\ \text{ITER}, \text{TOLE}, \text{NITR}, \text{MPIVT}, \text{CORE} \\ \text{SPAR}, \text{MEMSP} \\ \text{PARDISO}, m, n \end{array} \right]$
STARTING TIME (8.2.3)	=	$\left[\begin{array}{l} t \\ \{0.0\} \end{array} \right]$
STEP (5.3.3)	=	$\left[\begin{array}{l} n \\ \{1\} \end{array} \right]$
THREADS (5.3.2)	=	$\frac{\left[\text{NTHREAD} \right]}{\left[\{I\} \right]}$
TSEISMIC (8.2.3)	=	$\left[\begin{array}{l} \text{tseis} \\ \{30.0\} \end{array} \right]$

WARNING
(5.3.2), (8.2.2)

=

[{GO}
STOP]

C.2 Alphabetical Quick Reference List of All Data Groups

***ADDFREQUENCY (8.5.4)**

j FREQ1, FREQ2, NINCR

***ANGSEC (7.2.3)**

j ANG1, ANG2, ANG3, ANG4, ANG5, ANG6, ANG7, ANG8

***APLASTIC (6.5.7)**

j LABEL, MATID, NTYSSC, NYCRIT

k LABEL, IDIR, TEMPID, VAL1, VAL2, VAL3, VAL4,
 VAL5

***ARRIVALTIME (8.6.1)**

j IARR, ARRT

***BEAMLOAD, TCRV = n (7.3.5)**

§

j NELID, IPL, IRL, IFM, LASTEL, INC, X1, X2, P1, P2, LFN, ITCRV

***BMDATA (6.3.7)**

j IDBM, NODE3, NVECT, NCOD1, NCOD2, INTPR

k ISECT, THETA1

l AX, AY, AZ, BX, BY, BZ

***BMSECT (6.3.8)**

j IDBLE, IDSEC, IFORM, INTP1, INTP2, INTP3

k VAL1, VAL2, VAL3, VAL4, VAL5, VAL6, VAL7, VAL8

***BODYFORCE, TCRV = n₁, n₂, n₃ (7.3.6)**

\$

j OMEGAX, OMEGAY, OMEGAZ, ACCELX, ACCELY, ACCELZ

\$

k ALPHAX, ALPHAY, ALPHAZ, XO, YO, ZO

***CFLUX (7.4.3)**

j NODE, FLUX, LASTND, INC, IDCRV1, IDCRV2

***CFOLLOWER, TCRV = n (7.3.3)**

j NODE, FORCE, NELID, IDFACE, ITCRV

***CFORCE, TCRV = n (7.3.2)**

\$

k NODE, LABEL, FORCE, LASTND, NODINC, LFN, ITCRV

Quick Reference List

Alphabetical Quick Reference List of All Data Groups

***CONVBC (7.4.7)**

j NELID, LASTEL, INC, IDFACE, KEY, IDCRV1, IDCRV2, IDCRV3
k H1, TEMP1, H2, TEMP2, H3, TEMP3, H4, TEMP4

***CORRELATION (8.6.4)**

\$
j ID, NPRCS, NFN1, ..., NFN12

***CPDISP (6.6.3)**

\$
j LABEL, NSET, NODINC, NODSET, NODE1, NODE2, ..., NODE10

***CPTEMP (6.6.4)**

\$
j LABEL, NEST, NODINC, NODSET, NODE1, NODE2, ..., NODE10

***CREEP (6.5.8)**

j LABEL, MATID, NSIG, NTIME, NCON1, NCON2
\$
k LABEL, MATID, TEMPID, VAL1, VAL2, VAL3, VAL4, VAL5

***DAMPING (8.5.2)**

j IDAMP, DAMP1, DAMP2

***DCFORCE (8.7.3)**

\$

j NODE, LABEL, ID1, ID2, LAST, INCR, FORCE

***DFLUX (7.4.4)**

j NELID, LASTEL, INC, IDFACE, KEY, IDCRV1, IDCRV2

k P1, P2, P3, P4, P5, P6, P7, P8

***DPRESSURE (8.7.4)**

\$

j KELL, ID1, ID2, KFC, KY, LAST, INCR, PFU, CDLOW, CDUPP

k P1, P2, P3, P4, P5, P6, P7, P8

***DRIVER (8.7.5)**

j NODE, LABEL, ID

***EIGCNTL (7.1.2)**

j NNF, NEGK, MAXIT, NPA, COEFLO, COEFUP, RTOL, SHIFT1

***EIGOUT, ID = n (7.5.1)**

\$

j KELFR, KRCTN, KSTR, KSTN, LQ1, LQ2, LQ7

Quick Reference List

Alphabetical Quick Reference List of All Data Groups

*ELEMENTS (6.3.2)

\$

j NELID, MATID, NSRL, IDRC, KISO, NSET, NODINC, NELSET,
NELINC

k NODE1, NODE2, NODE3,..., NODE10

*ELHEATGEN (7.4.5)

\$

j NELID, HTGR, LASTEL, INC, IDCRV1, IDCRV2

*ELTYPE (6.3.1)

j NSRL, NKTP, NORDR, NG1, NG2, NG3

*ENDDATA (7.6.1), (8.9.1)

*EVENT, ID = n (7.1.3)

INCREMENTS = $\begin{bmatrix} \text{EQUAL, n} \\ \text{USER, n} \\ \text{AUTO, n} \\ \text{ARCL, n} \\ \{\text{EQUAL, 10}\} \end{bmatrix}$

TIMEATEND = $\begin{bmatrix} t \\ \{1, 0\} \end{bmatrix}$

NEWTON RAPHSON	=	$\begin{bmatrix} \text{MODI, n} \\ \{\text{FULL, 1}\} \end{bmatrix}$
MAXITERATIONS	=	$\begin{bmatrix} \text{n} \\ \{\text{6}\} \end{bmatrix}$
TOLERANCES	=	$\begin{bmatrix} \text{dto1, eto1, fto1} \\ \{\text{0.001, 0.001, 0.001}\} \end{bmatrix}$
DELTATIME	=	$\begin{bmatrix} \text{dt, opt} \\ \{\text{no default}\} \end{bmatrix}$
STEPLength	=	$\begin{bmatrix} \text{min, max} \\ \{\text{1E-4, 1E-4}\} \end{bmatrix}$
EQUILIBRIUM	=	$\begin{bmatrix} \text{OFF} \\ \text{ON, n} \\ \{\text{ON, 1}\} \end{bmatrix}$
TSFRE	=	$\begin{bmatrix} \text{tval} \\ \{\text{0, 0}\} \end{bmatrix}$
LINE SEARCH	=	$\begin{bmatrix} \text{ON, n} \\ \{\text{OFF}\} \end{bmatrix}$
CREEP	=	$\begin{bmatrix} \text{ON, n1, n2, n3} \\ \{\text{OFF}\} \end{bmatrix}$
ROTFORCE	=	$\begin{bmatrix} \text{icentr, icorio, itange} \\ \{\text{ON, ON, ON}\} \end{bmatrix}$

Quick Reference List

Alphabetical Quick Reference List of All Data Groups

TINTEGRATION = $\left[\begin{array}{l} \text{NEWMARK, beta, gama} \\ \{\text{NEWMARK, 0.3025, 0.6}\} \end{array} \right]$

HYPERELASTIC = $\left[\begin{array}{l} m, n \\ \{0, 0\} \end{array} \right]$

CHARCLENGTH = $\left[\begin{array}{l} \text{ratio} \\ \{\text{no default}\} \end{array} \right]$

*FRCNTL (7.2.1)

j NFOSET, MXFCN, MXFSN, NANGLE

*FREQCNTL (7.1.4)

j FREQ1, FREQ2, NINCR

*FREQFUNCTION (6.7.10)

j TITLE

\$

k ID, NP, NTABLE, SFTR, START, DATA1, DATA2

for NTABLE = 1

l FREQ, VAL

for NTABLE = 2 or 3

m FREQ, RVAL, IVAL

*FRCOEF (7.2.2)

j LABEL, LFN, CONST

k LABEL, n, COEFn, n, COEFn, n, COEFn, n, COEFn

***GROUND (8.7.2)**

\$

j LABEL, ID1, ID2, SFTR/XC, YC, ZC

***HEATCNTL, ID = n (7.1.5)**

j MAXITE, IREFM, NTRFM, RTOL

***HISTORY (8.8.2)**

\$

j TIME1, TIME2, NOPI, IOPT, ID1, ..., ID10

***HISTOUT (7.5.9)**

j LABEL, IDSET, TSTART, TEND, NFRQ, IOPT

***HYPEREL (6.5.5)**

j LABEL, MATID, NPOTN, NORDR1, NORDR2

\$

k LABEL, MATID, TEMPID, VAL1, VAL2, VAL3, VAL4, VAL5

***INITEMP (7.4.1)**

j NODE, TEMP, LASTND, NODINC

***INITIAL (7.3.9), (8.5.5)**

j NODE, LABEL, VALUE, LASTND, INC

Quick Reference List

Alphabetical Quick Reference List of All Data Groups

***LAMANGLE (6.3.5)**

j IDRT, NUMRT
k RA1, RA2, RA3, RA4, RA5, RA6, RA7, RA8

***LAMSQ2 (6.3.6)**

j NLAY, IDRA, I1, I2,... I18
k LS1,LS2,LS3,LS4,LS5,LS6,LS7,LS8,LS9,LS10

***LCSYSTEM (6.4.1)**

\$

IDSYS, ITYPE, METHOD, Variable depends on the value of 'METHOD'

***LCTITLE (7.1.7)**

load case title

***LDCASE, ID = n (7.1.1)**

\$

j KELFR, KRCTN, KSTR, KSTN, LQ1, LQ2, LQ7, TSFRE, TOL

***LDCOMB, ID = n (7.1.6)**

j IDISP, ISTRS, IFORCE
k LCASE1, FACT1, LCASE2, FACT2, LCASE3, FACT3, LCASE4,
FACT4

***MATDIR1 (6.5.2)**

\$

j NODE, INDEX, NSET, THETAX, THETAY, THETAZ

***MATDIR2 (6.5.3)**

\$

j NELID, INDEX, NSET, THETAX, THETAY, THETAZ, ISXY

***MATERIAL (6.5.1)**

\$

j LABEL, MATID, KTEMP, COEF0, COEF1, COEF2, COEF3, COEF4

***MATHEAT (6.5.4)**

\$

j LABEL, MATID, KTEMP, IDCURV, COEF0, COEF1, COEF2,
COEF3, COEF4, IPCHNG

***MDSPECTRUM (8.6.6)**

j IDMDS, ISPEC, DAMP

***MLOAD (6.9.1)**

j IDMVLD, IDPATH, LABEL, NF, NTABLE, NSTART, NEND, INC
for NTABLE = 0

k P1, F1, P2, F2, P3, F3

\$

j NODE, HTGR, LASTND, NODINC, IDCRV1, IDCRV2

***NDTEMPDIF**, TCRV = n, FN = file name, STEP = m (7.3.8)

j NODE, LABEL, DELTAT, LASTND, NODINC, ITCRV

***NDTEMPER**, TCRV = n, FN = file name, STEP = m (7.3.7)

j NODE, LABEL, TVAL, LASTND, NODINC, ITCRV

***NLOUT** (7.5.2)

j KRCTN, KSTR, LQ1, LQ2, LQ7, IFREQ, ISTYP, IRSFR

***NLSPRING** (6.3.9)

j IDSP, NCURV, ITFIT, IRFLCT, ICONS, IPOLTY

k TEMP, NPAIR, X1, Y1, X2, Y2, X3, Y3

***NODES** (6.4.2)

\$ \$

j NODE, IDCSYS, INDEX, NSET, X, Y, Z, IDDSYS

***NONSTATIONARY** (8.6.7)

j TITLE

k ID, NP, NTABLE, SFTR, DATA1, DATA2

l for NTABLE = 2

Quick Reference List

Alphabetical Quick Reference List of All Data Groups

T1, E1, T2, E2, T3, E3

m for NTABLE = 3

E1, E2, E3, E4, E5, E6

***PCHANGE1 (6.7.5)**

j T1, H1, T2, H2, T3, H3, T4, H4

***PDAMPING (6.5.9)**

j NELID, DAMP1, DAMP2, LASTEL, INC

***PLASTIC (6.5.6)**

j LABEL, MATID, NTYSSC, NYCRIT

\$

k LABEL, MATID, TEMPID, VAL1, VAL2, VAL3, VAL4, VAL5

***POSTCNTL (7.5.12)**

j LABEL, I1

***PRESSURE, TCRV = n (7.3.4)**

\$

j NELID, LASTEL, INC, IDFACE, NCARDS, LFN, UPRES, ITCRV

k P1, P2, P3, P4, P5, P6, P7, P8

***PRINCNTL (7.5.3)**

\$

j LABEL, I1, I2, I3, I4, I5, I6, I7, I8, I9

***PSDFUNCTION (8.6.3)**

j TITLE
\$
k ID, NP, NTABLE, SFTR, DATA1, DATA2
l F1, P1, F2, P2, F3, P3

***PSDOUT (8.8.4)**

\$
IDNO, NOPI, IOPT, ID1,..., ID12

***RADBC, SIGMA = val (7.4.8)**

j NELID, LASTEL, INC, IDFACE, KEY, IDCRV1, IDCRV2, IDCRV3
k E1, TEMP1, E2, TEMP2, E3, TEMP3, E4, TEMP4

***RCTABLE (6.3.4)**

j IDRC, NUMRC, IFORM, IDCRV
k RC1, RC2, RC3, RC4, RC5, RC6, RC7, RC8
or
l RC1, RC2, RC3, RC4

***REGIONS (7.5.4)**

j XMIN, XMAX, YMIN, YMAX, ZMIN, ZMAX

***RESPONSE (8.8.7)**

Quick Reference List

Alphabetical Quick Reference List of All Data Groups

\$

j METHOD, IOPT, LABEL, LAYR

***RIGLINK (6.6.1)**

\$

j IDRE, NODM, NSLV, NSET, INC, ND1, K1, ND2, K2, ..., ND5, K5

***RMSOUT (8.8.5)**

\$

j IDNO, IOPT, ID1, ..., ID12

***RSET (8.8.1)**

\$

① ID, NDEL, LABEL, LAST, INCR, LAYR

***SETS (6.7.1)**

\$ \$

j IDSET, LABEL, I1, J1, K1, I2, J2, K2, I3, J3, K3

***SFDCOMP (7.5.6)**

j SRATIO, ILSS, NSRL

***SNAPSHOT (8.8.3)**

\$

j TIME, IDPEAK, IOPT, ID1, ..., ID12

Quick Reference List

Alphabetical Quick Reference List of All Data Groups

***TBODYFORCE (7.3.10)**

j LABEL, VALUE, ITCURV

***TEMPFN (6.7.3)**

j IDCURV, NPOINT

k T1, P1, T2, P2, T3, P3, T4, P4

***TEMPHISTORY (7.5.7)**

j NT, NODE1, NODE2, NODE3, ..., NODE10

***TEMPOUT (7.5.8)**

j TSTART, TEND, TINC

***TIMEAMP (6.7.4)**

j IDCURV, NPOINT, IFORM

k T1, A1, T2, A2, T3, A3, T4, A4

***TIMEFUNCTION (8.6.2)**

j TITLE

\$

k ID, NP, NTABLE, SFTR, START, DELT
for NTABLE = 0

l T1, F1, T2, F2, T3, F3
for NTABLE = 1

m F1, F2, F3, F4, F5, F6

***TIMEINTEG (7.1.9)**

j ALPHA, DTO, TMAX, DTINC, DTMAX, DTDEC, NTDEC

***TITLE (6.2.1), (8.4.1)**

j Problem title

***TSTEP (8.5.3)**

① TIME1, TIME2, DELT

***VECTORS (6.7.2)**

j IDVEC, XCOMP, YCOMP, ZCOMP

D

User's Subroutines

D.1 Introduction

User's subroutines are a programmable feature to allow users to provide certain calculations which are not available in the standard features of NISA. For example, in creep analysis, different materials have their own creep laws. The creep laws offered by NISA may not cover the creep laws needed for certain materials. Users need to program the creep functions for those materials in order to do the analysis.

In order to let the users produce a custom version NISA, NISA provides project templates (..\Nisa2_Userdll) to generate Dynamically Linkable Libraries (DLL), which are written in FORTRAN and compiled with Intel Fortran Compiler (10.1). However user can build his DLLs in any programming environment. Users should have knowledge about how to write, compile and build DLL on their system. These user-created DLLs should replace the default DLLs provided with the standard installation in ALLMEG folder.

D.2 User's Subroutine USRCP

A subroutine USRCP in the directory src.usr is provided to generate USRCP.DLL to have a regular version NISA executable. To obtain a custom version NISA, users should modify subroutine USRCP provided in src.usr to generate customized USRCP.DLL and replace the default DLL in ALLMeg folder. Additional subroutines should be written and added to the DLL project for the calculation of users' creep laws.

The main purpose of the subroutine USRCP is to pass data to and obtain results from users' subroutines. Users can implement all calculation in this subroutine or use as many subroutines as it needs to perform the required calculations.

This subroutine is called in creep analysis to obtain value of creep strain functions at each Gauss point for user defined creep laws. This subroutine has four arrays (RINFO, INFO, ROUT, IOUT) in the argument list. The size of dimension of these arrays is 20. NISA passes data to USRCP through RINFO and INFO and expecting results from ROUT and IOUT. RINFO and ROUT are double precision arrays and INFO and IOUT are integer arrays (INTEGER *4). Not all 20 components of these arrays are used at the present time. Those components used are as follows:

(i) Input arrays:

INFO(1) = Material ID (value of MATID in *CREEP data group)

INFO(2) = User's element ID

RINFO(1) = Effective stress at the current time

RINFO(2) = Current time

RINFO(3) = Current temperature

RINFO(4) = Effective stress at the last converged step

RINFO(5) = Time of the last converged step

RINFO(6) = Young's modules

RINFO(7) = Poisson's ratio

RINFO(8) = Stress Sxx

RINFO(9) = Stress Syy

RINFO(10) = Stress Szz

RINFO(11) = Stress Sxy

RINFO(12) = Stress Syz

RINFO(13) = Stress Sxz

(ii) Output arrays:

ROUT(1) = F1(σ) (stress function)

ROUT(2) = F2(t) (time function)

ROUT(3) = F3(T) (temperature function)

ROUT(4) = G1(σ) (stress derivative of stress function)

ROUT(5) = G2(t) (time derivative of time function)

ROUT(6) = G3(T) (temperature derivative of temperature function)

These six values in ROUT are always required in order to calculate creep strain increments and the size of time steps.

D.3 User's Subroutine USRPLS

D.3.1 Introduction

A subroutine USRPLS in the directory src.usr is provided to generate USRPLS.DLL to have a regular version NISA executable. To obtain a custom version NISA, users should modify subroutine USRPLS provided in src.usr to generate customized USRPLS.DLL and replace the default DLL in ALLMeg folder. Additional subroutines should be written and added to the DLL project for the calculation of users' plasticity laws.

The main purpose of the subroutine USRPLS is to pass data to and obtain results from users' subroutines. Users can implement all calculation in this subroutine or use as many subroutines as it needs to perform the required calculations.

Subroutine USRPLS serves as an interface that links a user-defined elasto-plastic material model to NISA mainframe. With the interface, a user may utilize the finite element data generated from NISA, formulate a specific plasticity law, employ user's own numerical methods and incorporate all these into NISA. To this end, three arrays, RINFO, RPROP and INFO, which contain the stress-strain history, material constants and other programming flags are provided in USRPLS to facilitate computation and analysis history recording.

To activate USRPLS for a specific material ID, both NTYSSC and NYCRIT in *PLASTIC card (see NISA User's Manual, Section 6.5.6) must be set to -1. Ten additional spaces (led by keywords USR1 and USR2) are available to place material constants. Therefore, for each user-defined material type, up to 50 material constants and 10 temperature increments may be specified in a NISA input file. See the listings of RPROP below.

To avoid confusion, all stresses and back stresses are uniformly formatted in the standard 6-component unit, regardless of how many non-zero stress components there actually exist. The order for the components is 11,22,33,12,23,13. The material D-matrix is formatted in the standard 6x6 matrix form and, for convenience, is put in the one-dimensional array RINFO(45:80). Below is the listing of the three arrays, RINFO, RPROP and INFO. An asterisk * indicates that the quantities are required to be calculated and reassigned if plastic deformation occurs, see Section D.3.2. Entries in brackets represent their default value (if applicable); (1:100) means entries from the 1st to 100th in an array.

- RINFO(1:100)** : Stress history, material matrix and plasticity parameters and their history.
- RINFO(1:6) : Total stresses at the beginning of the current step, σ_n
- RINFO(7:12) : Into USRPLS: ELastic predicted incremental stresses for the current step, $[\Delta\sigma^e]$
Out USRPLS: Corrected incremental stresses for the current step, $[\Delta\sigma^e]$
- RINFO(13:18) : Total back stresses (for mixed or kinematic hardening material) at the beginning of the current step,
- *RINFO(19:24) : Incremental back stresses for the current step (for kinematic/mixed hardening only), $\Delta\alpha$, [0]
- RINFO(25:34) : Plasticity parameters at the beginning of the current step
- RINFO(25) : Total effective plastic strain at the beginning of the current step, an ε_n^p
- RINFO(26) : Yield stress at the beginning of the current step, on σ_n^y
- RINFO(27) : Total effective stress at the beginning of the current step, on σ_n^e
- RINFO(28) : Plastic deformation indicator of the previous step, [0]
= 0, no plastic deformation occurred
= 1, plastic deformation occurred
- RINFO(29) : User defined internal variable #1 (Optional) at the beginning of the current step, IV_1 , [0]
- RINFO(30) : User defined internal variable #2 (Optional) at the beginning of the current step, IV_2 , [0]
- RINFO(31) : User defined internal variable #3 (Optional) at the beginning of the current step, IV_3 , [0]

User's Subroutines

User's Subroutine USRPLS

- RINFO(32) : User defined internal variable #4 (Optional) at the beginning of the current step, IV_4 , [0]
- RINFO(33) : User defined internal variable #5 (Optional) at the beginning of the current step, IV_5 , [0]
- RINFO(34) : User defined internal variable #6 (Optional) at the beginning of the current step, IV_6 , [0]
- *RINFO(35:44) : Plasticity parameters at the end of current iteration
- *RINFO(35) : Total effective plastic strain at the end of the current iteration, ϵ_p^n
- *RINFO(36) : Yield stress at the end of the current iteration, σ_y^n
- *RINFO(37) : Total effective stress at the end of the current iteration
- *RINFO(38) : Plastic deformation indicator of the current step, [0]
= 0, no plastic deformation occurred
= 1, plastic deformation occurred
- *RINFO(39) : User defined internal variable #1 (Optional) at the end of the current iteration, [IV_1]
- *RINFO(40) : User defined internal variable #2 (Optional) at the end of the current iteration, [IV_2]
- *RINFO(41) : User defined internal variable #3 (Optional) at the end of the current iteration, [IV_3]
- *RINFO(42) : User defined internal variable #4 (Optional) at the end of the current iteration, [IV_4]
- *RINFO(43) : User defined internal variable #5 (Optional) at the end of the current iteration, [IV_5]
- *RINFO(44) : User defined internal variable #6 (Optional) at the end of the current iteration, [IV_6]
- *RINFO(45:80) : Into USRPLS: Elastic D-matrix (6x6), [D^e]
Out USRPLS: Elastic/Elasto-plastic D-matrix (6x6), [D^e]

RINFO(81) : Current temperature, T_{n+1}
RINFO(82) : Stress-free temperature, T_0
RINFO(83) : Young's modulus, E
RINFO(84) : Poisson's ratio, μ
RINFO(85:100) : not assigned
RPROP(1:60) : Element material constants, as defined in *PLAS card.
See NISA User's Manual Section 6.4.6 for the input format, all defaulted to 0.

RPROP(1) : TEMPID value for SY0
RPROP(2) : Value #1 for SY0
RPROP(3) : Value #2 for SY0
RPROP(4) : Value #3 for SY0
RPROP(5) : Value #4 for SY0
RPROP(6) : Value #5 for SY0

RPROP(7) : TEMPID value for HARD
RPROP(8) : Value #1 for HARD
RPROP(9) : Value #2 for HARD
RPROP(10) : Value #3 for HARD
RPROP(11) : Value #4 for HARD
RPROP(12) : Value #5 for HARD

RPROP(13) : TEMPID value for SIG7
RPROP(14) : Value #1 for SIG7
RPROP(15) : Value #2 for SIG7

User's Subroutines

User's Subroutine USRPLS

- RPROP(16) : Value #3 for SIG7
- RPROP(17) : Value #4 for SIG7
- RPROP(18) : Value #5 for SIG7

- RPROP(19) : TEMPID value for BETA
- RPROP(20) : Value #1 for BETA
- RPROP(21) : Value #2 for BETA
- RPROP(22) : Value #3 for BETA
- RPROP(23) : Value #4 for BETA
- RPROP(24) : Value #5 for BETA

- RPROP(25) : TEMPID value for EXPO
- RPROP(26) : Value #1 for EXPO
- RPROP(27) : Value #2 for EXPO
- RPROP(28) : Value #3 for EXPO
- RPROP(29) : Value #4 for EXPO
- RPROP(30) : Value #5 for EXPO

- RPROP(31) : TEMPID value for STRS
- RPROP(32) : Value #1 for STRS
- RPROP(33) : Value #2 for STRS
- RPROP(34) : Value #3 for STRS
- RPROP(35) : Value #4 for STRS
- RPROP(36) : Value #5 for STRS

- RPROP(37) : TEMPID value for STRN
- RPROP(38) : Value #1 for STRN

RPROP(39) : Value #2 for STRN
RPROP(40) : Value #3 for STRN
RPROP(41) : Value #4 for STRN
RPROP(42) : Value #5 for STRN

RPROP(43) : TEMPID value for FRIC
RPROP(44) : Value #1 for FRIC
RPROP(45) : Value #2 for FRIC
RPROP(46) : Value #3 for FRIC
RPROP(47) : Value #4 for FRIC
RPROP(48) : Value #5 for FRIC

RPROP(49) : TEMPID value for USR1
RPROP(50) : Value #1 for USR1
RPROP(51) : Value #2 for USR1
RPROP(52) : Value #3 for USR1
RPROP(53) : Value #4 for USR1
RPROP(54) : Value #5 for USR1

RPROP(55) : TEMPID value for USR2
RPROP(56) : Value #1 for USR2
RPROP(57) : Value #2 for USR2
RPROP(58) : Value #3 for USR2
RPROP(59) : Value #4 for USR2
RPROP(60) : Value #5 for USR2

INFO(1:20) : Integer information from NISA

INFO(1)	:	NKTP, element type ID
INFO(2)	:	not assigned
INFO(3)	:	Event ID number
INFO(4)	:	Step number of the current event
INFO(5)	:	Iteration number of the current step
INFO(6)	:	Unit number for writing out to screen
INFO(7)	:	Unit number for writing out to NISA out
INFO(8)	:	Analysis flag. = 0, calculate incremental stresses and plasticit = 1, calculate elasto-plastic D-matrix = 2, both above
INFO(9)	:	Non-zero number of stress components
INFO(10)	:	Non-zero number of rows and columns of
INFO(11)	:	Number of plasticity parameters other The effective plastic strain, yield stress, etc.
INFO(12)	:	Element ID
INFO (13)	:	Gauss point number
INFO (14)	:	NYLDAT
INFO (15)	:	MVWRD No. of real variable for each n
INFO (16)	:	NTYSSC Flag for selection of stress-s
INFO (17)	:	KNLTY analysis type
INFO (18:20)	:	not assigned

D.3.2 Modeling Tips

1. INFO(8), the analysis flag, is preset by NISA. Therefore, a user must direct the program to function accordingly. That is, when INFO(8)=0, only stress integration should

be performed; when INFO(8)=1, only the elasto-plastic material matrix should be formed; and when INFO(8)=2, both should be provided.

2. At the first iteration of the first step of analysis, the user should initialize the plasticity parameters with the material constants given by RPROP. For example,

RINFO(25) = 0 (Initial effective plastic strain)

RINFO(26) = SY0 (or initial yield stress)

RINFO(27) = 0 (Initial effective stress)

RINFO(28) = 0 (Assuming elastic initially)

When existing USRPLS, the user should at least provide information for these entries:

RINFO(35) = Current effective plastic strain or

RINFO(36) = Current yield stress or RINFO(26)

RINFO(37) = Current effective stress

RINFO(38) = 1 if plastic, or = 0 otherwise

and

RINFO(45) to RINFO(80) for Elasto-Plastic D-matrix or Elastic D-matrix.

1. The elastic trial stresses are obtained by adding the elastic incremental stresses RINFO(7:12) to RINFO(1:6) when stress calculation is requested; while the total current stress are obtained by doing so when forming elastic-plastic D-matrix is requested.
2. By assigning any desirable quantities (up to six) to RINFO(39:44), the user may print them out at the end of step in the NISA output file whenever element stress print-out is opted (See *NLOUT in Section 7.5.2). Also RINFO(39:44) is assigned to RINFO(29:34) as a part of history record at the first iteration of each step such that the user may utilize them to calculate new RINFO(39:44).
3. The user-defined material model is only applicable to NKTP=1,2,3,4,20,39 and 44.

D.4 User's Subroutine USRFAL

D.4.1 Introduction

A subroutine USRFAL in the directory src.usr is provided to generate USRFAL.DLL to have a regular version NISA executable. To obtain a custom version NISA, users should modify subroutine USRFAL provided in src.usr to generate customized USRFAL.DLL and replace the default DLL in ALLMeg folder. Additional subroutines should be written and added to the DLL project for the calculation of users' failure criteria.

The main purpose of the subroutine USRFAL is to pass data to and obtain results from users' subroutines. Users can implement all calculation in this subroutine or use as many subroutines as it needs to perform the required calculations.

To provide users the flexibility to obtain failure criterion other than those provided by NISA for layer composite elements such as element NKTP=7, 32, and 33. A user-defined failure criterion feature has been added to allow users to use their own failure criterion model.

Subroutine USRFAL serves as an interface between NISA program and the calculation of user-defined failure criterion. This subroutine is called by NISA unconditionally during stress calculation to calculate Tsai-Hill criterion at Gauss points as well as at nodal points. Users can program their failure criterion calculation to replace the existing calculation of Tsai-Hill criterion in the subroutine.

Information such as element ID, stress components, and material properties are passed through three arrays, INFO, RINFO1, and RINFO2 to the subroutine and NISA expects the value of failure criterion and the name of criterion from variables ROUT and COUT as output from the calculation. The stress components given to this subroutine are in the material principal directions. The name of criterion (COUT) will be used as the title for the user-defined criterion in the output file. All real variables or arrays should be in double precision.

The following is the list of the details of input and output arrays:

Input arrays

1. Integer array INFO

INFO(1) = Element ID

INFO(2) = Element type (NKTP value)

2. Real array RINFO1

RINFO1(1) = SXX

RINFO1(2) = SYX

RINFO1(3) = SZZ

RINFO1(4) = SXY

RINFO1(5) = SYZ

RINFO1(6) = SXZ

3. Real array RINFO2

RINFO2(1) = EX

RINFO2(2) = EY

RINFO2(3) = EZ

RINFO2(4) = NUXY

RINFO2(5) = NUXZ

RINFO2(6) = NUYZ

RINFO2(7) = ALPX

RINFO2(8) = ALPY

RINFO2(9) = ALPZ

RINFO2(10) = DENS

RINFO2(11) = GXY

RINFO2(12) = GYZ

User's Subroutines

User's Subroutine USRFAL

RINFO2(13)	=	GXZ
RINFO2(14)	=	FXC
RINFO2(15)	=	FXT
RINFO2(16)	=	FYC
RINFO2(17)	=	FYT
RINFO2(18)	=	FS
RINFO2(19)	=	F12
RINFO2(20)	=	FZC
RINFO2(21)	=	FZT
RINFO2(22)	=	F13
RINFO2(23)	=	F23
RINFO2(24)	=	FSXZ
RINFO2(25)	=	FSYZ

Output arrays

1. Real variable ROUT

ROUT = value of failure criterion

2. Character variable COUT

COUT = Name of the criterion (Character*10)

E

References

Chapter 2

1. K. Washizu, *Variational Methods in Elasticity and Plasticity*, 3rd Edition, Pergamon Press, 1982.
2. C. Truesdell, *First Course in Rational Continuum Mechanics - Vol I: General Concepts*, Academic Press, New York, 1977.
3. J.T. Oden, *Finite Elements of Nonlinear Continua*, McGraw-Hill Book Co., New York, 1972.
4. J.T. Oden and E.R.A. Oliveira (Editor), *Lectures on Finite Element Methods in Continuum Mechanics*, The University of Alabama in Huntsville, 1973.
5. K.J. Bathe, *Finite Element Procedures in Engineering Analysis*, Prentice Hall Inc., Eaglewood Cliffs, New Jersey, 1982.
6. M.S. Gadala and G.A. Oravas, *Numerical Solutions of Nonlinear Problems of Continua - I; Survey of formulation methods and solution techniques*, Computers & Structures, 19(5), 865-877, 1984.
7. M.S. Gadala and G.A. Oravas, *Formulation methods of geometric and material nonlinearity problems*, International Journal for Numerical Methods in Engineering, 20, 887-914, 1984.
8. R. Hill, *On constitutive inequalities for simple materials - I*, Journal of the Mechanics and Physics of Solids, 16, 229-242, 1968.
9. M.S. Gadala, G.A. Oravas and M.A. Dokainish, *A consistent Eulerian formulation of large strain static and dynamic problems*, International Journal of Non-Linear Mechanics, 18(1), 21-35, 1983.

References

10. M.S. Gadala and M. Abo-Elkhier, *A comparison of finite element formulations in non-linear structural mechanics*, Res Mechanica, 18(2), 157-177, 1986.
11. Meirovitch, *Analytical Methods in Vibrations*, The MacMillan Company, New York, 1967.
12. W.C. Hurty and M.F. Rubinstein, *Dynamics of Structures*, Prentice Hall Inc., New Jersey, 1972.
13. R.W. Clough and J. Penzien, *Dynamics of Structures*, McGraw-Hill Book Co., New York, 1975.
14. E. Hinton, T. Rock and O.C. Zienkiewicz, *A Note on Mass Lumping and Related Processes in the Finite Element Method*, Earthquake Engineering and Structural Dynamics, 4, 245-249, 1976.
15. K.S. Surana, *Lumped Mass Matrices with Non-Zero Inertia for General Shell and Axisymmetric Shell Elements*, International Journal for Numerical Methods in Engineering, 12, 1635-1650, 1978.
16. L.E. Malvern, *Introduction to the mechanics of continuous medium*, Prentice Hall Inc., Englewood Cliffs, New Jersey, 1969.
17. R.M. Barker, F.T. Lin and J.R. Dona, *Three Dimensional finite element analysis of laminated composites*, Computers & Structures, 2, 1013-1029, 1972.
18. R. Hill, *The mathematical theory of plasticity*, Oxford University Press, London - U.K., 1971.
19. O.C. Zienkiewicz, *The finite element method*, McGraw-Hill Book Co., London, 1977.
20. W. Prager, *An introduction to plasticity*, Addison - Wesley, Amsterdam and London, 1959.
21. W. Prager, *A new method of analyzing stresses and strains in work-hardening plastic solids*, Journal of Applied Mechanics, 23, 493-496, 1956.
22. H. Ziegler, *A modification of Prager's hardening rule*, Quarterly of Applied Mathematics, 17, 55-65, 1959.
23. M. Ortiz and E.P. Popov, *Accuracy and stability of integration algorithms for elastoplastic constitutive relations*, International Journal for Numerical Methods in Engineering, 21, 1561-1576, 1985.
24. M.S. Gadala and M.S. Hamid, *On the solution of viscoplastic and creep problems*, Proceedings of the 3rd International Conference on Constitutive Laws, January 1991, ASME Press, 701-705.
25. C. Truesdell and W. Noll, *The nonlinear field theories of mechanics*, In Encyclopedia of Physics, Edited by S. Fliigge, 313, Springer - Verlag, Berlin, 1965.

-
26. M. Mooney, *A theory of large elastic deformation*, Journal of Applied Physics, 11, 582-597, 1940.
 27. R.S. Rivlin, *Large elastic deformations of isotropic materials, IVL: Further developments of the general theory*, Philosophical Transactions of the Royal Society of London, A241, 379-397, 1948.
 28. L.R.G. Treloar, *The physics of Rubber Elasticity*, Oxford University Press, 1958.
 29. R.S. Rivlin, *Large Elastic Deformations of Isotropic Materials. I. Fundamental Concepts*, Philosophical Transactions of the Royal Society of London, A240, 459-490, 1948.
 30. J.T. Oden, *Finite Elements of Nonlinear Continua*, McGraw-Hill Book Co., New York, 1972.
 31. S.R. Swanson, *A constitutive model for High Elongation Elastic Materials*, Journal of Engineering Materials and Technology, ASME Transactions, 107, 110-114, 1985.
 32. P.J. Blatz and W.L. KO, *Applications of Finite Element Theory to the Deformation of Rubbery Materials*, Trans. Soc. Rheol., 6, 223-251, 1962.
 33. A. Isihara, N. Hashitume and M. Tatibana, *Statistical Theory of Rubber-like Elasticity, IV: Two dimensional Stretching*, Journal of Chemical Physics, 19, 1508-1512, 1951.
 34. V.L. Biderman, *Calculation of Rubber Parts*, (in Russian), Rascheti na Prochnost, Moscow, 1958.
 35. J. Klosner and A. Segal, *Mechanical Characterization of Natural Rubber*, PIBAL Report, No. 69-42, Polytechnic Institute of Brooklyn, New York, 1969.
 36. L.J. Hart-Smith, *Elasticity Parameters for Finite Deformations of Rubber-like Materials*, Zietschrift fur Angewandte Mathematik und Physik, 17, 608-625, 1966.
 37. H. Alexander, *A Constitutive Relations for Rubber-like Materials*, International Journal of Engineering Science, 6, 549-563, 1968.
 38. M.S. Gadala, *The use of bubble functions for the solution of problems with incompressibility*, Proceedings of the 3rd International Conference on Constitutive Laws, January 1991, ASME Press, 101-105.
 39. O.C. Zienkiewicz, R.L. Taylor and J.M.W. Baynham, *Mixed and irreducible formulations infinite element analysis*, Hybrid and Mixed Finite Element Methods, Edited by S.N. Atlar et. al. John-Wiley and Sons, 405-431, 1983.
 40. K.J. Bathe, *Finite element procedures in engineering analysis*, Prentice Hall Inc., Eaglewood Cliffs, New Jersey, 1982.
 41. R.L. Taylor, J.P. Beresford and E.L. Wilson, *A nonconforming element for stress analysis*, International Journal for Numerical Methods in Engineering, 10, 1211-1219, 1976.

References

42. G.R. Cowper, *Gaussian quadrature formulas for triangles*, International Journal for Numerical Methods in Engineering, 7, 405-408, 1973.
43. H. Kardestuncer and D.H. Norrie (Editors), *Finite element handbook*, McGraw-Hill Book Co., New York, 1987.
44. D.E. Newland, *An Introduction to Random Vibration and Spectral Analysis*, Longman Group Ltd., London and New York, 1983.
45. E.L. Wilson, A. Der Kiureghian and E.P. Bayo, *A replacement for the SRSS Method in Seismic Analysis*, Earthquake Engineering and Structural Dynamics, 9, 187-194, 1981.
46. C.H. Norris, J.B. Wilbur and S. Utku, *Elementary Structural Analysis*, 3rd Edition, McGraw-Hill Book Co., New York, 1976.
47. J.L. Meriam, *Dynamics*, 2nd Edition, John Wiley and Sons, New York, 1971.
48. M. El-Essawi, *A Discrete Model for Nonlinear Structural Dynamics of Rotating Cantilevers*, Ph.D. Dissertation, Duke University, Durham, North Carolina, 1982.
49. M. Salaina, M. Trubert, M. El-Essawi and S. Utku, *Second Order Nonlinear Equations of Motion for Spinning Highly Flexible Line-Elements*, Journal of Sound and Vibration, 80, 461-472, 1982.
50. K.K. Gupta, *Formulation of Numerical Procedures for Dynamic Analysis of spinning Structures*, International Journal for Numerical Methods in Engineering, 23, 2347-2357, 1986.
51. M.A.J. Bossak and O.C. Zienkiewicz, *Free Vibration of Initially Stressed Solids with particular Reference to Centrifugal Force Effects in Rotating Machinery*, Journal of Strain Analysis, 8, 245-252, 1973.
52. R.D. Cook, D.S. Malbus and M.E. Plesha, *Concepts and Applications of Finite Element Analysis*, 3rd Edition, John Wiley and Sons, Inc., 1989.
53. A.A. Ilyushin, *Plasticite*, EyroBes, Paris, 1956.
54. K.J. Bathe and E. Dvorkin, *Our discrete - Kirchhoff and Isoparametric Shell Elements for Nonlinear Analysis -An Assessment*, Computers & Structures, 16, 89-98, 1983.
55. T.R. Hsu, *The Finite Element Method in Thermomechanics*, Allen and Urwin, Boston, 1986.
56. M. Mazurkiewicz and W. Ostachowicz, *Theory of Finite Element Method for Elastic Contact of Solid Bodies*, Computers & Structures, 17, 51-59, 1983.
57. M.U. Polat and M.A. Dokainish, *A Finite Element Procedure for the Simulation of Superplastic Sheet Metal Forming Processes*, Computers & Structures, 36, 251-259, 1990.

58. USNRC Regulatory Guide 1.92, February 1976.
59. A.K. Chopra, *Dynamics of Structures: Theory & Application to Earthquake Engineering*. Pearson Education, 2nd Edition, New Delhi, 2001.
60. A. Der Kiureghian and Y. Nakumara, *CQC Modal Combination Rule For High Frequency Modes*, Earthquake Engineering and Structural Dynamics, 22, 943-956, 1993.
61. M Vardharajan, Spectrum compatible PSD for non stationary earthquake excitations, 1992, MS(Engg.), Thesis, Department of Civil Engineering, IISc., Bangalore.
62. ASCE Code 4-98: Seismic Analysis of Safety-Related Nuclear Structures and Commentary.
63. W.F. Chen, *Plasticity in reinforced concrete*, McGraw-Hill Book Co., New York, 1982.
64. C. Channakeshava and K.T. Sundara Raja Iyengar, *Elasto-Plastic Cracking Analysis of Reinforced Concrete*, Journal of Structural Engineering, 114, 2421-2438, 1988.

Chapter 3

1. J.T. Oden, *Finite Elements of Nonlinear Continua*, McGraw-Hill Book Co., New York, 1972.
2. K.J. Bathe, *Finite Element Procedures in Engineering Analysis*, Prentice Hall Inc., Eagle wood Cliffs, New Jersey, 1982.
3. M.S. Gadala, G.E. Dravas and M. A. Dokainish, *Formulation Methods of Geometric and Material Nonlinearity Problems*, International Journal for Numerical Methods in Engineering, 20, 887-914, 1984.
4. M.S. Gadala and M. Abo-Elkhier, *A Comparison of Finite Element Formulations In Nonlinear Structural Mechanics*, Res. Mechanica, 18, 157-177, 1986.
5. R.W. Clough and J. Penzien, *Dynamics of Structures*, McGraw-Hill Book Co., 1975.
6. W.C. Huny and M.F. Rubinstein, *Dynamics of Structures*, Prentice-Hall Inc., Englewood, U.S.A.
7. Y.K. Lin, *Probabilistic Theory of Structural Dynamics*, McGraw-Hill Book Co., 1967.
8. G. Augusti, A. Baratta and F. Casciati, *Probabilistic Methods in Structural Engineering*, Chapman and Hall, London, 1984.
9. O.C. Zienkiewicz, *The Finite Element Method*, McGraw-Hill Book Co., New York, 3rd Edition, 1977.
10. F.P. Incropera and D.P. Dewitt, *Fundamentals of Heat Transfer*, John Wiley and Sons, 1981.
11. H.S. Carslaw and J.C. Jaeger, *Conduction of Heat in Solids*, Oxford University Press, 2nd Edition, 1959.
12. J.P. Holman, *Heat Transfer*, McGraw-Hill Book Co., New York.
13. R. Hill, *The Mathematical Theory of Plasticity*, Oxford University Press, 1950.
14. W. Prager, *An Introduction to Plasticity*, Addison-Wesley, Amsterdam-London, 1959.
15. C.H. Norris, J.B. Wilbur and S. Utku, *Elementary Structural Analysis*, 3rd Edition, McGraw-Hill Book Co., New York, 1976.
16. J.L. Meriam, *Dynamics*, 2nd Edition, John Wiley and Sons, New York, 1971.
17. B.M. Irons, *Roundoff Criteria in Direct Stiffness Solution*, AIAA Journal, 6, 1308-1312, 1968.
18. B.M. Irons, *A Frontal Solution Program for Finite Element Analysis*, International Journal for Numerical Methods in Engineering, 2, 5-32, 1970.

19. E.D.L. Pugh, E. Hinton and O.C. Zienkiewicz, *A Study of Quadrilateral Plate Bending with Reduced Integration*, International Journal for Numerical Methods in Engineering, 12(7), 1059-1079, 1978.
20. C.F. Shih and D. Lee, *Further developments in anisotropic plasticity*, Journal of Engineering Materials and Technology, ASME Transactions, 100, 294-302, 1978.
21. M.A. Crisfield, *Accelerated Solution Techniques and Concrete Cracking*, Computer Methods in Applied Mechanics and Engineering, 33, 585-607, 1982.
22. M.A. Crisfield, *Variable Step - Length for Non Linear Structural Analysis*, Transport and Road Research Laboratory Report 1049, Bridge Division, Structures Department, Crowthome, Berkshire, U.K., 1982.
23. B. Patel, *Thermo-elasto-plastic Finite Element Formulation for Deformation and Residual Stresses due to Welds*, Ph.D Dissertation Carleton University, Ottawa, Canada, 1985.
24. Intel® Math Kernel Library (MKL), Reference Manual, 2007.<http://www.intel.com/software/products/mkl/>

Chapter 4

1. R.M. Jones, *Mechanics of Composite Materials*, McGraw-Hill Book Co., New York, 1975.
2. S.W. Tsai and E.M. Wu, *A General Theory of Strength for Anisotropic Materials*, Journal of Composite Materials, 5, 58-80, 1971.
3. R.B. Pipes and B.W. Cole, *On the Off-Axis Strength Test for Anisotropic Materials*, Journal of Composite Materials, 7, 246-256, 1973.
4. J.T. Oden and E.A. Ripperger, *Mechanics of Elastic Structures*, McGraw-Hill Book Co., New York, 1981.
5. R. Hill, *The Mathematical Theory of Plasticity*, Oxford University Press, London, 1950.
6. R. D. Cook, *Concepts and Applications of finite element analysis*, 2nd Edition, John Wiley and Sons, New York, 1981.
7. ASNE Botler and Pressurr, Vessel Code, Section 111, Subsection NB and NC, 1977.
8. Dodge and Moore, *Empirical factors from the Welding Research Council Bulletin 179*.

Index

I.1 Keywords Index

- ABS modal combination, 2-68, 3-53
Absolute Response, 3-39
Acceleration, 2-14, 2-37, 3-40, 3-52, 7-66, 7-67, 7-123, 8-4, 8-7, 8-48, 8-81, 8-95
Adiabatic, 2-85
Alexander model, 2-132, 3-85
Analysis data, 3-117, 5-1, 7-1, 8-30
Analysis types, 1-6, 3-1, 3-138, 3-142
Angular acceleration, 2-13, 2-144, 2-149, 3-97, 7-66, 7-75
Angular velocity, 2-13, 2-144, 7-66, 7-75
Anisotropic plasticity, 4-11, 6-90
Arc length, 3-22, 7-15
Arrival time, 3-42
Associated plasticity, 2-115
Autocorrelation function, 2-62, 3-45
Average constraint approach, 2-126
Average wavefront, 3-107
Axisymmetric Shell elements, 4-4
Axisymmetric Solid elements, 4-5
Axisymmetric solid with non-axisymmetric loading, 4-6
- Bar elements, 4-5, 4-225, 4-226
Base excitation, 2-46, 3-38
Beam elements, 4-6, 4-12, 4-64, 4-72, 4-160, 6-11, 6-13, 6-14, 6-20
Beam loads, 3-96, 7-63
Biderman model, 2-131, 3-85, 6-81
Blatz-Ko model, 2-130, 3-85
- Body forces, 2-13, 3-10, 3-61, 7-4, 7-67
Boundary conditions, 3-74, 3-76, 3-94
Bubble functions, 2-161, 2-170, 4-17
Buckling analysis, 2-79, 2-148, 4-92
- Cable element, 2-144, 4-174
CAD/CAM Interface, 1-1
Cartesian system, 3-3
Centrifugal acceleration, 7-20
Centrifugal softening, 2-33
Centroidal layer stresses, 8-100
Centroidal stress, 8-99
Check run, 3-100, 3-138
Combination rules, 3-57, 8-14
Commands, 3-117, 8-18, 8-112
Comment cards, 5-5
Composite Elements, 4-40
Concentrated loads, 2-61, 3-96, 4-141, 7-56, 7-63
Concentrated mass elements, 2-143, 4-108
Conduction, 2-81, 3-73
Conductivity matrix, 2-6
Consistent mass matrix, 2-37, 3-28
Constitutive equations, 2-17, 2-18, 2-97
Constraints, 1-4, 3-28
Contact problems, 2-17, 3-15, 4-203
Convection, 2-85, 3-73, 6-24, 7-96
Convergence criteria, 3-18, 7-37
Coordinate systems, 3-3
Coriolis acceleration, 7-20
Coriolis force, 3-63
Correlation, 2-63
Coulomb Friction, 2-108, 4-193

Coupled displacements, 1-3, 1-4, 3-61, 3-90
Coupled temperatures, 3-78
CQC modal combination, 2-69, 3-58
Crack-tip analysis, 2-156
Creep, 3-87, 7-19
Cross PSD, 8-105
Curved Beam elements, 4-228

Damping, 2-44, 3-28
Data
 Checking, 3-100
 deck setup, 5-66
 groups, 3-2, 5-3
Decomposed stiffness matrix, 3-11, 3-136
Deformation gradient, 2-6
Degrees of freedom, 3-7
Direct Frequency Response Analysis, 2-77
Displacement constraints, 3-6, 3-91
DISPLAY program, 3-1, 3-13, 3-100, 3-117, 3-118
Drucker-Prager yield criterion, 2-105, 3-83
Dynamic analysis, 2-53, 3-27, 3-40, 8-1

Eccentricity, 4-6, 4-69
Effective modal mass, 2-49
Eigenvalue
 buckling, 2-79
Eigenvalue
 analysis, 3-27, 3-29
 buckling, 3-69
 extraction methods, 3-69
Eigenvectors, 2-79
Elastic material, 2-97, 3-81
Element
 attributes, 4-10
 coordinate systems, 3-3
 library, 1-2, 3-54, 4-1
 loads, 3-94
End release, 4-64, 4-72
energy, 7-125

Envelope Spectrum Method, 2-72
Equivalent nodal loads, 3-94
Equivalent plastic strain, 2-111
Error messages, 3-100
Executive commands, 5-21
extreme value characteristics, 3-44

Fiber orientation, 4-10, 6-28, 6-32
Field problems, 3-79
File switching, 5-6
Files, 3-136
Flux, 3-76
Follower forces, 2-34, 3-94
Forced vibration, 2-38, 3-36
Fourier series, 4-133, 4-147
Free format, 5-5
Free vibration, 2-54
FREQCNTL, 7-26
FREQFUNCTION, 6-143
FREQUENCY, 3-37, 3-50
Frequency response analysis, 3-50
Friction, 2-188, 4-193, 4-200

Gap elements, 4-8
Generalized coordinates, 2-41
Geometric stiffness matrix, 2-29, 2-148, 3-69
Global coordinate System, 3-3
Gravity loads, 2-13, 3-97, 7-66
Green-Lagrange, 2-20, 3-82
Ground motion, 2-46, 3-38

Hardening rules, 3-87
Hart-Smith model, 3-85, 6-78
Heat transfer analysis, 2-81, 3-73
Heat transfer elements, 4-1
Hexahedron elements, 4-28, 4-36

Hybrid solid element, 4-48
Hyperelastic material, 2-126, 3-85, 6-77

IHT model, 2-131
Ill-conditioning, 3-102
influence matrix, 2-50, 7-105
Initial conditions, 3-40
Initial tension, 4-5
Input data setup, 5-46
Internal forces, 2-10, 3-6, 7-7
Internal heat generation, 2-7, 3-74, 7-92
Inverse Iteration, 3-30
Isoparametric elements, 4-55
Isotropic hardening, 6-92
Isotropic material, 2-105, 3-75

Kinematic constraints, 3-88
Kinematic hardening, 2-113, 3-84
Kinetic energy, 2-5
Klosner-Segel model, 2-131, 3-85, 6-81

Lagrange, 2-20
Laminated
 composite elements, 4-40
 composite shells, 4-110
Lanczos method, 3-30
Large displacements, 2-80, 2-152
Large mass concept, 3-39
Layered composite elements, 4-6
Line element forces, 4-83, 4-87, 4-139
Line search, 3-17, 3-19, 7-19
Load combination, 7-28
Load stepping, 3-16, 3-66
Local coordinate system, 3-3, 3-96
Local displacement system, 3-6
LTRANSIENT, 3-61, 5-22
Lumped mass matrix, 2-37

Mass matrix, 6-102
Material models, 2-1, 3-75
Maximum wavefront, 3-107
Membrane element, 4-45, 4-51
Modal
 combination, 8-111
 covariance matrix, 8-23
 dynamic analysis, 8-1
 mass, 3-28
 superposition, 2-53
Mode
 selection, 3-37
 shapes, 3-29
Model Data, 3-2, 5-53, 6-1
Mohr-Coloumb Yield criterion, 2-108
Moment of inertia, 4-109, 6-1
Mooney-Rivlin model, 2-126, 3-85, 6-79
MSEXITATION, 8-21
Multiple load cases, 3-99
Multipoint constarints, 3-61

Natural frequencies, 1-4, 2-38, 3-52
Neo-Hookean model, 2-128, 3-85, 6-79
Newmark Beta method, 3-62
Newton-Raphson, 3-16
NKTP, 4-1
NLSTATIC, 5-22
Nodal forces, input, 7-56, 8-82
Nodal forces, unbalanced, 3-10, 3-117, 7-7
Nodeless degrees of freedom, 2-157, 4-2
Nonconservative loads, 6-49
Normal mode method, 2-41, 8-1
Normality criterion, 2-115
NRL modal combination, 3-58

Octave band, 3-46
Offsets, 4-55

Orthogonality properties, 2-41
Orthotropic material, 2-102, 3-76
Output control, 7-103, 8-96
Output quantities, 1-8

Participation factors, 2-49, 3-38
PEAK modal combination, 3-27
Pentahedron elements, 4-26
Phase angle, 2-58, 3-52
Pipe elements, 4-177
Plane strain elements, 4-18
Plane stress elements, 2-151
Plasticity, 2-105, 2-111, 6-82
Plate elements, 4-7
Point loads, 7-14
Point mass elements, 2-34
Poisson's equation, 2-81
Poisson's ratio, 3-102, 3-146
Positive definite, 2-37
Post data, 7-5
Postprocessing, 1-7, 3-118
Potential energy, 2-9
Power spectral density, 2-62
Pressure loads, 2-61, 3-95, 7-60, 8-85
Prestored response Spectra, 8-5, 8-15
Prestress restart, 3-138
Principal material directions, 3-86, 6-93
PRMS modal combination, 2-69, 3-53
Probability density function, 2-62, 3-27
Proportional damping, 2-44, 2-45
PSD, 2-62, 3-126

Quadrilateral elements, 4-17, 4-232

R.M.S. response, 2-62, 2-67, 8-1
Radiation, 2-86, 3-73, 7-99

Random vibration analysis, 2-38, 3-126, 8-10
Rayleigh damping, 2-44
Real constants, 4-234, 6-1
Reference configuration, 2-16, 3-14, 5-35
Reliability estimation, 3-48
Response spectrum, 2-67, 3-52
Restart capabilities, 3-138
Resultant forces and moments, 4-56, 4-158, 4-179, 4-184
Rigid body modes, 2-34, 3-31
Rigid element forces, 7-29
Root Mean square value, 8-106
Rotating structures, 2-33, 3-71
Round-off errors, 3-103
Rubber-like material, 2-123, 6-77

Sandwich shell elements, 4-174
Saving files, 5-22
Shape Factor, 8-106
shape factor, 3-48
Shape functions, 2-144, 2-152, 2-166, 2-172
Shear center, 4-55
Shear modulus, 2-97
SHEAT, 3-73
SHOCK, 5-60, 8-1
Shock spectrum analysis, 8-13
Sign convention, 3-3
Singularities, 3-101
Solid elements
 structural, 2-D, 4-166
 structural, 3-D, 4-217
solid elements
 thermal, 3-D, 4-221
Solution errors, 3-101
Solution techniques, 1-5, 3-11
Spar elements, 2-144
Specified local system, 3-4
Spherical coordinates, 3-3
Spin softening, 2-33, 3-139

Spring elements, 2-144
Spurious normal rotation, 3-103
SRSS modal combination, 2-69, 3-57
Starin
 energy, 4-12
STATIC, 3-94, 5-1
Static analysis, linear, 2-1, 2-9, 3-10
Static analysis, nonlinear, 2-16
Static condensation, 2-163, 2-164
Stiffness matrix, 2-25, 2-168, 3-99
Strain
 energy, 3-31
 energy function, 2-123
 output, 3-117
 tensor, 2-20
Stress resultants for composite elem., 4-170, 4-176
Stress resultants for line elements, 4-55, 4-66, 4-79, 4-83
Stress stiffened eigenvalue, 3-136
Stress stiffening matrix, 2-29, 2-79, 3-69
Stresses
 2nd Piola-Kirchhoff, 2-22, 3-85, 7-109
 averaged, 4-146, 7-114
 Cauchy, 2-23, 3-122, 7-109
 deviatoric, 2-106, 2-107
 equivalent, 2-106, 3-9, 3-15
 Gauss points, 4-101
 maximum shear, 3-9, 4-12
 modal, 2-8
 nodal, 3-13, 4-12, 7-116
 octahedral shear, 3-9, 3-83, 4-12
 principal, 3-9, 4-12
 sign convention, 3-8
 tensor, 2-32
 von Mises, 3-87, 4-12
Stress-free temperature, 7-70
Stress-strain relations, 2-11, 2-114, 3-81
Structural damping, 2-45, 3-29
Sturm sequence, 3-32
Subspace iteration, 3-30
Surface tractions, 2-12, 2-19
Swanson model, 2-130, 3-85
Tangential loads, 2-144, 2-149
Temperature-dependent properties, 2-87, 3-102, 6-80
Tetrahedron elements, 4-26
THEAT, 3-74
Thermal loads, 3-98, 7-69
Thermal stress analysis, 5-59
Thick shell elements, 4-31
Thin shell elements, 4-161
Time step size, 3-63
Torsional spring elements, 2-148, 4-102, 4-105, 4-106
Total Lagrangian formulation, 2-24, 2-25, 3-15, 5-40
Transfer function, 3-52
Transient dynamic analysis, 2-38, 3-27
Transition elements, 4-2
Translational spring elements, 2-148, 4-90, 4-93
Tresca yield criterion, 2-107
Triangular elements, 4-96
Units, 3-142
Updated Lagrangian formulation, 2-18, 3-15, 5-40
User's subroutines, D-1
Velocity, 2-23, 3-28, 7-123, 8-13
View Factor, 2-94
Viscous damping, 2-45, 3-29, 6-4, 8-19
von Mises stress, 8-100
Von Mises yield criterion, 2-106, 3-83, 6-84
Warning messages, 3-101
Wavefront minimization, 3-100, 5-30
Wavefront solution, 3-1, 3-103
Wedge elements, 4-220

Yield criteria, 2-105, 3-83, 6-90

Young's modulus, 1-3, 3-81

Zero Mean Crossing Rate, 8-106

Zero mean crossing rate, 3-48

I.2 Executive Command Index

ACUTOFF 4, 22
ANALYSIS 13, 4, 18
AUTOCONSTRAINT 37, 81

Symbols

SOLVER 5-41

A

ACUTOFF 8-3, 8-22
ANALYSIS 5-13, 5-22, 8-3, 8-18
AUTOCONSTRAINT 5-35, 5-80

B

BLANK COMMON 5-24
BLANKCOMMON 5-80, 8-3, 8-19

C

CONMethod 5-36
CONTACTALGORITHM 5-36

D

DAMPING 8-3, 8-19
DELTATIME 3-42, 8-3, 8-24
DIRECTION 8-3, 8-24
DPROB 8-19

E

EIGENEXTRACTION 5-37, 5-80
ELEMENT ECHO 5-25
ELEMENTECHO 5-80
ENDTIME 8-3, 8-25
EXECUTION 3-100, 3-107, 3-119, 5-25, 5-80, 8-3, 8-19

F

FILE NAME 5-25
FILE SIZE 5-26
FILENAME 5-13, 5-81, 8-4, 8-20
FLOWER 8-4, 8-25
FRQRDF 8-4, 8-26
FSIZ 5-81
FSMALL 8-4, 8-20
FUPPER 8-4, 8-26

G

GENFREQUENCY 8-4, 8-27
GEOM PROPERTIES 5-26
GEOMPROPERTIES 5-81
GFACTOR 8-4, 8-20

I

INERTIA RELIEF 5-45
INITIALTEMPERATURE 5-81
INITIALTEMPERATURE 5-37
INPHASE 8-4, 8-27
INPOLATION 8-4, 8-28
INTEGRATION 8-4
INTEGRATION . 8-28

M

MASSFORMULATION 5-38, 5-81
MAXCPU TIME 5-38, 5-81
MRESPONSE 8-4, 8-29
MSEXITATION 8-21

N

NLTYPE 5-38, 5-81
NODE ECHO 5-27
NODEECHO 5-81

O

ORTHOTROPIC 5-39, 5-81

P

PAT1 5-81

PAT2 5-81

PAT3 5-81

PAT4 5-81

PATH 5-27, 5-81

R

REFCONFIG 5-40, 5-81

REPL 5-29

RESEQUENCE 3-100, 3-109, 3-110, 3-130,
5-29, 5-46, 5-47

RESTART 5-14, 5-30, 5-67

S

SAVE FILE 5-32

SAVEFILE 5-15, 8-4, 8-21

SIGMA 5-24, 5-41

SOLVER 5-33

SORT STRESS 5-42

SORTSTRESS 5-81

SRAD 5-42, 5-81

STARTINGTIME 8-4, 8-29

STEP 5-43, 5-81

T

TSEISMIC 8-29

U

UFIJ 5-43

W

WARNING 5-35, 8-22

I.3 Data Group Index

Symbols

- *ADDFREQUENCY 8-37
- *ANGSEC 4-152, 7-3
- *APLASTIC 4-11, 6-4
- *ARRIVALTIME 3-42, 8-44
- *BEAMLOAD 3-97, 7-4
- *BMDATA 4-10, 6-37
- *BMSECT 4-10, 6-41
- *BODYFORCE 3-98, 7-4
- *CELEMENTS 6-157
- *CFLUX 3-77, 7-5
- *CFOLLOWER 3-95, 7-58
- *CFORCE 3-95, 7-4, 7-56
- *CONVBC 3-77, 7-96
- *CONVERGENCE 7-37
- *CORRELATION 3-45, 8-52
- *CPDISP 3-91, 6-4
- *CPTEMP 6-4
- *CREEP 6-97
- *CSURFACE 6-147
- *DAMPING 3-30, 8-34
- *DCFORCE 3-44, 8-83
- *DFLUX 3-77, 7-90
- *DPRESSURE 3-41, 8-85
- *DRIVER 8-87
- *E1 6-3, 6-18
- *ECHO 5-7
- *EIGCNTL 7-3, 7-11
- *EIGOUT 7-103
- *ELEMENTS 3-7, 4-10, 6-9
- *ELHEATGEN 3-79, 7-92
- *ELTYPE 6-7
- *ENDDATA 7-132, 8-13, 8-15, 8-118
- *ENVELOPE 8-62
- *EVENT 7-3, 7-14
- *FRCNTL 7-48
- *FRCOEF 7-49
- *G2 6-64
- *GROUND 8-80
- *HEATCNTL 7-3, 7-27
- *HISTORY 8-102
- *HISTOUT 7-5, 7-123
- *HYPEREL 3-86, 6-4, 6-77
- *I5 7-125
- *INITEMP 3-80, 7-86
- *INITIAL 7-73, 8-38
- *L1 7-77
- *LAMANGLE 4-10, 6-25
- *LAMSEQ 4-10
- *LCSYSTEM 3-4, 6-53
- *LCTITLE 7-31
- *LDCASE 7-7
- *LDCOMB 7-28
- *MATDIR1 3-77, 4-11
- *MATDIR2 4-11, 6-71
- *MATERIAL 3-82, 6-4, 6-66
- *MATHEAT 4-11, 6-74
- *MDSPECTRUM 3-54, 8-59
- *MLOAD 6-163
- *MODEOUT 7-32
- *MODESELECTION 3-38, 8-32
- *MPATH 6-166
- *MPCEQN 3-89, 6-111
- *MVELOAD 3-43, 8-89
- *MVEPATH 3-43, 8-92
- *N5 7-127
- *NDHEATGEN 3-79, 7-94
- *NDTEMPDIF 3-99, 7-71
- *NDTEMPER 3-99, 7-69
- *NLOUT 3-118, 7-107
- *NLSRING 4-10, 6-47
- *NODES 3-3, 6-57
- *NONSTATIONARY 8-61
- *PCHANGE1 6-130
- *PDAMPING 6-102
- *PLASTIC 3-83, 4-11, 6-82
- *POSTCNTL 3-135, 7-128
- *PRESSURE 3-96, 7-60
- *PRINTCNTL 3-118, 7-111

- *PRSPECTRA 8-5
- *PSDFUNCTION 8-11, 8-48
- *PSDOUT 8-106
- *RADBC 3-78, 7-99
- *RADSURFACE 3-79, 7-101
- *RCTABLE 4-10, 6-23
- *READ 5-6
- *REGIONS 7-116
- *RESPONSE 3-54, 8-112
- *RIGLINK 3-89, 6-107
- *RMSOUT 3-128, 8-109
- *RSET 3-127, 8-97
- *SETS 6-119
- *SFDCOMP 7-119
- *SNAPSHOT 3-127, 8-104
- *SPDISP 3-92, 7-53
- *SPECTRUM 8-5, 8-56
- *SPOUT 3-129, 8-110
- *SPTEMP 3-79
- *STEPSIZE 7-36
- *STRSFILTER 3-124, 7-117
- *TBODYFORCE 7-75
- *TEMPFN 6-123
- *TEMPHISTORY 7-121
- *TEMPOUT 3-131, 7-122
- *TIMEAMP 6-125
- *TIMEFUNCTION 3-42, 8-45
- *TIMEINTEG 7-34
- *TITLE 6-6, 8-31
- *TSTEP 3-42, 8-36
- *VECTORS 4-10, 6-122
- *WELEMENTS 6-52



**Cranes Software
International Ltd.**

MAIN OFFICES

USA

Cranes Software, Inc.
1133 E Maple Road, Suite 103
Troy, Michigan 48083 USA
P.O. Box 696
Troy, MI, 48099, USA
Phone : (248) 729-3010
Fax : (248) 689-7479
E-mail : sales@nisasoftware.com

INDIA

Cranes Software International Ltd.,
Cranes Engineering,
#52, Bannerghatta Road, B T M Layout,
1st Stage, 1st Phase,
Bangalore – 560029, India.
Phone : +91-80 4128 1111
Fax : +91-80 4128 0203
E-mail : nisa@cranessoftware.com

www.nisasoftware.com

www.cranessoftware.com