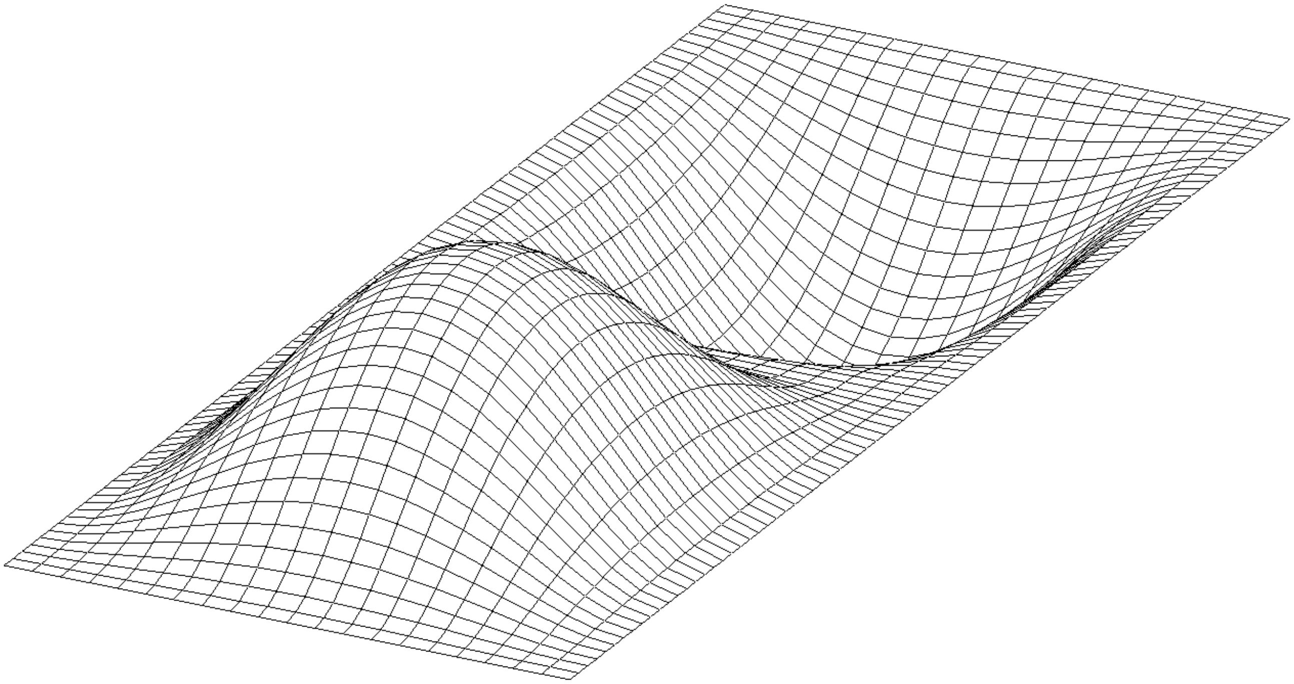
Autodesk Inventor Nastran Solver 2021

User's Manual



© 2020 Autodesk, Inc. All rights reserved.

Autodesk® Inventor® Nastran® 2021

Except as otherwise permitted by Autodesk, Inc., this publication, or parts thereof, may not be reproduced in any form, by any method, for any purpose.

Certain materials included in this publication are reprinted with the permission of the copyright holder.

Trademarks

The following are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and other countries: 123D, 3ds Max, Alias, ATC, AutoCAD LT, AutoCAD, Autodesk, the Autodesk logo, Autodesk 123D, Autodesk Homestyler, Autodesk Inventor, Autodesk MapGuide, Autodesk Streamline, AutoLISP, AutoSketch, AutoSnap, AutoTrack, Backburner, Backdraft, Beast, BIM 360, Burn, Buzzsaw, CADmep, CAiCE, CAMduct, Civil 3D, Combustion, Communication Specification, Configurator 360, Constructware, Content Explorer, Creative Bridge, Dancing Baby (image), DesignCenter, DesignKids, DesignStudio, Discreet, DWF, DWG, DWG (design/logo), DWG Extreme, DWG TrueConvert, DWG TrueView, DWGX, DXF, Ecotect, Ember, ESTmep, Evolver, FABmep, Face Robot, FBX, Fempro, Fire, Flame, Flare, Flint, ForceEffect, FormIt, Freewheel, Fusion 360, Glue, Green Building Studio, Heidi, Homestyler, HumanIK, i-drop, ImageModeler, Incinerator, Inferno, InfraWorks, InfraWorks 360, Instructables, Instructables (stylized robot design/logo), Inventor, Inventor HSM, Inventor LT, Lustre, Maya, Maya LT, MIMI, Mockup 360, Moldflow Plastics Advisers, Moldflow Plastics Insight, Moldflow, Moondust, MotionBuilder, Movimento, MPA (design/logo), MPA, MPI (design/logo), MPX (design/logo), MPX, Mudbox, Navisworks, ObjectARX, ObjectDBX, Opticore, Pixlr, Pixlr-o-matic, Productstream, Publisher 360, RasterDWG, RealDWG, ReCap, ReCap 360, Remote, Revit LT, Revit, RiverCAD, Robot, Scaleform, Showcase, Showcase 360, SketchBook, Smoke, Socialcam, Softimage, Sparks, SteeringWheels, Stitcher, Stone, StormNET, TinkerBox, ToolClip, Topobase, Toxik, TrustedDWG, T-Splines, ViewCube, Visual LISP, Visual, VRED, Wire, Wiretap, WiretapCentral, XSI.

NASTRAN® is a registered trademark of the National Aeronautics Space Administration. All other brand names, product names or trademarks belong to their respective holders.

Disclaimer

THIS PUBLICATION AND THE INFORMATION CONTAINED HEREIN IS MADE AVAILABLE BY AUTODESK, INC. “AS IS.” AUTODESK, INC. DISCLAIMS ALL WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE REGARDING THESE MATERIALS.

TABLE OF CONTENTS

1. INTRODUCTION 13
  1.1 About This Manual 13
  1.2 The Finite Element Method 13
  1.3 Other NASTRAN References 13
  1.4 Obtaining Technical Support 13
2. AUTODESK INVENTOR NASTRAN FILE SYSTEM 15
  2.1 Model Input 15
    2.1.1 Model Initialization File 16
    2.1.2 Model Input File 34
  2.2 Model Output 40
    2.2.1 Model Results Output File 42
    2.2.2 Bulk Data Output File 56
    2.2.3 System Log File 58
    2.2.4 Model Status File 61
    2.2.5 Results Neutral Files 65
    2.2.5.1 Grid Point Displacement Vector Neutral File 66
    2.2.5.2 Grid Point Force Vector Neutral File 67
    2.2.5.3 Element Results Neutral File 69
    2.2.5.4 Grid Point Results Neutral File 71
3. LINEAR BUCKLING ANALYSIS 75
  3.1 Introduction 75
  3.2 How to Setup a Model Input File for Linear Buckling Analysis 75
  3.3 Interpreting Results 76
  3.4 Assumptions and Limitations of Linear Buckling 85
4. NORMAL MODES ANALYSIS 87
  4.1 Introduction 87
  4.2 How to Setup a Model Input File for Normal Modes Analysis 87
  4.3 Interpreting Results 88
  4.4 Rigid-Body Modes 95
  4.5 Direct Matrix Export and Import 99
  4.6 Model Reduction Using ASET 105
  4.7 Model Reduction Using Component Mode Synthesis (Craig-Bampton Reduction) 109
  4.8 Model Reduction Using Superelements 116
  4.9 Modal Database Storage and Retrieval 124
  4.10 Modal Correlation 125
5. LINEAR TRANSIENT RESPONSE ANALYSIS 133
  5.1 Introduction 133
    5.1.1 Direct Transient Response Analysis 133
    5.1.2 Modal Transient Response Analysis 135
  5.2 How to Setup a Model Input File for Transient Response Analysis 138
    5.2.1 Direct Transient Response 138
    5.2.2 Modal Transient Response 138
    5.2.3 Transient Load Definition 138
    5.2.4 Integration Time Step 139

5.2.5 Dynamic Data Recovery 140
5.3 Interpreting Results 141
5.4 Enforced Motion 154
6. SHOCK AND RESPONSE SPECTRUM ANALYSIS 159
6.1 Introduction 159
6.2 Generating Response Spectra Data 159
6.3 How to Setup a Model Input File for Response Spectrum Analysis –Spectrum Generation 161
6.3.1 Response Spectrum Definition 161
6.3.2 Response Spectrum Output 162
6.4 Interpreting Results 163
6.5 Application of Response Spectra Data 169
6.6 How to Setup a Model Input File for Response Spectrum Analysis –Modal Combination 171
6.6.1 Response Spectrum Application 171
6.6.2 Response Spectrum Input 172
6.7 Interpreting Results 173
6.8 Dynamic Design Analysis Method (DDAM). 180
6.9 Interpreting Results 181
7. FREQUENCY RESPONSE ANALYSIS 187
7.1 Introduction 187
7.1.1 Direct Frequency Response Analysis 188
7.1.2 Modal Frequency Response Analysis 189
7.2 How to Setup a Model Input File for Frequency Response Analysis 191
7.2.1 Direct Frequency Response 191
7.2.2 Modal Frequency Response 191
7.2.3 Frequency-Dependent Load Definition 191
7.2.4 Solution Frequencies 192
7.2.5 Dynamic Data Recovery 193
7.3 Interpreting Results 195
7.4 Enforced Motion 201
8. RANDOM RESPONSE ANALYSIS 207
8.1 Introduction 207
8.2 How to Setup a Model Input File for Random Response Analysis 210
8.3 Interpreting Results 211
9. COMPLEX EIGENVALUE ANALYSIS 219
9.1 Introduction 219
9.2 How to Setup a Model Input File for Complex Eigenvalue Analysis 221
9.3 Interpreting Results 222
10. LINEAR PRESTRESS STATIC ANALYSIS 227
10.1 Introduction 227
10.2 How to Setup a Model Input File for Linear Prestress Static Analysis 227
10.3 Interpreting Results 228
11. LINEAR PRESTRESS MODAL ANALYSIS 233
11.1 Introduction 233
11.2 How to Setup a Model Input File for Linear Prestress Modal Analysis 233
11.3 Interpreting Results 234
12. LINEAR PRESTRESS TRANSIENT RESPONSE ANALYSIS 241

12.1 Introduction 241
12.2 How to Setup a Model Input File for Linear Prestress Transient Response Analysis 241
13. LINEAR PRESTRESS FREQUENCY RESPONSE ANALYSIS 243
13.1 Introduction 243
13.2 How to Setup a Model Input File for Linear Prestress Frequency Response Analysis 243
14. LINEAR PRESTRESS COMPLEX EIGENVALUE ANALYSIS 245
14.1 Introduction 245
14.2 How to Setup a Model Input File for Linear Prestress Complex Eigenvalue Analysis 245
15. NONLINEAR STATIC ANALYSIS 247
15.1 Introduction 247
15.2 How to Setup a Model Input File for Nonlinear Static Analysis 248
15.3 Interpreting Results 250
15.3.1 Large Rotations 250
15.3.2 Large Displacements 255
15.3.3 Nonlinear-Elastic Material 261
15.3.4 Thermal-Elastic Material 267
15.3.5 Elastic-Plastic Material 278
15.3.6 Combined Large Displacement and Nonlinear-Elastic Material 286
15.3.7 Combined Gap Contact with Large Displacement and Rotation 291
15.3.8 Gap Contact with Friction 298
15.3.9 Slide Line Contact 304
15.3.10 Tension Only Cable 311
15.3.11 Creep and Viscoelastic Material 320
15.3.12 Arc-length Methods 326
16. NONLINEAR TRANSIENT RESPONSE ANALYSIS 333
16.1 Introduction 333
16.2 How to Setup a Model Input File for Nonlinear Transient Response Analysis 334
16.3 Interpreting Results 336
16.3.1 Impact Analysis 336
16.3.2 Elastic-Plastic Material with Rupture 343
17. NONLINEAR STEADY STATE HEAT TRANSFER ANALYSIS 347
17.1 Introduction 347
17.2 How to Setup a Model Input File for Nonlinear Heat Transfer Analysis 349
17.3 How to Setup a Model Input File for Linear Steady State Heat Transfer Analysis 350
17.4 Interpreting Results 351
17.4.1 Nonlinear Conduction 351
17.4.2 Volume Heat Addition 365
17.4.3 Nonlinear Convection 370
17.4.4 Nonlinear Radiation 378
17.5 Grid Point Temperature Generation 395
18. NONLINEAR TRANSIENT HEAT TRANSFER ANALYSIS 397
18.1 Introduction 397
18.2 How to Setup a Model Input File for Nonlinear Transient Heat Transfer Analysis 398
18.3 Interpreting Results 399
18.3.1 Volume Heat Addition 399
18.3.2 Nonlinear Convection 404
19. NONLINEAR PRESTRESS MODAL ANALYSIS 413
19.1 Introduction 413

19.2 How to Setup a Model Input File for Nonlinear Prestress Modal Analysis.............................. 413
19.3 Interpreting Results 414
20. NONLINEAR PRESTRESS TRANSIENT RESPONSE ANALYSIS... 417
20.1 Introduction 417
20.2 How to Setup a Model Input File for Nonlinear Prestress Transient Response Analysis 417
21. NONLINEAR PRESTRESS FREQUENCY RESPONSE ANALYSIS... 419
21.1 Introduction 419
21.2 How to Setup a Model Input File for Nonlinear Prestress Frequency Response Analysis 419
22. NONLINEAR PRESTRESS COMPLEX EIGENVALUE ANALYSIS 421
22.1 Introduction 421
22.2 How to Setup a Model Input File for Nonlinear Prestress Complex Eigenvalue Analysis....... 421
23. NONLINEAR BUCKLING ANALYSIS 423
23.1 Introduction 423
23.2 How to Setup a Model Input File for Nonlinear Buckling Analysis 423
24. SPECIAL TOPICS 425
24.1 Stress Coordinate Systems 425
24.2 Quad Element Formulation Options 428
24.3 Hex Element Formulation Options 429
24.4 2-Dimensional Composite Analysis 430
24.5 3-Dimensional Composite Analysis 445
24.6 Using Rigid and Interpolation Elements... 452
24.7 Sparse Solver Operation 459
24.8 Optimal Parameter Settings 462
APPENDIX A - OUTPUT FORMATS... 1
Listing A-1. List of Figures for Output Formats.. 1
Figure A-1. Database Subcase Constraint and Load Set Definition. 6
Figure A-2. Database Rectangular Coordinate System Definition... 7
Figure A-3. Database Cylindrical Coordinate System Definition. 8
Figure A-4. Database Spherical Coordinate System Definition. 9
Figure A-5. Database Grid Point Definition. 10
Figure A-6. Database Concentrated Mass Element Definition. 11
Figure A-7. Database Gap Element Definition.. 12
Figure A-8. Database Spring Element Definition. 13
Figure A-9. Database Rod Element Definition.. 14
Figure A-10. Database Bar Element Definition... 15
Figure A-11. Database Beam Element Definition. 16
Figure A-12. Database Quad Element Definition. 17
Figure A-13. Database Tri Element Definition. 18
Figure A-14. Database Shear Element Definition. 19
Figure A-15. Database Hex Element Definition. 20
Figure A-16. Database Pent Element Definition. 21
Figure A-17. Database Tet Element Definition. 22
Figure A-18. Database Gap Element Property Definition. 23

Figure A-19. Database Spring Element Property Definition. 24
Figure A-20. Database Rod Element Property Definition. 25
Figure A-21. Database Bar Element Property Definition. 26
Figure A-22. Database Beam Element Property Definition. 27
Figure A-23. Database Composite Shell Element Property Definition. 28
Figure A-24. Database Shell Element Property Definition. 29
Figure A-25. Database Shear Element Property Definition. 30
Figure A-26. Database Solid Element Material Property Definition. 31
Figure A-27. Database Isotropic Material Definition. 32
Figure A-28. Database Anisotropic Shell Element Material Definition. 33
Figure A-29. Database Isotropic Material Definition (Heat Transfer Analysis). 34
Figure A-30. Database Anisotropic Element Material Definition (Heat Transfer Analysis). 35
Figure A-31. Database Orthotropic Material Definition. 36
Figure A-32. Database Anisotropic Solid Element Material Definition. 37
Figure A-33. Database Nonlinear Material Definition. 38
Figure A-34. Database Isotropic Material Table Definition. 39
Figure A-35. Database Anisotropic Shell Element Material Table Definition. 40
Figure A-36. Database Isotropic Material Table Definition (Heat Transfer Analysis). 41
Figure A-37. Database Orthotropic Material Table Definition (Heat Transfer Analysis). 42
Figure A-38. Database Orthotropic Shell Element Material Table Definition. 43
Figure A-39. Database Anisotropic Solid Element Material Table Definition. 44
Figure A-40. Database Material Property Table Definition. 45
Figure A-41. Database Dynamic Load Table Definition. 46
Figure A-42. Database Modal Damping Table Definition. 47
Figure A-43. Database Single Point Constraint Definition. 48
Figure A-44. Database Single Point Constraint Addition Definition. 49
Figure A-45. Database Multipoint Constraint Definition. 50
Figure A-46. Database Multipoint Constraint Addition Definition. 51
Figure A-47. Database Grid Point Force Vector Definition. 52
Figure A-48. Database Grid Point Moment Vector Definition. 53
Figure A-49. Database Gravity Load Vector Definition. 54
Figure A-50. Database Centrifugal Load Vector Definition. 55
Figure A-51. Database Grid Point Temperature Definition. 56
Figure A-52. Database Dynamic Load Scale Factor Definition. 57
Figure A-53. Database Dynamic Load Time Delay Definition. 58
Figure A-54. Database Dynamic Load Phase Lead Definition. 59
Figure A-55. Database Transient Time Step Definition. 60
Figure A-56. Database Frequency List Definition. 61
Figure A-57. Database Frequency List Generation Definition. 62
Figure A-58. Database Element Initial Deformation Definition. 63
Figure A-59. Database Bar Element Pressure Definition. 64
Figure A-60. Database Shell Element Pressure Definition. 65

Figure A-61. Database Solid Element Pressure Definition. 66
Figure A-62. Database Bar Element Thermal Definition. 67
Figure A-63. Database Shell Element Thermal Definition. 68
Figure A-64. Database Static Load Addition Definition. 69
Figure A-65. Database Transient Response Dynamic Load Definition. 70
Figure A-66. Database Frequency Response Dynamic Load Definition. 71
Figure A-67. Database Power Spectral Density Definition. 72
Figure A-68. Database Power Spectral Density Definition. 73
Figure A-69. Database Static Load Sequence Definition. 74
Figure A-70. Database Grid Point Scalar Load Definition. 75
Figure A-71. Database Line and Point Grid Point Boundary Heat Flux Definition. 76
Figure A-72. Database Surface Grid Point Boundary Heat Flux Definition. 77
Figure A-73. Database Element Boundary Heat Flux Definition. 78
Figure A-74. Database Element Volume Heat Addition Definition. 79
Figure A-75. Database Element Boundary Convection Definition. 80
Figure A-76. Database Subcase Vector Output Set Definition. 81
Figure A-77. Database Subcase Vector Output Set Definition (Heat Transfer Analysis). 82
Figure A-78. Database Subcase Element Output Set Definition. 83
Figure A-79. Database Subcase Element Output Set Definition (Heat Transfer Analysis). 84
Figure A-80. Database Subcase Grid Output Set Definition. 85
Figure A-81. Database Set Definition. 86
Figure A-82. Database Volume Definition. 87
Figure A-83. Database Surface Definition. 88
Figure A-84. Database Spring Element Stress Recovery Property Definition. 89
Figure A-85. Database Rod Element Stress Recovery Property Definition. 90
Figure A-86. Database Bar Element Axis Output Location Definition. 91
Figure A-87. Database Bar Element Stress Recovery Property Definition. 92
Figure A-88. Database Beam Element Stress Recovery Property Definition. 93
Figure A-89. Database Shell Element Stress Recovery Property Definition. 94
Figure A-90. Database Result Limits Definition. 95
Figure A-91. Database Model Parameter Definition. 96
Figure A-92. Model Database Size. 97
Figure A-93. Grid Point Resequencer Output. 98
Figure A-94. Element Geometry Statistics. 99
Figure A-95. Global Stiffness Matrix Assembly Statistics. 100
Figure A-96. Global Mass Matrix Assembly Statistics. 101
Figure A-97. Global Stiffness Matrix Factorization Statistics. 102
Figure A-98. Eigenvalue Extraction Statistics. 103
Figure A-99. Real Eigenvalue Output. 104
Figure A-100. Modal Participation Factor Output. 105
Figure A-101. Rigid-Body Eigenvalue Output. 106
Figure A-102. Modal Effective Mass Output. 107

Figure A-103. Real Eigenvector Output. 108
Figure A-104. Grid Point Weight Output. 109
Figure A-105. Part Definition Output. 110
Figure A-106. Part Mass Properties Output. 111
Figure A-107. Part Element Geometry Summary. 112
Figure A-108. Grid Point Singularity Table. 113
Figure A-109. Mass Matrix Singularity Table. 114
Figure A-110. Load Vector Output. 115
Figure A-111. Load Vector Output (Heat Transfer Analysis). 116
Figure A-112. Displacement Vector Output. 117
Figure A-113. Complex Displacement Vector Output. 118
Figure A-114. Displacement Vector Power Spectral Density Output. 119
Figure A-115. Displacement Vector RMS Output. 120
Figure A-116. Displacement Vector Number of Positive Crossings Output. 121
Figure A-117. Temperature Vector Output. 122
Figure A-118. Single Point Constraint Force Vector Output. 123
Figure A-119. Single Point Constraint Heat Flow Vector Output. 124
Figure A-120. Multipoint Constraint Force Vector Output. 125
Figure A-121. Multipoint Constraint Heat Flow Vector Output. 126
Figure A-122. Load Vector Resultant. 127
Figure A-123. Single Point Constraint Force Vector Resultant. 128
Figure A-124. Maximum Applied Loads. 129
Figure A-125. Maximum Displacements. 130
Figure A-126. Maximum Single Point Constraint Forces. 131
Figure A-127. Nonlinear Forces in Gap Elements. 132
Figure A-128. Forces in Spring Elements. 133
Figure A-129. Stresses in Spring Elements. 134
Figure A-130. Forces in Rod Elements. 135
Figure A-131. Stresses in Rod Elements. 136
Figure A-132. Forces in Bar Elements. 137
Figure A-133. Stresses in Bar Elements. 138
Figure A-134. Thermal Gradients and Heat Fluxes in Bar Elements. 139
Figure A-135. Forces in Shear Elements. 140
Figure A-136. Stresses in Shear Elements. 141
Figure A-137. Heat Flow Into Hbdy Elements. 142
Figure A-138. Forces in Quad Elements (Without Corner Option). 143
Figure A-139. Forces in Quad Elements (With Corner Option). 144
Figure A-140. Stresses in Quad Elements (With Corner Option). 145
Figure A-141. Stresses in Quad Elements (Without Corner Option). 146
Figure A-142. Strains in Quad Elements. 147
Figure A-143. Strain Energy in Quad Elements. 148
Figure A-144. Thermal Gradients and Heat Fluxes in Quad Elements. 149

Figure A-145. Surface Grid Point Stresses. 150
Figure A-146. Surface Grid Point Thermal Gradients and Heat Fluxes. 151
Figure A-147. Stresses in Composite Quad Elements. 152
Figure A-148. Failure Indexes in Composite Quad Elements. 153
Figure A-149. Strength Ratios in Composite Quad Elements. 154
Figure A-150. Stresses in Composite Hex Elements. 155
Figure A-151. Nonlinear Stresses in Quad Elements. 156
Figure A-152. Stresses in Hex Elements (Without Corner Option). 157
Figure A-153. Strains in Hex Elements (Without Corner Option). 158
Figure A-154. Strain Energy in Hex Elements. 159
Figure A-155. Thermal Gradients and Heat Fluxes in Hex Elements. 160
Figure A-156. Volume Grid Point Stresses. 161
Figure A-157. Volume Grid Point Thermal Gradients and Heat Fluxes. 162
Figure A-158. Nonlinear Stresses in Hex Elements. 163
Figure A-159. Complex Stresses in Hex Elements. 164
Figure A-160. Element Result Limits (Subcase Search). 165
Figure A-161. Element Result Limits (Global Search). 166
Figure A-162. Grid Point Result Limits (Subcase Search). 167
Figure A-163. Grid Point Result Limits (Global Search). 168
Figure A-164. Execution Summary. 169
APPENDIX B - LIMITS 1
APPENDIX C - REFERENCES 1

LIST OF ACRONYMS

ACF	Auto Correlation Function
AIR	Automated Inertial Relief
AECG	Automated Edge Contact Generation
ASCG	Automated Surface Contact Generation
CMS	Component Modes Synthesis
DDAM	Dynamic Design Analysis Method
DOF	Degree of Freedom
MAC	Modal Assurance Criteria
MCT	Multicontinuum Theory
MXO	Mass Cross Orthogonality
NPX	Number of Positive Crossings
NRL	Naval Research Laboratories
PCG	Preconditioned Conjugate Gradient
PCGLSS	Preconditioned Conjugate Gradient Linear Sparse Solver
PPFA	Progressive Ply Failure Analysis
PSD	Power Spectral Density
PSI	Pounds per Square Inch
PSS	Parallel Sparse Solver
RAM	Random Access Memory
RMS	Root Mean Square
SDOF	Single Degree of Freedom
SRSS	Square Root of the Sum of the Square
VIS	Vector Iterative Solver
VSS	Vector Sparse Solver

LIST OF SYMBOLS AND ABBREVIATIONS

A	Cross-sectional area
ABS	Absolute value
B	Global damping vector
D	Global displacement vector
DOF	Degree of freedom
E	Young’s modulus
E_T	Tangent modulus
f	Cyclic frequency
G	Modulus of rigidity
g	Gravitational acceleration
H	Plastic modulus
H	Free convection heat transfer coefficient
Hz	Hertz
I	Moment of inertia
ID	Identification number
J	Torsional constant
K	Global stiffness matrix
k	Stiffness
M	Global mass matrix
m	Mass
P	Applied load vector
p	Applied load
ρ	Material density
T	Global temperature vector
υ	Poisson’s ratio
ω	Circular frequency

1. INTRODUCTION

1.1 About This Manual

This manual is intended as a companion to the *Nastran Solver Reference Guide*. It is intended as a guide for users who have experience with NASTRAN. Autodesk Inventor Nastran is different from other versions of NASTRAN. This manual explains these differences and demonstrates program operation through the use of example problems. This manual is not intended to teach you how to build finite element models. For a list of texts on finite element modeling see Section 1.3 *Other NASTRAN References*.

1.2 The Finite Element Method

Autodesk Inventor Nastran uses the finite element method of structural analysis. In this method, the actual structure is subdivided into a finite number of small regions called elements to generate a mathematical model. Within an element displacements and stresses are approximated using polynomial shape functions. An element is connected to adjacent elements at a finite number of points called grid points. Interaction among elements is solely through the forces they exert at the grid points. Element material properties and geometry are used to generate the stiffness of the entire structure, discretized at the grid points. Known loads acting on the structure are represented as forces, also at the grid points. The solution involves using these known loads and stiffnesses to solve for unknown displacements. These are then used to generate element results such as force per unit length, stress, strain, etc.

1.3 Other NASTRAN References

Adams, Vince and Abraham Askenazi, *Building Better Products with Finite Element Analysis*. Santa Fe, NM: OnWord Press, 1999.

Cifuentes, Arturo O., *Using MSC/NASTRAN: Statics and Dynamics*. New York, NY: Springer-Verlag, Inc., 1989.

MacNeal, Richard H., Finite Elements: *Their Design and Performance.* New York, NY: Marcel Dekker, Inc., 1994.

Schaeffer, Harry G., *MSC/NASTRAN Primer: Static and Normal Modes Analysis*. Milford, NH: Wallace Press, Inc., 1984.

1.4 Obtaining Technical Support

If you need help or feel you have discovered a problem in the software, go to https://accounts.autodesk.com. Based on your subscription terms, you can get some online help or you can place a web-request.

Please provide the following information to help us locate the problem and solve it faster:

a) A detailed description of the problem (error messages, problem size, directive, command, and entry types used).

b) Total free physical and virtual memory and free disk space at program execution.

c) If applicable, include a copy of your Model Initialization (*Nastran.INI*), Model Input (*filename.NAS*), Model Results Output (*filename.OUT*), and System Log (*filename.LOG*) files.

d) Any other information you think might be useful.

2. AUTODESK INVENTOR NASTRAN FILE SYSTEM

2.1 Model Input

Autodesk Inventor Nastran uses two files for input definition: the Model Initialization File and the Model Input File. The Model Initialization File configures Autodesk Inventor Nastran to run on your system. It allows you to specify where to get input files, where to put database and output files, how output files should be formatted, how much and what kind of memory to use and how to control program execution. The Model Input File describes your model by specifying the structure’s geometry, material properties, boundary conditions, and loads. Some entities, such as model parameters, can be common to both files.

To illustrate the format of input and output files, we have chosen the simple cantilever beam problem shown in Figure 2-1.

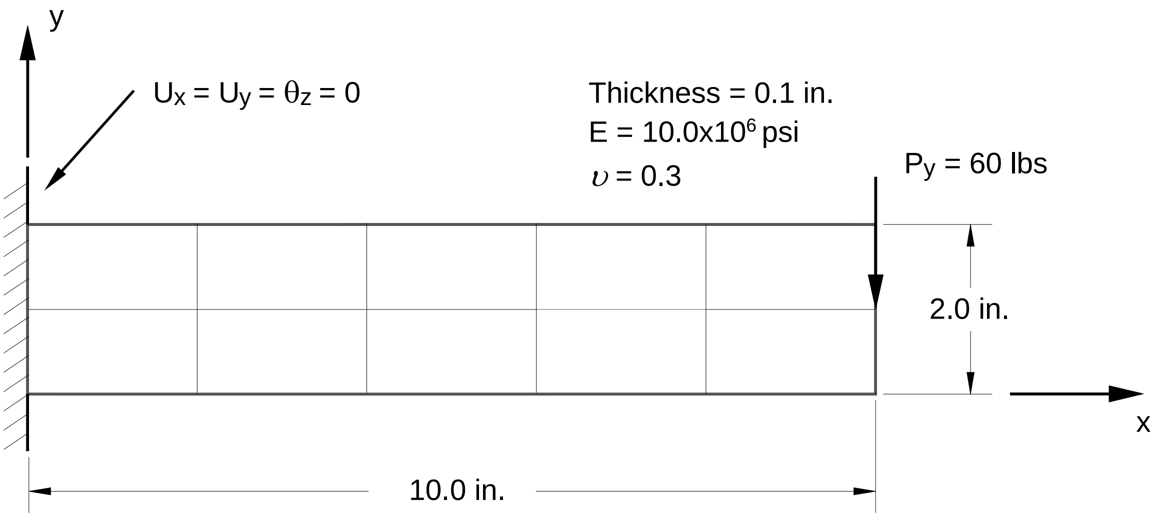


Figure 2-1. Cantilever Beam Example Problem.

The structure is a 0.1 inch thick flat aluminum plate. One end is completely clamped and the other is free. The free end has a 60 pound point load, which results in deflections in the x and y directions. All files in this section pertain to this problem.

2.1.1 Model Initialization File

The Model Initialization File configures Autodesk Inventor Nastran to run on your system. The default Model Initialization File is *Nastran.INI*. It is divided into the following five sections:

Section	Purpose
[File Management]	File Management directives allow you to specify the names and locations of input, output, and database files.
[Output Control]	Output Control directives allow you to control what output files are generated and what they have in them.
[Memory Management]	Memory Management directives allow you to control what type of memory (virtual or real) and how much will be used for memory intensive tasks such as matrix assembly and decomposition. By optimizing memory usage you can optimize performance.
[Program Control]	Program Control directives allow you to customize program execution. Various options are available to change how the program performs certain operations.
[Parameters]	Parameter statements that are specified using the PARAMETER command or entry can be specified in this section using the directive format. See the *Nastran Solver Reference Guide*, Section 5, *Parameters*.

Each section has associated with it a group of related directives and each directive has a default setting. If the default is adequate, the directive need not be included in the file. For more information on directive format see the *Nastran Solver Reference Guide*, Section 2: *Initialization*.

A typical Model Initialization File is shown in Listing 2-1. Note that the Model Input filename and most Model Initialization directives can be specified on the Nastran command line (see *Nastran Solver Reference Guide*, Section 1: *NASTRAN Command Line*). Any file specification or directive specified on the Nastran command line will override the same specification or directive in the Model Initialization File.

Listing 2-1. Example Model Initialization File.

```
[FILE MANAGEMENT DIRECTIVES]

DATABASE          = c:\TEST.NDB
FILESIGNATURE     = T4LT
FILEBUFFERSIZE    = 10
FILESPEC          = C:\Users\timmera\AppData\Local\Temp\NDBT4LT.TMP\
NFILEBUFFER       = 1
PURGE             = ON
RSLTFILEPURGE     = ON

[OUTPUT CONTROL DIRECTIVES]

BULKDATAOUT       = OFF
BULKDATASORT      = ON
DISKSTATUS        = ON
ELAPSEDTIME       = OFF
FEMAPRSLTVECTID   = ON
INCRRSLTOUT       = OFF
LEFTMARGIN        = 1
LINE              = 75
MEMORYSTATUS      = ON
MODLDATAFORMAT    = 3
MODLDATAOUT       = ON
MODLINITOUT       = ON
MODLSTATUS        = DISPLAY
OUTCONTSYMBOL     = OFF
OUTDISPGEOMMODE   = 1
OUTDISPSETID      = 100
OUTGRIDOFFSET     = 100000
OUTLOADSETID      = 100
OUTPAGEFORMAT     = OFF
OUTSPCSETID       = 100
OUTSTRNSETID      = 100
OUTTEMPSETID      = 100
OUTZEROVECT       = OFF
PCHFILEDBLEPRCS   = OFF
PCHFILETYPE       = NASTRAN
RSLTFILECOMP      = AUTO
RSLTFILEDBLEPRCS  = OFF
RSLTFILETYPE      = FEMAP BINARY
SECONDS           = ON
SYSTEMSTATUS      = OFF
TRSLDISPDATA      = OFF
TRSLDMIDATA       = OFF
TRSLMODLDATA      = OFF
TRSLPRESDATA      = OFF
TRSLRBSEDATA      = OFF
TRSLSPCDATA       = OFF
TRSLSTRNDATA      = OFF
TRSLTEMPDATA      = OFF
XYPLOTCSVOUT      = OFF

[MEMORY MANAGEMENT DIRECTIVES]

MAXRAM            = 0
MINRAM            = 200
RAM               = 1200
RESERVEDRAM       = 0
```

Listing 2-1. Example Model Initialization File. (Continued)

```
[PROGRAM CONTROL DIRECTIVES]

DECOMPAUTOSIZE   = 100000
DECOMPMETHOD     = PSS
DYNRSLTMETHOD    = AUTO
EXTRACTAUTOSIZE  = 20000
EXTRACTMETHOD    = AUTO
GPWEIGHTMETHOD   = AUTO
LICENSEMANAGER   = FLEXLM
RESTART          = ON
RSPECDISPMETHOD  = NODAL
RSPECVECTMETHOD  = ON
SHELLEGRID       = OFF
SOLIDEGRID       = OFF
WAITFORLICENSE   = 100
```

2.1.1.1 File Management Directives

The File Management Section contains directives that allow you to specify filenames, file locations, buffer sizes and file deletion. Input and output filenames are determined using file specifications. Input and output file specifications contain two parts, the filename and the path. Database file specifications contain only a path.

Using file specifications you can organize your files into folders or directories and split large models over several hard disks. The only file specification that is required is for the Model Input File. Default values for other file specifications are based on this specification. The following table describes each file specification.

Directive	Description
BULKDATAFILE	Bulk Data Output File specification.
DATABASE	Model Database File specification.
DATINFILE1	Data Input File specification 1.
DATINFILE2	Data Input File specification 2.
DISPFILE	Grid Point Displacement Vector Neutral File specification.
ELEMFILE	Element Results Neutral File specification.
FILEBUFFERSIZE	File buffer size 1 – 4.
FILEBUFFERSIZE1	File buffer size for Model Translator functions.
FILEBUFFERSIZE2	File buffer size for Geometry and Results Processor functions.
FILEBUFFERSIZE3	File buffer size for Solution Processor functions.
NFILEBUFFER	Number of file buffers 1 – 4.
NFILEBUFFER1	Number of file buffers for Model Translator functions.
NFILEBUFFER2	Number of file buffers for Geometry and Results Processor functions.
NFILEBUFFER3	Number of file buffers for Solution Processor functions.
FILESPEC	Model Database File specification 1 – 4.
FILESPEC1	Model Database File specification 1.
FILESPEC2	Model Database File specification 2.
FILESPEC3	Model Database File specification 3.
FILESPEC4	Model Database File specification 4.
FORCFILE	Grid Point Force Vector Neutral File specification.
GRIDFILE	Grid Point Results Neutral File specification.
LOADFILE	Element Internal Load Vector Neutral File specification.
LOGFILE	System Log File specification.
MODALDATFILE	Modal Database File specification.
MODLINFILE	NASTRAN Model Input File specification.
MODLOUTFILE	Model Results Output File specification.
OUTFILESPEC	Output file specification.
PURGE	Automatic output file and database deletion.
RSLTFILEPURGE	Automatic FEMAP Binary and Model Data Output File deletion.

2.1.1.2 Output Control Directives

Output control directives allow you to customize the format and contents of generated model output.

Directive	Description
BULKDATAOUT	Case Control and Bulk Data echo in Model Results Output File.
BULKDATASORT	Output Bulk Data sorting.
DISKSTATUS	Disk space status during critical phases of program execution.
ELAPSEDTIME	System Log File elapsed time output.
FEMAPRSLTVECTID	FEMAP result vector identification numbers in FEMAP results neutral file.
INCRRSLTOUT	Incremental results neutral file output during nonlinear analysis.
LEFTMARGIN	Model Results Output File left margin size in characters.
LINE	Model Results Output File lines per page.
MEMORYSTATUS	Memory status during critical phases of program execution.
MODLDATAOUT	Expanded model data output in Model Results Output File.
MODLINITOUT	Model Initialization File directives echo in Model Results Output File.
MODLSTATUS	Destination of program status information.
OUTDISPGEOMMODE	Specifies the subcase, mode number, or time step for translated deformed geometry.
OUTDISPSETID	Translated enforced displacement set identification number.
OUTGRIDOFFSET	Specifies the starting grid point id associated with generated PLOADG entries.
OUTLOADSETID	Translated force and moment set identification number.
OUTPAGEFORMAT	Model Results Output File page format.
OUTSPCSETID	Translated automatic single point constraint set identification number.
OUTSTRNSETID	Translated element strain set identification number.
OUTTEMPSETID	Translated grid point temperature set identification number.
OUTWIDEFIELD	Option for wide field output in Bulk Data Output File generation.
OUTZEROVECT	Output a zero global vector at a grid point.
PCHFILEDBLEPRCS	Double precision option for Nastran ASCII Result File (.PCH).
PCHFILETYPE	Punch file compatibility option.
RSLTFILECOMP	Results Neutral File compression option.
RSLTFILEMETHOD	Nastran Binary Results File generation method.
RSLTFILETYPE	Results neutral file type and format.
RSLTLABEL	Label location in the title block for the results neutral files.
SECONDS	Process time output in seconds.
SYSTEMSTATUS	System status at the start of program execution.
TRSLDDAMDATA	DDAM data translation option for Bulk Data Output File generation.
TRSLDFGMDATA	Deformed grid point translation option for Bulk Data Output File generation.
TRSLDISPDATA	Enforced displacement translation option for Bulk Data Output File generation.

2.1.1.2 Output Control Directives (Continued)

TRSLDMIDATA	Direct matrix input data translation option for Bulk Data Output File generation.
TRSLLOADDATA	Applied load translation option for Bulk Data Output File generation.
TRSLMODLDATA	Model data translation option for Bulk Data Output File generation.
TRSLPRESDATA	Applied pressure load translation option for Bulk Data Output File generation.
TRSLRBSEDATA	Automatic spring element translation option for Bulk Data Output File generation.
TRSLSPCDATA	Automatic single point constraint translation option for Bulk Data Output File generation.
TRSLSTRNDATA	Solid and shell element strain translation option for Bulk Data Output File generation.
TRSLTEMPDATA	Temperature data translation option for Bulk Data Output File generation.
TRSLTOQEDATA	Reverted tension-only element translation option for Bulk Data Output File generation.
XYPLOTCSVOUT	MS Excel Comma Separated Variable file (.CSV) generation option.

2.1.1.3 Memory Management Directives

Autodesk Inventor Nastran uses two types of built-in memory managers. For acquiring memory from Windows, Autodesk Inventor Nastran uses the Windows Virtual Memory Manager. To control how memory is used internally, Autodesk Inventor Nastran uses a Program Memory Manager. You have limited control of the Windows Virtual Memory Manager in that you can control the size of the paging file and thus the amount of virtual memory available (refer to virtual memory in Windows Help for more information). The Program Memory Manager allocates memory for stiffness matrix assembly, decomposition, and other memory intensive operations. The Autodesk Inventor Nastran Program Memory Manager gives you a few more options for controlling memory through the use of memory management directives. The following table describes each memory management directive.

Directive	Description
MAXRAM	Maximum amount of system memory.
MINRAM	Minimum amount of system memory for the VSS solver out-of-core mode.
RAM	Amount of available system memory.
RESERVEDRAM	Amount of reserved system memory.

2.1.1.4 Program Control Directives

Program Control directives allow you to customize program execution. The below directives deal mainly with the two most numerically demanding and time consuming portions of the program execution, namely stiffness matrix assembly and decomposition. In certain situations values other than default settings may provide increased performance.

Directive	Description
DECOMPMETHOD	Decomposition method used for static solution sequences.
DECOMPAUTOSIZE	Threshold model size for DECOMPMETHOD AUTO setting.
DYNRSLTMETHOD	Dynamic results calculation method.
EXTRACTMETHOD	Eigenvalue extraction method used for modal solution sequences.
EXTRACTAUTOSIZE	Threshold model size for EXTRACTMETHOD AUTO setting.
GPWEIGHTMETHOD	Mass properties calculation method.
HEXEGRID	Hex element automatic edge grid option.
KRIGIDELEM	Stiffness value assigned to spring elements generated from rigid elements.
LICENSECODE	License manager feature code string containing requested license types.
LICENSEMANAGER	License manager type.
MPCMODMETHOD	Multipoint constraint modification method.
NDISKS	Number of physical disk drives for parallel I/O operations.
NPROCESSORS	Number of processors for parallel processing operations.
OPTIMIZESETTINGS	Selects optimum settings for speed, accuracy, or a combination of both.
PCGLSSDMI	DMIG support option for the PCGLSS solver and LANCZOS eigensolver.
PENTEGRID	Pent element automatic edge grid option.
PYREGRID	Pyr element automatic edge grid option.
QUADEGRID	Quad element automatic edge grid option.
RIGIDELEM2ELAS	Rigid element to spring element conversion option.
RESTART	Database restart option when a database is specified for an input file name.
RSPECTDISPMETHOD	Method used in response spectrum analysis for calculating vector results.
RSPECTVECTMETHOD	Method used in response spectrum analysis for calculating element results.
SHELLEGRID	Shell element automatic edge grid option.
SOLIDEGRID	Solid element automatic edge grid option.
SOLUTION	Type of solution sequence.
TETEGRID	Tet element automatic edge grid option.
TRIEGRID	Tri element automatic edge grid option.
WAITFORLICENSE	Wait time for license acquisition from license manager.

2.1.1.5 Parameters

Model parameters are generally specified in the Model Input File. In some cases, however, it is more convenient to specify certain parameters in the Model Initialization File or on the Nastran command line. For example, in the Model Input File you would specify the `K6ROT` parameter as:

```
PARAM, K6ROT, 100.
```

In the Model Initialization File it would be:

```
K6ROT = 100.
```

And on the Nastran command line it would be:

```
NASTRAN test.nas K6ROT=100.
```

Parameters are discussed further in the *Nastran Solver Reference Guide*, Section 5, *Parameters*, and summarized below.

Parameter	Description
ACBINTERACTTOL	Tolerance for removing negligible off-diagonal interaction terms from the acoustic coefficient matrix.
ACBPRESSET	Specifies the remote acoustic grid point output set.
ACBREFPRES	Specifies the acoustic reference pressure used to convert sound pressure into decibels.
ACBVC	Defines the speed of sound in the fluid medium used in boundary acoustic analysis.
ADAPTTIMESTEP	Option for adaptive time stepping in linear direct transient response.
ADDNLTOQUADLOAD	Option for adding loads in tension-only quad and shear panel elements to adjacent line elements.
ADDPRESTRESS	Option for adding prestress subcase results to subsequent subcases.
ADPCON	Specifies the initial penalty values used in slide line and surface contact analysis
ALIGNEDGENODE	Parabolic solid element geometry correction for excessive edge curvature.
ALPHA	Rayleigh damping stiffness matrix scale factor.
ALTFAILINDEXFORM	Alternate failure index formulation for the LaRC02 failure theory.
AUTOBPD	Automatic global mass matrix singularity and non-positive definite correction option.
AUTOCORDROTATE	Option for automatically rotating a projected coordinate system axis that is normal to an element plane.
AUTOFIXELEMGEOM	Option for automatically correcting elements that are singular due to incorrect grid point ordering.
AUTOFIXRIGIDELEM	Option for automatically correcting improperly defined RBE3 elements.
AUTOFIXRIGIDSPC	Option for automatically correcting improperly defined or constrained rigid and interpolation elements.
AUTOSPC	Automatic single point constraint option.
BARDKMETHOD	Specifies how differential stiffness is applied to rod, bar, and beam elements.
BAREQVLOAD	Bar and beam element equivalent load vector formulation option.
BETA	Rayleigh damping mass matrix scale factor.
BISECT	Controls what operation will be performed when the nonlinear iteration limit is reached.
BPDEFDIAG	Mass diagonal coefficient to be used for correcting singular and non-positive definite matrixes.

2.1.1.5 Parameters (Continued)

Parameter	Description
CB1	Scale factor for the DMIG total damping matrix.
CB2	Scale factor for the DMIG total damping matrix.
CHECKOUT	Model check run option used for diagnostics.
CHECKRUN	Model check run option used for diagnostics.
CK1	Used to specify scale factors for the DMIG total stiffness matrix.
CK2	Used to specify scale factors for the DMIG total stiffness matrix.
CLOSE	Tolerance for grouping close modes in modal summation analysis.
CM1	Used to specify scale factors for the DMIG total mass matrix.
CM2	Used to specify scale factors for the DMIG total mass matrix.
COMPE1RSF	Nonlinear composite progressive ply failure E1 reduction scale factor.
COMPE1RSFTID	Nonlinear composite progressive ply failure E1 stress-strain table identification number.
COMPE2RSF	Nonlinear composite progressive ply failure E2 reduction scale factor.
COMPE2RSFTID	Nonlinear composite progressive ply failure E2 stress-strain table identification number.
COMPE3RSF	Nonlinear composite progressive ply failure E3 reduction scale factor.
COMPE3RSFTID	Nonlinear composite progressive ply failure E3 stress-strain table identification number.
COMPG12RSF	Nonlinear composite progressive ply failure G12 reduction scale factor.
COMPG12RSFTID	Nonlinear composite progressive ply failure G12 stress-strain table identification number.
COMPG1ZRSF	Nonlinear composite progressive ply failure G1Z reduction scale factor.
COMPG1ZRSFTID	Nonlinear composite progressive ply failure G1Z stress-strain table identification number.
COMPG23RSF	Nonlinear composite progressive ply failure G23 reduction scale factor.
COMPG23RSFTID	Nonlinear composite progressive ply failure G23 stress-strain table identification number.
COMPG2ZRSF	Nonlinear composite progressive ply failure G2Z reduction scale factor.
COMPG2ZRSFTID	Nonlinear composite progressive ply failure G2Z stress-strain table identification number.
COMPG31RSF	Nonlinear composite progressive ply failure G31 reduction scale factor.
COMPG31RSFTID	Nonlinear composite progressive ply failure G31 stress-strain table identification number.
COMPK1	Foam core composite sandwich stability allowable coefficient.
COMPK2	Honeycomb core composite sandwich stability allowable coefficient.
COMILSMETHOD	Option for defining how composite bond material failure indexes and strength ratios are calculated.
CONTACTGEN	Automated surface contact generation option.
CONTACTSTAB	Surface contact solution stabilization option.
CONTACTTOL	Automated surface contact tolerance.
CONVMATRIX	Convection matrix formulation option.
COUPMASS	Coupled mass matrix generation option.
CP1	Used to specify scale factors for the DMIG total load vector.
CP2	Used to specify scale factors for the DMIG total load vector.

2.1.1.5 Parameters (Continued)

Parameter	Description
CYSYMGEN	Option for automatically generating cyclic symmetric boundary conditions on an axisymmetric model.
CYSYMTOL	Tolerance to identify boundary grid points for the application of cyclic symmetric boundary conditions.
DATABASEACCEL	Model database acceleration option.
DDAMPHASE	DDAM multiphase analysis option.
DELTASTRAINEGOUT	Delta strain energy output option.
DFREQ	Specifies the threshold for the elimination of duplicate frequencies.
DIRSTRESSTYPE	Direct stress type option.
DISPGEOMSFACT	Specifies the scale factor applied to deformed geometry output.
DMILABEL	Specifies the base label for exported matrix data (NAME field on the DMIG Bulk Data entry).
DMIPDIAG	Option to add DMIGP diagonal terms at the DMIGG assembly point.
DYNLMDIRECTDIF	Controls the type of differentiation used in the large mass enforced motion method.
DYNRESPEIGVOUT	Controls the output of normal modes results in modal response solutions.
DYNSOLACCEL	Modal response solution acceleration option.
DYNSOLRELGRID	Specifies the reference point for enforced motion in linear transient and frequency response solutions.
DYNSOLDIRECTINT	Controls the type of solution integration used in linear transient response.
EDGENODETOL	Parabolic solid element geometry correction tolerance for repositioning edge nodes.
EIGENFLEXFREQ	Specifies the threshold frequency for defining the first flexible mode in a modal analysis.
EIGENSHIFTSFACT	Specifies the shift scale multiplier used to increase the shift scale for an eigensolver restart.
EIGENSOLACCEL	Subspace eigensolver acceleration option.
ELEMGEOMCHECKS	Element geometry check option.
ELEMGEOMFATAL	Option to handle certain geometry warnings as fatal errors.
ELEMGEOMOUT	Option to output individual element geometry statistics.
ELEMRSLTCORD	Default SURFACE and VOLUME coordinate system used for computing element results.
ELEMRSLTMAXTYPE	Element location where maximum/minimum stress/strain results are output.
EMODES	Specifies the number of modes extracted during the initialization phase of Automated Impact Analysis.
ENHCBARRSLT	Option for enhanced CBAR and CBEAM element results.
ENHCQUADRSLT	Option for enhanced CQUADR element results.
EPSILONFLOAT	Floating point precision constant for stiffness matrix factorization.
EPZERO	See STIFFRATIOTOL.
EQVSTRESSTYPE	Equivalent stress type option.
EXTOUT	Model and matrix data output option.
FACTDIAG	See SOLUTIONERROR.
FACTRATIOTOL	Stiffness matrix factor diagonal tolerance.
FIXNLTOQUAD	Option to control the reversion of tension-only shell elements.
FLOATINZERO	Character input floating point zero tolerance.

2.1.1.5 Parameters (Continued)

Parameter	Description
FLOATOUTZERO	Model results floating point zero tolerance.
FREQRESPRSLTINCR	Specifies the precision used in calculating real results from complex ones using a sinusoidal sweep.
FREQRESPRSLTOUT	Controls neutral file frequency response output during random response solutions.
G	Specifies the uniform structural damping coefficient in direct transient solutions.
GPFORCEMETHOD	Specifies how grid point forces are calculated.
GPSTRESS	Grid point stress output option.
GPWEIGHT	Grid point weight generator option.
GRDPNT	Grid point weight generator option.
GRIDCOLTOL	Grid collocation tolerance.
GRIDTEMPASGN	Option to assign element temperatures to adjacent grid points.
GRIDTEMPAVE	Element grid point temperature averaging option.
HEXARTOL	Hex element aspect ratio tolerance.
HEXENODE	Hex element edge node option.
HEXFACEMAXIATOL	Hex element face maximum interior angle tolerance.
HEXFACEMINIATOL	Hex element face minimum interior angle tolerance.
HEXFACESKEWTOL	Hex element face skew angle tolerance.
HEXFACETAPERTOL	Hex element face taper ratio tolerance.
HEXFACEWARPTOL	Hex element face warping angle tolerance.
HEXINODE	Hex element internal node option.
HEXMAXEPADTOL	Hex element maximum edge-point angular deviation tolerance.
HEXMINEPLRTOL	Hex element minimum edge-point length ratio tolerance.
HEXREDORD	Hex element reduced order integration option
HFREQ	Specifies the upper modal frequency range to be used in normal modes and dynamic response analysis.
HPNLMATREDORD	Hyperelastic element volumetric reduced order integration option.
HPNLMATSFACT	Specifies the scale factor applied to the hyperelastic element material stiffness matrix.
INERTIALRELIEF	Option to control the calculation of inertial relief or enforced acceleration in STATIC solutions.
INITSTRNSFACT	Specifies the scale factor applied to initial strain values defined on STRAIN Bulk Data entries.
INREL	Option to control the calculation of inertial relief or enforced acceleration in STATIC solutions.
J4ROT	Specifies the stiffness to be added to the torsional rotation for bar and beam elements.
K6ROT	Specifies the stiffness to be added to the normal rotation for CQUAD4 and CTRIA3 elements.
KDAMP	Option for specifying viscous modal damping as structural damping.
KRIGIDELEM	Stiffness value assigned to bush elements generated from converted RBE2 rigid elements.
LANCZOSVECT	Initial starting vector formulation to be used by the Subspace eigensolver.
LANGLE	Specifies the method for processing large rotations in nonlinear analysis.
LARC02TSAITOL	Option to revert failure theory used in composite laminate individual ply results.

2.1.1.5 Parameters (Continued)

Parameter	Description
LFREQ	Specifies the lower modal frequency range to be used in normal modes and dynamic response analysis.
LGDISP	Controls large displacement and follower force effects and differential stiffness in nonlinear analysis.
LINEARCONTACT	Option to control surface contact in linear static solutions.
LMODES	Specifies the number of lowest modes to use in normal modes and dynamic response analysis.
LNCONTACTITERTOL	Linear contact analysis iteration convergence tolerance.
M6ROT	Specifies the inertia to be added to the normal rotation for CQUAD4 and CTRIA3 elements.
MAXADJEDGE	Option for adjusting storage space when using slide line and/or surface contact elements.
MAXBISECTRESTART	Nonlinear solver restart option after maximum bisection error.
MAXEIGENRESTART	Defines the permitted number of eigensolver restarts when an invalid shift scale is estimated.
MAXELEMGEOMMSG	Specifies the maximum element geometry warning/fatal messages that will be output.
MAXIMPACTSTEP	Specifies the maximum number of output steps in Automated Impact Analysis.
MAXINCREFSTRAINP	Specifies the maximum increment of effective plastic strain per nonlinear material subincrement.
MAXLNCONTACTITER	Linear contact analysis maximum number of convergence iterations permitted.
MAXRATIO	Stiffness matrix factor diagonal tolerance.
MAXSPARSEITER	Iterative solver maximum number of iterations permitted.
MAXSRITER	Specifies the maximum number of iterations used in determining composite LaRC02 strength ratios.
MECHSTRAIN	Controls the type of strain output.
MINSPARSEITER	Iterative solver minimum number of iterations performed regardless of convergence.
MODALDATABASE	Controls the storage and retrieval of modal data in dynamic response analysis.
MODEPFACTOR	Controls the calculation and output of modal participation factors and modal effective mass.
MODEFSPCSTORE	Controls the storage and calculation of single point constraint forces in the modal database.
MODEVAROUT	Controls the output of modal variables in modal response solutions.
NBEAMINTNODE	The number of beam internal nodes used when tapered material properties are specified.
NCBMODE	Defines the number of component modes for superelement analysis.
NCONTACTGEOMITER	Specifies the number of iterations for repositioning surface contact element secondary.
NDAMP	Numerical damping option for direct transient solutions.
NITERCUPDATE	Nonlinear solver contact stiffness update option.
NITERKSUPDATE	Nonlinear differential stiffness update option.
NITERMUPDATE	Nonlinear solver material stiffness update option.
NITERPFUPDATE	Nonlinear composite ply failure and stiffness update option.
NLAYERS	Specifies the number of nonlinear material layers in quad and tri elements.
NLINDATABASE	Controls the storage and retrieval of nonlinear restart data used in nonlinear static analysis.
NLINSOLACCEL	Nonlinear solver iteration acceleration option.
NLINSOLTOL	Nonlinear solver default convergence tolerance option.
NLKDIAGAFACT	Specifies the stiffness to be added to diagonal terms of the global stiffness matrix.

2.1.1.5 Parameters (Continued)

Parameter	Description
NLKDIAGCOMP	Specifies component numbers that NLKDIAGAFACT will augment.
NLKDIAGMINAFACT	Specifies the minimum NLKDIAGAFACT value used in nonlinear static solutions.
NLKDIAGSET	Specifies which grid points NLKDIAGAFACT will be applied to by reference to an output set command.
NLLSSTRAINTYPE	Specifies the type of large strain strain output.
NLLSSTRAINTYPE	Specifies the type of large strain stress output.
NLMATSFACT	Specifies the scale factor applied to the material nonlinear portion of the element material stiffness matrix.
NLMATTABLGEN	Option to convert all bi-linear materials defined on MATS1 entries to stress-strain tables.
NLNPDKRESET	Option to use last previously converged tangent stiffness when a non-positive definite is detected.
NLSUBCREINIT	Option to reinitialize the nonlinear database for each subcase thereby restarting the simulation from zero.
NLTOL	Nonlinear solver default convergence tolerance option.
NLTRUESTRESS	Option to output true stress and strain in large displacement nonlinear solutions.
NOCOMPS	Controls the computation and output of composite element ply results.
NSLDPLYINTPOINT	The number of layered solid element ply integration points in the thickness direction of the ply.
NSUBINCRBISECT	Defines the maximum number of nonlinear material subincrements permitted before bisection is initiated.
OGEOM	Controls the output of geometry data blocks to the Nastran Binary Results File.
OPTION	Defines the summation method used to combine modal results in response spectrum analysis.
OUTSETTOL	Tolerance for identifying real values in output set lists.
PARTGEOMOUT	Individual part geometry statistics output option.
PARTMASSOUT	Individual part mass properties output option.
PENTARTOL	Pent element aspect ratio tolerance.
PENTFACEMAXIATOL	Pent element face maximum interior angle tolerance.
PENTFACEMINIATOL	Pent element face minimum interior angle tolerance.
PENTFACESKEWTOL	Pent element face skew angle tolerance.
PENTFACETAPERTOL	Pent element face taper ratio tolerance.
PENTFACEWARPTOL	Pent element face warping angle tolerance.
PENTMAXEPADTOL	Pent element maximum edge-point angular deviation tolerance.
PENTMINEPLRTOL	Pent element minimum edge-point length ratio tolerance.
PENTREDORD	Pent element reduced order integration option.
POST	Controls the output of data blocks to the Nastran Binary Results File.
PRGPST	Option to control the printout of singularities.
PYRARTOL	Pyr element aspect ratio tolerance.
PYRFACEMAXIATOL	Pyr element face maximum interior angle tolerance.
PYRFACEMINIATOL	Pyr element face minimum interior angle tolerance.
PYRFACESKEWTOL	Pyr element face skew angle tolerance.
PYRFACETAPERTOL	Pyr element face taper ratio tolerance.

2.1.1.5 Parameters (Continued)

Parameter	Description
PYRFACEWARPTOL	Pyr element face warping angle tolerance.
PYRMAXEPADTOL	Pyr element maximum edge-point angular deviation tolerance.
PYRMINEPLRTOL	Pyr element minimum edge-point length ratio tolerance.
PYRREDORD	Pyr element reduced order integration option.
QUADARTOL	Quad element aspect ratio tolerance.
QUADBNDREDORD	Quad element bending reduced order integration option.
QUADELEMTYPE	Quad element bending formulation option.
QUADEQVLOAD	Quad element equivalent load vector formulation option.
QUADINODE	Quad element internal node option.
QUADMAXEPADTOL	Quad element maximum edge-point angular deviation tolerance.
QUADMAXIATOL	Quad element maximum interior angle tolerance.
QUADMINEPLRTOL	Quad element minimum edge-point length ratio tolerance.
QUADMINIATOL	Quad element minimum interior angle tolerance.
QUADMEMREDORD	Quad element membrane reduced order integration option.
QUADREDORD	Quad element membrane and bending reduced order integration option.
QUADRNODE	Quad element drill degree of freedom option.
QUADSECT	Quadsect on bisect condition option.
QUADSKEWTOL	Quad element skew angle tolerance.
QUADTAPERTOL	Quad element taper ratio tolerance.
QUADWARPLIMIT	Quad element warping correction option.
QUADWARPTOL	Quad element warping angle tolerance.
RADMATRIX	Radiation matrix formulation option.
RANDRESPINVLEVEL	Controls invariant stress output in frequency and random response solutions.
RANDRESPRSLTOUT	Controls neutral file power spectral density output during random response solutions.
RBCHECKLEVEL	Stiffness matrix equilibrium checks option.
RBCHECKMODES	Specifies the number of modes to solve for in an automated modal rigid body check.
RESEQGRID	Grid point resequence option.
RESEQSTARTGRID	Grid point resequence start grid point identification number.
RESVEC	Residual vector generation option.
RESVPGF	Residual vector zero tolerance.
RIGIDBODYMODE	Subspace eigensolver option to specify how rigid body motion is detected and handled.
RIGIDELEM2ELAS	Rigid element to spring element conversion option.
ROTINERTIA	Diagonal element mass matrix rotary inertia option.
RSLTDATABASE	Controls the storage and retrieval of results data used for restarts in fatigue and explicit dynamics.
RSPECTRA	Option for response spectra generation in a transient response analysis.
SCRSPEC	Modal summation (response spectra input) option for a normal modes analysis.

2.1.1.5 Parameters (Continued)

Parameter	Description
SHEARELEMTYPE	Shear element formulation option.
SHELLEQVLOAD	Shell element equivalent load vector formulation option.
SHELLRNODE	Shell element drill degree of freedom option.
SHELLTVSMATTYPE	Orthotropic shell element transverse shear stiffness type.
SIGMA	Stefan-Boltzmann constant used in heat transfer solutions.
SLINEKAVG	Option for using average of component stiffnesses to determine surface contact penalty values.
SLINEKSFACT	Specifies the initial penalty values used in slide line and surface contact analysis
SLINEKSFACT2TC	Option for treating slide line and surface contact SFACT as contact conductance in heat transfer solutions.
SLINEMAXACTCORD	Specifies the surface contact activation coordinate system.
SLINEMAXACTDIR	Specifies the direction of surface contact movement.
SLINEMAXACTDIST	Specifies the maximum slide line and surface contact element activation distance.
SLINEMAXACTRATIO	Specifies the maximum surface contact element activation ratio.
SLINEMAXACTWIDTH	Defines the total width of the surface contact activation vector.
SLINEMAXDISPTOL	Specifies the normalized maximum allowable contact surface penetration.
SLINEMAXPENDIST	Specifies the maximum slide line and surface contact element penetration distance.
SLINEOFFSETTOL	Specifies the tolerance for automatically converting surface weld elements to offset weld elements.
SLINEPENTOL	Specifies tolerance for adjusting initial penetration errors on contact surfaces.
SLINEPLANEZDIR	Alternate slide line plane normal definition.
SLINEPOSTOL	Specifies tolerance for contact surface segment overlap.
SLINEPROTOL	Specifies tolerance for adjusting initial protrusion errors on contact surfaces.
SLINESLIDETYPE	Contact penalty stiffness update method.
SLINESTABKSFACT	Used to stabilize surface contact in nonlinear static solutions.
SLINEUNLOADTOL	Specifies tolerance for determining a contact surface unload condition.
SOLUTIONERROR	Option to correct the factored stiffness matrix when a singularity or non-positive definite is detected.
SORTMODEMASS	Modal data sorting option.
SPARSEITERMETHOD	Iterative solver preconditioner method.
SPARSEITERMODE	Iterative solver implicit matrix-vector multiply option.
SPARSEITERTOL	Iterative solver convergence factor.
SPARSEMETHOD	Specifies the VSS sparse direct solver matrix reordering method.
SPARSEOUTOFCORE	Parallel sparse direct solver out-of-core option.
SPCGEN	Grid point singularity translation option for Bulk Data Output File generation.
STIFFRATIOTOL	Specifies the minimum global stiffness matrix diagonal ratio for automatic singularity detection.
STIFFZEROTOL	Specifies the minimum value for an off-diagonal stiffness or mass matrix term to be considered nonzero.
STRENGTHRATIO	Tsai Strength Ratio option.
STRESSERROR	Controls the output of normalized grid point stress error (mesh convergence error).

2.1.1.5 Parameters (Continued)

Parameter	Description
TABS	Scale factor for absolute temperature.
TETARTOL	Tet element aspect ratio tolerance.
TETFACEMAXIATOL	Tet element face maximum interior angle tolerance.
TETFACEMINIATOL	Tet element face minimum interior angle tolerance.
TETFACESKEWTOL	Tet element face skew angle tolerance.
TETMAXEPADTOL	Tet element maximum edge-point angular deviation tolerance.
TETMINEPLRTOL	Tet element minimum edge-point length ratio tolerance.
TETREDORD	Tet element reduced order integration option.
TRIARTOL	Tri element aspect ratio tolerance.
TRIBNDREDORD	Tri element bending reduced order integration option
TRIELEMTYPE	Tri element bending formulation option.
TRIEQVLOAD	Tri element equivalent load vector formulation option.
TRIMAXEPADTOL	Tri element maximum edge-point angular deviation tolerance.
TRIMAXIATOL	Tri element maximum interior angle tolerance.
TRIMINEPLRTOL	Tri element minimum edge-point length ratio tolerance.
TRIMINIATOL	Tri element minimum interior angle tolerance.
TRIMEMREDORD	Tri element membrane reduced order integration option
TRIREDORD	Tri element membrane and bending reduced order integration option.
TRIRNODE	Tri element drill degree of freedom option.
TRISKEWTOL	Tri element skew angle tolerance.
TSAI2LARC02	Option to use the LaRC02 failure theory when the Tsai-Wu failure theory is specified.
TSAI2MCT	Option to use the MCT failure theory when the Tsai-Wu failure theory is specified.
TSAI2MCTBVF	Bundle volume fraction used to automatically convert MAT8 Bulk Data entries to MATL8.
TSAI2MCTFVF	Fiber volume fraction used to automatically convert MAT8 Bulk Data entries to MATL8.
UNRESEQGRID	Unresequence model database option.
UNITS	Defines the model units system for output labeling and report generation.
USAWETSURFACE	Underwater Shock Analysis (USA) interface option.
W3	Frequency of interest for global stiffness based structural damping in transient response solutions.
W4	Frequency of interest for element stiffness based structural damping in transient response solutions.
WARNING	Option for disabling output of warning messages.
WTMASS	Global mass matrix scaling factor.
VFM2ACB	Option to perform boundary acoustic analysis when a virtual fluid mass boundary is defined.
VFMADDMETHOD	Specifies when in the solution sequence virtual fluid mass is added to the global mass matrix.
VFMINTERACTTOL	Tolerance for removing negligible off-diagonal fluid interaction terms from the fluid mass matrix.
VFMNORMTOL	Angular tolerance for excluding adjacent grid point surfaces in the fluid mass matrix.

2.1.1.5 Parameters (Continued)

Parameter	Description
VMOPT	Specifies when in the solution sequence virtual fluid mass is added to the global mass matrix.
XDAMP	Controls the use of structural damping in modal response solutions.
ZONADATAOUT	Zona aeroelastic solver output option.

2.1.2 Model Input File

Generating the Model Input File is the first step in performing an analysis using Autodesk Inventor Nastran. The Model Input File defines the structure's geometry, material properties, boundary conditions and loads. In addition, it specifies how the analysis is to be performed and what output is to be included in the Model Results Output File. The Model Input File is an 80 column ASCII text file and can be created using any text editor or one of the many preprocessors that interface with Autodesk Inventor Nastran.

The problem we are analyzing is shown in Figure 2-1 and the corresponding Autodesk Inventor Nastran Model Input File in Listing 2-2. Like most NASTRAN Model Input Files it can be divided into two distinct sections: the Case Control Section and the Bulk Data Section. Input in the Case Control Section is referred to as a command and in the Bulk Data Section as an entry. The Case Control and Bulk Data sections must be assembled in the following sequence (`BEGIN BULK` and `ENDDATA` are required delimiters):

1. *Case Control Commands*

`BEGIN BULK`

2. *Bulk Data Entries*

`ENDDATA`

The Case Control Section begins with the first command and ends with the command, `BEGIN BULK`. It defines the subcase structure for the problem, makes selections from the Bulk Data Section, defines the output coordinate system for element and grid point results, and makes output requests for the Model Results Output File. Case Control commands are described in the *Nastran Solver Reference Guide*, Section 3, *Case Control*.

The Bulk Data Section begins with the entry following `BEGIN BULK` and ends with the entry `ENDDATA`. It contains all of the details of the structural model and the conditions for the solution. `BEGIN BULK` and `ENDDATA` must be present even though no new bulk data is being introduced into the problem or all of the bulk data is coming from an alternate source, such as user-generated input. The format of the `BEGIN BULK` entry is free field. Generally speaking, only one structural model can be defined in the Bulk Data Section. However, some of the bulk data, such as entries associated with loading conditions and constraints, may exist in multiple sets. Only sets selected in the Case Control Section will be used in any particular solution. Bulk Data entries are described in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data*.

Comments may be inserted in either section of the Model Input File. They are identified by a "$" in Column 1 with columns 2-72 containing any desired text. Comments may also be added to Case Control commands and free field Bulk Data entries with a " $" after the last character of data.

Unlike the conventional NASTRAN input file, there is no File Management or Executive Control Sections. These tasks are grouped into one section called Initialization and are handled in the Model Initialization File (see Section 2.1.1, *Model Initialization File*). Input in the Model Initialization File is referred to as a directive.

Once the generation of the Model Input File is complete, you can analyze your model by executing the Nastran command. See the *Nastran Solver Reference Guide*, Section 1, *NASTRAN Command Line*, for more information.

Listing 2-2. Example Model Input File.

```
$
$ STATIC SOLUTION.
$
SOL LINEAR STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = 2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH
$
DISPLACEMENT = ALL
ELFORCE(CORNER) = ALL
ELSTRESS(CORNER) = ALL
$
SPC = 1
SUBCASE 1
 LABEL = 60 LB POINT LOAD IN Y-DIRECTION
 LOAD = 2
$
$ ELEMENT AND GRID POINT STRESS COORDINATE SYSTEM (BASIC).
$
SET 1 = ALL
SURFACE 1, SET 1, SYSTEM BASIC, AXIS X, NORMAL Z
$
BEGIN BULK
$
$ GEOMETRY DEFINITION (10" X 2" RECTANGULAR FLAT PLATE WITH A 5 X 2 MESH).
$
GRID,   1,  ,  10.,  0.,  0.
GRID,   2,  ,  10.,  1.,  0.
GRID,   3,  ,  10.,  2.,  0.
GRID,   4,  ,   2.,  0.,  0.
GRID,   5,  ,   2.,  1.,  0.
GRID,   6,  ,   2.,  2.,  0.
GRID,   7,  ,   6.,  0.,  0.
GRID,   8,  ,   6.,  1.,  0.
GRID,   9,  ,   6.,  2.,  0.
GRID,  10,  ,   4.,  0.,  0.
GRID,  11,  ,   4.,  1.,  0.
GRID,  12,  ,   4.,  2.,  0.
GRID,  13,  ,   8.,  0.,  0.
GRID,  14,  ,   8.,  1.,  0.
GRID,  15,  ,   8.,  2.,  0.
GRID,  16,  ,   0.,  0.,  0.
GRID,  17,  ,   0.,  1.,  0.
GRID,  18,  ,   0.,  2.,  0.
$
$ FLAT PLATE MODELED WITH SHELL ELEMENTS.
$
CQUADR,   1,  10,  16,   4,   5,  17
CQUADR,   2,  10,   4,  10,  11,   5
CQUADR,   3,  10,  10,   7,   8,  11
CQUADR,   4,  10,   7,  13,  14,   8
CQUADR,   5,  10,  13,   1,   2,  14
CQUADR,   6,  10,  17,   5,   6,  18
CQUADR,   7,  10,   5,  11,  12,   6
CQUADR,   8,  10,  11,   8,   9,  12
CQUADR,   9,  10,   8,  14,  15,   9
CQUADR,  10,  10,  14,   2,   3,  15
$
$ ELEMENT MATERIAL AND THICKNESS (0.1").
$
PSHELL,  10,  100,  0.1,  100,  ,  100
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ FIXED BOUNDARY CONDITION AT ONE END.
$
SPC1,  1,  123456,  16,  17,  18
$
$ POINT LOAD ON FREE END (Y-DIRECTION).
$
FORCE,  2,  2,  0,  60.,  0.,  1.,  0.
ENDDATA
```

2.1.2.1 Case Control

The Case Control Section consists of commands which are used to:

- Define the subcase structure for the analysis.
- Select loads and constraints.
- Define the contents of the Model Results Output File.
- Define the output coordinate system for element and grid point results.

The Case Control Section starts with the first line in the Model Input File and ends with the `BEGIN BULK` command. For the cantilever beam example, a title and subtitle are defined that will appear on each page of the Model Results Output File. Reactions and stresses will be included for all grid points and elements. Displacements will be included by default. Subcase 1 is defined using the `LABEL`, `SPC`, and `LOAD` commands. The `SPC` command directs Autodesk Nastran to apply constraints defined by the `SPC1` entry with an identification number (ID) of 1 in the Bulk Data Section. The `LOAD` command directs Autodesk Nastran to apply loading defined by the `FORCE` entry with an ID of 2 in the Bulk Data Section. The `SURFACE` and `SET` commands define the element results output coordinate system. The default coordinate system is defined using the `ELEMRSLTCORD` model parameter (see the *Nastran Solver Reference Guide*, Section 5, *Parameters,* for more information).

In the cantilever beam example, only one set of loads and constraints were specified. In general, a separate subcase is defined for each loading condition and/or each set of constraints. Subcases may also be used in connection with output requests, such as requesting different output for a load case. Only one level of subcase definition is provided. All items placed above the subcase level (ahead of the first subcase) will be used for all following subcases unless overridden within the individual subcase.

Consider the cantilever beam example again. Suppose we wanted to look at additional load conditions, for example, point loads in the x-direction and z-direction. In addition, for the x-direction load case, we would like a different boundary condition at the fixed end and element strain energy output. For the z-direction load case, we do not want to output element or grid point stresses. The Case Control Section would then look like this:

```
SPCFORCES = ALL
ELSTRESS = ALL
GPSTRESS = ALL
SPC = 1
SUBCASE 1
 LABEL = 60 LB POINT LOAD IN Y-DIRECTION
 LOAD = 2
SUBCASE 2
 LABEL = 60 LB POINT LOAD IN Z-DIRECTION
 LOAD = 3
 ELSTRESS = NONE
 GPSTRESS = NONE
SUBCASE 3
 LABEL = 60 LB POINT LOAD IN X-DIRECTION
 SPC = 2
 LOAD = 4
 ESE = ALL
```

Three subcases are defined in this example. Since the constraints are the same for `SUBCASE 1` and `SUBCASE 2` and the subcases are contiguous, the stiffness matrix factorization portion of the solution will only be repeated once for `SUBCASE 3`. Displacements (by default) and element and grid point stresses will be output for all subcases. Forces of single point constraint (reactions) will be output for all subcases except `SUBCASE 2`. Element strain energies will only be output for `SUBCASE 3`.

2.1.2.2 Bulk Data

The Bulk Data Section defines your model by allowing you to specify geometry (grid points, element connectivity, etc.), material properties, boundary conditions (constraints) and loading (forces, moments, pressures, etc.). The start of the Bulk Data Section is denoted by the `BEGIN BULK` delimiter and the end, the `ENDDATA` delimiter. Both are delimiters are required.

The Case Control Section has control over entries that describe boundary conditions and loading. In the cantilever beam example that would mean the `SPC1` and `FORCE` entries only. All other entries are always included in the model regardless of what the Case Control Section specifies. This allows you to have multiple load cases and control what load cases are used for a given analysis. Constraint and load entries can exist in the model and not be called unless needed. In addition, material property and coordinate system entries can exist that are never referenced. An error message will result, however, if an element or grid point references a material property or coordinate system that does not exist.

In the cantilever beam example, the model's geometry is defined via the `GRID` entry. Each grid point coordinate is defined in the default basic coordinate system and all units are in inches. You may pick whatever units you like as long as you are consistent. Element connectivity is defined via the `CQUADR` entries. The plate thickness and material property are defined on the `PSHELL` entry. The isotropic material that the beam is made from is defined using the `MAT1` entry.

Within the Bulk Data Section, entries may be in any order since a sort is performed prior to the execution of the Model Translator. Bulk data will be echoed in the Model Results File if `ECHO = SORT` or `ECHO = UNSORT` is specified in the Case Control Section or `BULKDATAOUT = ON` in the Model Initialization File. Also, a sorted copy of the Model Input File will be written to the Bulk Data File if the `TRSLMODLDATA` directive is set to `ON` in the Model Initialization File.

Bulk Data entries may be entered either in fixed- or free field format. Free field format will be discussed at the end of this section. Fixed field format is divided into small and large field formats. Large field format can be used when small field does not provide enough significant digits. For small field format a data line is divided into 10 fields, each with eight characters as shown below:

1	2	3	4	5	6	7	8	9	10
←8→	←8→	←8→	←8→	←8→	←8→	←8→	←8→	←8→	←8→

The following is an example of small field fixed format:

1	2	3	4	5	6	7	8	9	10
GRID	100	20	1.0	10.5	0.0	17			

Large field format requires (at least) two lines for each entry: the first and last field of each data line contains eight characters and the four fields between contain 16 characters as shown below:

Line 1:

1A	2	3	4	5	6
←8→	←16→	←16→	←16→	←16→	←8→

Line 2:

1B	6	7	8	9	10B
←8→	←16→	←16→	←16→	←16→	←8→

The following is an example of large field fixed format:

Line 1:

1A	2	3	4	5	6
GRID∗	100	20	1.0	10.5	∗C0001

Line 2:

1B	6	7	8	9	10
∗C0001	0.0	17			

Large field entries are denoted by an asterisk (∗) immediately following the character string in field 1A of the first line and immediately preceding the character string in field 1B of the second line.

For all formats the name of the Bulk Data entry is input in field 1 beginning in column 1. Fields 2-9 are for data items. The only limitations in data items are that they must lie completely within the designated field, have no embedded blanks, and must be of the proper type (i.e., blank, integer, real, or character). All real numbers, including zero, must contain a decimal point. A blank will be interpreted as a real zero or integer zero, as required. Real numbers may be encoded in various ways. For example, the real number `7.0` may be encoded as `7.0`, `.7E1`, `0.7+1`, `70.-1`, `.70+1`, `7+0`, etc.

Field 10 of the Bulk Data entry is used for two purposes. If the Bulk Data entry does not have a continuation line, field 10 may be used as an optional comment field. A “;” is still required as the first character of the comment. If the Bulk Data entry has a continuation line, field 10 is used for the continuation identifier. The continuation contains the symbol + in column 1 followed by the same seven characters that appeared in columns 74-80 of field 10 of the entry that is being continued. Character strings used as continuation identifiers cannot contain the symbol “$” in column 1 or “;” in any column. The continuation identifier must be unique with respect to all the other identifiers in your Bulk Data.

Continuation fields can also be generated automatically by Autodesk Nastran. To automatically generate a continuation, the continuation line (or lines) must immediately follow the parent Bulk Data entry. In addition, fields 1 and 10 of the continuation line (or lines) must be left blank.

Free field format provides an easier method for inputting data manually. An example of free field is shown in Listing 2-2. In the free field format, commas are used to separate the fields. An entry in free field format is identified by a comma or equal sign in any of the first nine columns. The following rules apply to the use of the free field format:

- Free field data entries must start in column 1.
- A comma must separate data items.
- Data must be eight characters or less.
- If automatic continuation is to be used, the continuation line starts with a comma in field 1.

The following is an example of free field format:

```
GRID, 100, 20, 1., 10.5, , 17
```

The following is an example of free field format with automatic continuation.

```
CBAR, 10, 100, 201, 202, 0., 0., 1., ,
, , ,1., 0., 0., 1., 0., 0.
```

2.2 Model Output

This section discusses the output files that are produced for a typical run. As is the previous section we will use the cantilever beam example shown in Figure 2-1 as an example.

When Autodesk Nastran is executed it generates several output files. Some output files are always generated and others are optionally controlled through the use of Model Initialization directives (see 2.1.1 Model Initialization File). Table 2-1 provides a description of all the files that will be generated if you execute Autodesk Nastran using the Model Initialization File in Listing 2-1 and the Model Input File in Listing 2-2. The Model Input filename is *Test.NAS* and can be found in the installation folder.

All files in Table 2-1, except the Model Database files, are ASCII files and can be viewed with any text editor. The Model Results Output File lines per page setting can be changed with the `LINE` Model Initialization directive. This value is usually set to correspond to your system printer.

The following sections provide further discussion and an example listing of each file.

Table 2-1. Files Generated in the Cantilever Beam Example.

Filename	Definition	Description	Associated Initialization Directives
Test.INI	Model Initialization File	Model specific system configuration and model parameter settings. If not specified, *Nastran.INI* is used.	All directives are specified in this file.

Test.OUT	Model Results Output File	The main output file containing model definition and results output (displacements, forces, stresses, strains, etc.)	`BULKDATAOUT`
			`BULKDATASORT`
			`LEFTMARGIN`
			`LINE`
			`OUTPAGEFORMAT`
			`OUTZEROVECT`
			`SECONDS`

Test.BDF	Bulk Data Output File	A complete NASTRAN input file generated from the Model Database. Note that a *translated* Bulk Data Output File is not generated if a fatal error occurs in the Model Translator Module. In this case, the file will be a duplicate of the Model Input File with the appropriate error messages.	`OUTCONTSYMBOL`
			`OUTSPCSETID`
			`OUTTEMPSETID`
			`TRSLDMIDATA`
			`TRSLMODLDATA`
			`TRSLTEMPDATA`

Test.STA	Model Status File	Displayed program status information can be written to this file, if requested. Useful for checking program status remotely and debugging.	`DISKSTATUS`
			`MEMORYSTATUS`
			`MODLSTATUS`
			`SECONDS`
			`SYSTEMSTATUS`

Test.LOG	System Log File	An abbreviated summary of the system and model status generated as the program executes. All error messages are written to this file.	`DISKSTATUS`
			`MEMORYSTATUS`
			`SECONDS`
			`SYSTEMSTATUS`

Table 2-1. Files Generated in the Cantilever Beam Example. (Continued)

Filename	Definition	Description	Associated Initialization Directives
Test.RSF	Results Summary File	An abbreviated summary of results data obtained by scanning the System Log File for user specified search strings defined in the Model Initialization File.	`RSLTSUMFILEENTRY`

Test.NDB	Database Pointer File	Stores the locations of the Model Database files. Database files are located in a user supplied folder specified in the Model Initialization File. By specifying the `PURGE` Model Initialization directive, these files will automatically be deleted prior to normal program termination. Other than normal program termination will require you to manually delete these files. If you plan to re-run the model this is not necessary. Database files all end with a .NDB extension.	`DATABASE`
			`FILESPEC1`
			`FILESPEC2`
			`FILESPEC3`
			`NFILEBUFFER1`
			`NFILEBUFFER2`
			`NFILEBUFFER3`
			`FILEBUFFERSIZE1`
			`FILEBUFFERSIZE2`
			`FILEBUFFERSIZE3`

Test.DIS	Grid Point Displacement Vector Neutral File	Displacement results neutral file used for storing displacement vector data.	`DISPFILE`
			`RSLTLABEL`
			`RSLTFILETYPE`

Test.GPF	Grid Point Force Vector Neutral File	Nodal results neutral file used for storing vector results (applied loads, reactions, velocities, and accelerations) calculated at the grid points.	`FORCFILE`
			`RSLTLABEL`
			`RSLTFILETYPE`

Test.ELS	Element Results Neutral File	Element results neutral file used for storing element results data (forces, stresses, strains, etc.) calculated at the element centroid and corner nodes.	`ELEMFILE`
			`RSLTLABEL`
			`RSLTFILETYPE`

Test.GPS	Grid Point Results Neutral File	Nodal results neutral file used for storing element results data (forces, stresses, strains, etc.) calculated at the grid points.	`GRIDFILE`
			`RSLTLABEL`
			`RSLTFILETYPE`

Test.NEU
Test.FNO
	FEMAP Neutral File	FEMAP compatible results neutral file used to import results into FEMAP.	`FEMAPRSLTVECTID`
			`INCRRSLTOUT`
			`RSLTLABEL`
			`RSLTFILETYPE`

Test.OP2	NASTRAN Binary Results File	NASTRAN Output 2 compatible results neutral file used to import results into HyperMesh, Patran, I-Deas, etc.	`RSLTFILETYPE` `RSLTFILEPURGE`
Test.XDB	NASTRAN XDB Results File	NASTRAN XDB compatible results neutral file used to import results into Patran, Pro/E, etc.	`RSLTFILETYPE` `RSLTFILEPURGE`
Test.PCH	NASTRAN ASCII Results File	Vector results data in NASTRAN PUNCH format.	`MODLOUTFILE`

2.2.1 Model Results Output File

The format and contents of the Model Results Output File can vary from statistical information only to output of all translated and calculated data. Listing 2-3 represents a typical output file. The contents of Listing 2-3 are now briefly discussed.

Autodesk Nastran is divided into 18 separate modules:

- Initialization Processor Module
- Model Translator Module
- Geometry Processor Module
- Linear Solution Processor Module
- Results Processor Module
- Real Eigenvalue Processor Module
- Nonlinear Solution Processor Module
- Initial Stress Processor Module
- Modal Transient Response Processor Module
- Modal Frequency Response Processor Module
- Modal Complex Eigenvalue Processor Module
- Direct Transient Response Processor Module
- Direct Frequency Response Processor Module
- Nonlinear Transient Response Processor Module
- Nonlinear Transient Heat Transfer Processor Module
- Matrix Reduction Processor Module
- Component Assembly Processor Module
- Superelement Assembly Processor Module

Output from each module contributes to the Model Results Output File.

The Model Initialization directive, `MODLINITOUT`, controls output from the Initialization Processor module. Since `MODLINITOUT` was set to `ON` in the Model Initialization File, all Model Initialization directives and their assigned values are included.

The Model Initialization directives, `MODLDATAOUT` and `MODLDATAFORMAT`, control output from the Model Translator Module. All displayed model data comes directly from the Model Database. Because of the unambiguous format, this method is preferred over the Model Input File echo generated with the `BULKDATAOUT` directive.

Output from the Geometry Processor Module consists of statistical information from the grid point resequencing and global stiffness matrix assembly processes.

The Grid Point Resequencer internally renumbers the grid points to optimize performance and reduce disk space usage. It is controlled with the `RESEQGRID`, `UNRESEQGRID`, `RESEQSTARTGRID`, and `RESEQGRIDMETHOD` parameters and the directive. The goal is to minimize the global stiffness matrix profile. If the profile is not decreased, the original numbering is used. Note that the matrix size shown is a conservative estimate.

After the grid point renumbering, the global stiffness matrix is formed. During assembly, each element's geometry is checked and statistical information relative to this process is output. Output from the Solution Processor Module consists of statistical information and grid point vector results (applied loads, displacements, and reactions). This information will repeat for each subcase.

The first step in the solution sequence consists of the application of prescribed and automatic constraints. An automatic constraint will be applied to a degree of freedom if it has very little or no stiffness associated with it. This feature is controlled by the `AUTOSPC` parameter. When this parameter is used, a table is included in the output indicating which degrees of freedom in the model are constrained. Maximum and minimum matrix diagonal terms are also included and should always be checked.

The next and usually the most time consuming step in the solution sequence is the factorization of the global stiffness matrix. Output from this process consists of statistical information. If a problem occurs during factorization, additional diagnostic information will be provided.

After factorization, the global load vector is formed and the unknown displacements and forces are determined. These are output in tabular form and are controlled with the `DISPLACEMENT` and `SPCFORCES` Case Control commands. Following the displacement vector is the solution's epsilon and strain energy. Epsilon is a measure of the numeric conditioning of the model, while the strain energy is the work due to applied loads. Both of these values are useful for model checkout and are discussed further in Appendix A, *Output Formats*.

Output from the Results Processor Module consists of element and grid point results data derived from the global displacement vector. The `ELSTRESS` and `GPSTRESS` Case Control commands control which elements are output. Element and grid point forces, stresses, and strains are output in the surface coordinate system defined in the Case Control Section of the Model Input File. Grid point results derived from averaged element corner stresses and strains. Following the results on each element on a surface are the maximum and minimum values for selected result types.

Listing 2-3. Example Model Results Output File.

```
                                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
2



                              M O D E L   I N I T I A L I Z A T I O N   D I R E C T I V E S



 [FILE MANAGEMENT DIRECTIVES]

 DATABASE         = c:\TEST.NDB
 FILESIGNATURE    = T4LT
 FILEBUFFERSIZE   = 10
 FILESPEC         = C:\Users\timmera\AppData\Local\Temp\NDBT4LT.TMP\
 NFILEBUFFER      = 1
 PURGE            = ON
 RSLTFILEPURGE    = ON

 [OUTPUT CONTROL DIRECTIVES]

 BULKDATAOUT      = OFF
 BULKDATASORT     = ON
 DISKSTATUS       = ON
 FEMAPRSLTVECTID  = ON
 INCRRSLTOUT      = OFF
 LEFTMARGIN       = 1
 LINE             = 75
 MEMORYSTATUS     = ON
 MODLDATAOUT      = ON
 MODLDATAFORMAT   = 1
 MODLINITOUT      = ON
 MODLSTATUS       = DISPLAY
 OUTCONTSYMBOL    = OFF
 OUTDISPSETID     = 100
 OUTDISPGEOMMODE  = 1
 OUTDISPSETID     = 100
 OUTGRIDOFFSET    = 100000
 OUTLOADSETID     = 100
 OUTPAGEFORMAT    = OFF
 OUTSPCSETID      = 100
 OUTSTRNSETID     = 100
 OUTTEMPSETID     = 100
 OUTZEROVECT      = OFF
 PCHFILEDBLEPRCS  = OFF
 PCHFILETYPE      = NASTRAN
 RSLTFILECOMP     = AUTO
 RSLTFILEDBLEPRCS = OFF
 RSLTFILETYPE     = FEMAP BINARY
 SECONDS          = ON
 SYSTEMSTATUS     = OFF
 TRSLDISPDATA     = OFF
 TRSLDMIDATA      = OFF
 TRSLMODLDATA     = OFF
 TRSLPRESDATA     = OFF
 TRSLRBSEDATA     = OFF
 TRSLSPCDATA      = OFF
 TRSLSTRNDATA     = OFF
 TRSLTEMPDATA     = OFF
 XYPLOTCSVOUT     = OFF

 [MEMORY MANAGEMENT DIRECTIVES]

 MAXRAM           = 0
 MINRAM           = 0
 RAM              = 1800
 RESERVEDRAM      = 0

 [PROGRAM CONTROL DIRECTIVES]

 DECOMPAUTOSIZE   = 100000
 DECOMPMETHOD     = AUTO
 DYNRSLTMETHOD    = AUTO
 EXTRACTAUTOSIZE  = 20000
 EXTRACTMETHOD    = AUTO
 GPWEIGHTMETHOD   = AUTO
 LICENSEMANAGER   = FLEXLM
 RESTART          = ON
 RSPECDISPMETHOD  = NODAL
 RSPECVECTMETHOD  = ON
 SHELLEGRID       = OFF
 SOLIDEGRID       = OFF
 WAITFORLICENSE   = 100
```

Listing 2-3. Example Model Results Output File. (Continued)

```
INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
3
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                       S U B C A S E   C O N S T R A I N T   A N D   L O A D   S E T   D E F I N I T I O N

  SUBCASE                                LABEL                               CONSTRAINT SET IDS             LOAD SET IDS
    ID                                                                         SPC        MPC       LOAD      DEFORM   TEMPERATURE
      1     60 LB EDGE LOAD IN Y-DIRECTION                                       1       NONE          2       NONE       NONE



INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
4
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                                           G R I D   P O I N T   D E F I N I T I O N

        1            0         1.000E+01    0.000E+00    0.000E+00
        2            0         1.000E+01    1.000E+00    0.000E+00
        3            0         1.000E+01    2.000E+00    0.000E+00
        4            0         2.000E+00    0.000E+00    0.000E+00
        5            0         2.000E+00    1.000E+00    0.000E+00
        6            0         2.000E+00    2.000E+00    0.000E+00
        7            0         6.000E+00    0.000E+00    0.000E+00
        8            0         6.000E+00    1.000E+00    0.000E+00
        9            0         6.000E+00    2.000E+00    0.000E+00
       10            0         4.000E+00    0.000E+00    0.000E+00
       11            0         4.000E+00    1.000E+00    0.000E+00
       12            0         4.000E+00    2.000E+00    0.000E+00
       13            0         8.000E+00    0.000E+00    0.000E+00
       14            0         8.000E+00    1.000E+00    0.000E+00
       15            0         8.000E+00    2.000E+00    0.000E+00
       16            0         0.000E+00    0.000E+00    0.000E+00
       17            0         0.000E+00    1.000E+00    0.000E+00
       18            0         0.000E+00    2.000E+00    0.000E+00



INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
5
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                                            Q U A D   E L E M E N T   D E F I N I T I O N

  ELEMENT     PROPERTY      GRID-1       GRID-2       GRID-3       GRID-4       MATERIAL     ORIENTATION     OFFSET
    ID           ID           ID           ID           ID           ID       COORDINATE ID     ANGLE
      1           10           16            4            5           17         ELEMENT          0.00      0.000E+00
      2           10            4           10           11            5         ELEMENT          0.00      0.000E+00
      3           10           10            7            8           11         ELEMENT          0.00      0.000E+00
      4           10            7           13           14            8         ELEMENT          0.00      0.000E+00
      5           10           13            1            2           14         ELEMENT          0.00      0.000E+00
      6           10           17            5            6           18         ELEMENT          0.00      0.000E+00
      7           10            5           11           12            6         ELEMENT          0.00      0.000E+00
      8           10           11            8            9           12         ELEMENT          0.00      0.000E+00
      9           10            8           14           15            9         ELEMENT          0.00      0.000E+00
     10           10           14            2            3           15         ELEMENT          0.00      0.000E+00



INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
6
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                                  S H E L L   E L E M E N T   P R O P E R T Y   D E F I N I T I O N

 PROPERTY                       MATERIAL IDS                        THICKNESS      TS/T        12I/T3         NSM
    ID        MEMBRANE      BENDING    TRANSVERSE    COUPLING
     10          100          100          100                      1.000E-01    8.333E-01    1.000E+00    0.000E+00
```

Listing 2-3. Example Model Results Output File. (Continued)

```
INSTALLATION TEST CASE                                                    01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
7
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                                      I S O T R O P I C   M A T E R I A L   D E F I N I T I O N

 MATERIAL           E            G            NU           RHO          ALPHA        T-REF
    ID
    100        1.000E+07    3.846E+06    3.000E-01    1.000E-01    0.000E+00    0.000E+00




INSTALLATION TEST CASE                                                    01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
8
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                                 S I N G L E   P O I N T   C O N S T R A I N T   D E F I N I T I O N

   SET          GRID         COMPONENT       ENFORCED
    ID           ID           NUMBERS      DISPLACEMENT
     1           16           123456         0.000E+00
     1           17           123456         0.000E+00
     1           18           123456         0.000E+00




INSTALLATION TEST CASE                                                    01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
9
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                                 G R I D   P O I N T   F O R C E   V E C T O R   D E F I N I T I O N

   SET          GRID      COORDINATE                FORCE VECTOR
    ID           ID           ID            V1           V2           V3
     2            1            0         0.000E+00    1.500E+01    0.000E+00
     2            2            0         0.000E+00    3.000E+01    0.000E+00
     2            3            0         0.000E+00    1.500E+01    0.000E+00




INSTALLATION TEST CASE                                                    01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
10
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                              S U B C A S E   V E C T O R   O U T P U T   S E T   D E F I N I T I O N

  SUBCASE          LOAD        DISPLACEMENT       VELOCITY       ACCELERATION       SPC FORCE        MPC FORCE
    ID            VECTOR          VECTOR           VECTOR          VECTOR           VECTOR           VECTOR
     1            NONE             ALL              NONE            NONE              ALL              NONE




INSTALLATION TEST CASE                                                    01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
11
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                             S U B C A S E   E L E M E N T   O U T P U T   S E T   D E F I N I T I O N

  SUBCASE         ELEMENT          ELEMENT          ELEMENT          ELEMENT
    ID             FORCE           STRESS           STRAIN        STRAIN ENERGY
     1             NONE             ALL              NONE             NONE




INSTALLATION TEST CASE                                                    01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
12
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                                S U B C A S E   G R I D   O U T P U T   S E T   D E F I N I T I O N

  SUBCASE        GRID POINT       GRID POINT       GRID POINT     GRID POINT STRESS
    ID             FORCE            STRESS           STRAIN        DISCONTINUITY
```

```
       1            NONE               ALL              NONE              NONE
```

Listing 2-3. Example Model Results Output File. (Continued)

```
INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
13
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                                                     S E T   D E F I N I T I O N

   SET   ELEMENT/GRID IDS
    ID
      1          ALL


INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
14
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                                                 S U R F A C E   D E F I N I T I O N

  SURFACE    ELEMENT SET     COORDINATE       X-AXIS       NORMAL
    ID           ID              ID
      1            1                0            X            Z




INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
15
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                  S H E L L   E L E M E N T   S T R E S S   R E C O V E R Y   P R O P E R T Y   D E F I N I T I O N

 PROPERTY          Z1           Z2
    ID
     10        -5.000E-02    5.000E-02




INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
16
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                                         M O D E L   P A R A M E T E R   D E F I N I T I O N


 ELEMENT STIFFNESS MATRIX FORMULATION PARAMETERS

 QUAD ELEMENT VERTEX ROTATION                      = ON
 QUAD ELEMENT INTERNAL NODE                        = ON
 QUAD ELEMENT MEMBRANE REDUCED ORDER INTEGRATION   = ON
 QUAD ELEMENT BENDING REDUCED ORDER INTEGRATION    = ON



 ELEMENT LOAD VECTOR FORMULATION PARAMETERS

 QUAD ELEMENT EQUIVALENT LOAD VECTOR FORMULATION   = ON


 MODEL SOLUTION SEQUENCE PARAMETERS

 STIFFNESS MATRIX AUTOMATIC SINGLE POINT CONSTRAINT                                = ON
 STIFFNESS MATRIX DIAGONAL RATIO TOLERANCE FOR AUTOMATIC SINGLE POINT CONSTRAINT   = 1.000000E-08
 STIFFNESS MATRIX FACTOR DIAGONAL RATIO TOLERANCE FOR MECHANISM DETECTION          = 1.000000E+05
 FLOATING POINT PRECISION CONSTANT FOR STIFFNESS MATRIX FACTORIZATION              = 1.000000E-15
 MODEL RESULTS FLOATING POINT ZERO TOLERANCE                                       = 1.000000E-15
```

Listing 2-3. Example Model Results Output File. (Continued)

```
INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
17
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                                                M O D E L   D A T A B A S E   S I Z E



 MODEL DATABASE SIZE

 SUBCASES                     =        1
 GRID POINTS                  =       18
 ELEMENTS                     =       10
 ELEMENT PROPERTIES           =        1
 MATERIAL PROPERTIES          =        1
 SINGLE POINT CONSTRAINTS     =        3
 GRID POINT FORCES            =        3
 SETS                         =        1
 SURFACES                     =        1




INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
18
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH




 MODEL SIZE    =        108 DEGREES OF FREEDOM
 MATRIX SIZE   =       2070 WORDS        0.0 MEGABYTES
 SEMIBANDWIDTH =         47 WORDS

 ELEMENT GEOMETRY STATISTICS

 MAXIMUM QUAD ELEMENT WARPING ANGLE =   0.00 DEGREES ON ELEMENT 10
 MAXIMUM QUAD ELEMENT SKEW ANGLE    =   0.00 DEGREES ON ELEMENT 10
 MAXIMUM QUAD ELEMENT TAPER RATIO   =   0.00         ON ELEMENT 10
 MAXIMUM QUAD ELEMENT ASPECT RATIO  =   2.00         ON ELEMENT 10

 GLOBAL STIFFNESS MATRIX ASSEMBLY STATISTICS

 SPARSE MATRIX SIZE =       1962 WORDS        0.0 MEGABYTES
 MEMORY ALLOCATED   =       3105 WORDS        0.0 MEGABYTES

 MAXIMUM GLOBAL STIFFNESS MATRIX TERM ZEROED =  9.9206E-17
 MINIMUM GLOBAL STIFFNESS MATRIX TERM ZEROED =  0.0000E+00
 REDUCTION IN GLOBAL STIFFNESS MATRIX SIZE   =  58.00 PERCENT

 ASSEMBLY TIME FOR 10 ELEMENTS = 0.1 SECONDS




INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
19
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH

60 LB EDGE LOAD IN Y-DIRECTION                                      SUBCASE 1




 MAXIMUM STIFFNESS MATRIX DIAGONAL =  3.0578E+06  AT GRID 14 COMPONENT 2
 MINIMUM STIFFNESS MATRIX DIAGONAL =  1.5359E+04  AT GRID 1  COMPONENT 5




INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
20
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH

60 LB EDGE LOAD IN Y-DIRECTION                                      SUBCASE 1




 GLOBAL STIFFNESS MATRIX FACTORIZATION STATISTICS

 NUMBER OF NEGATIVE TERMS ON FACTOR DIAGONAL = 0
 MAXIMUM MATRIX FACTOR DIAGONAL RATIO = 2.773E+03 AT GRID 8 COMPONENT 3

 FACTORED SPARSE MATRIX SIZE =       1308 WORDS        0.0 MEGABYTES
 ADDITIONAL MEMORY ALLOCATED =      53670 WORDS        0.4 MEGABYTES
```

```
FACTORIZATION TIME FOR 1308 WORDS = 0.0 SECONDS
```

Listing 2-3. Example Model Results Output File. (Continued)

```
INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
21
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH

60 LB EDGE LOAD IN Y-DIRECTION                                      SUBCASE 1

                                             D I S P L A C E M E N T   V E C T O R

   GRID      COORDINATE        T1            T2            T3            R1            R2            R3
    ID           ID
     1             0      4.514572E-03  3.047095E-02  0.000000E+00  0.000000E+00  0.000000E+00  4.332973E-03
     2             0      0.000000E+00  3.045806E-02  0.000000E+00  0.000000E+00  0.000000E+00  4.018986E-03
     3             0     -4.514572E-03  3.047095E-02  0.000000E+00  0.000000E+00  0.000000E+00  4.332973E-03
     4             0      1.584370E-03  1.820868E-03  0.000000E+00  0.000000E+00  0.000000E+00  1.424074E-03
     5             0      0.000000E+00  1.659872E-03  0.000000E+00  0.000000E+00  0.000000E+00  1.507010E-03
     6             0     -1.584370E-03  1.820868E-03  0.000000E+00  0.000000E+00  0.000000E+00  1.424074E-03
     7             0      3.792008E-03  1.321103E-02  0.000000E+00  0.000000E+00  0.000000E+00  3.666624E-03
     8             0      0.000000E+00  1.315702E-02  0.000000E+00  0.000000E+00  0.000000E+00  3.348591E-03
     9             0     -3.792008E-03  1.321103E-02  0.000000E+00  0.000000E+00  0.000000E+00  3.666624E-03
    10             0      2.877639E-03  6.417065E-03  0.000000E+00  0.000000E+00  0.000000E+00  2.774258E-03
    11             0      0.000000E+00  6.299692E-03  0.000000E+00  0.000000E+00  0.000000E+00  2.549374E-03
    12             0     -2.877639E-03  6.417065E-03  0.000000E+00  0.000000E+00  0.000000E+00  2.774258E-03
    13             0      4.330951E-03  2.147459E-02  0.000000E+00  0.000000E+00  0.000000E+00  4.151929E-03
    14             0      0.000000E+00  2.145137E-02  0.000000E+00  0.000000E+00  0.000000E+00  3.860329E-03
    15             0     -4.330951E-03  2.147459E-02  0.000000E+00  0.000000E+00  0.000000E+00  4.151929E-03




 MAXIMUM DISPLACEMENT MAGNITUDE =  3.080358E-02  AT GRID 3
 MAXIMUM ROTATION MAGNITUDE     =  4.332973E-03  AT GRID 1

 EPSILON       =  4.727113E-14
 STRAIN ENERGY =  9.139352E-01

 SOLUTION TIME FOR 108 DEGREES OF FREEDOM = 0.0 SECONDS




INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
22
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH

60 LB EDGE LOAD IN Y-DIRECTION                                      SUBCASE 1

                               F O R C E S   O F   S I N G L E - P O I N T   C O N S T R A I N T

   GRID      COORDINATE        T1            T2            T3            R1            R2            R3
    ID           ID
    16             0     -3.000000E+02 -6.983513E+01  0.000000E+00  0.000000E+00  0.000000E+00  2.430190E+01
    17             0     -1.008971E-12  7.967025E+01  0.000000E+00  0.000000E+00  0.000000E+00 -4.860380E+01
    18             0      3.000000E+02 -6.983513E+01  0.000000E+00  0.000000E+00  0.000000E+00  2.430190E+01



INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
23
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH

60 LB EDGE LOAD IN Y-DIRECTION                                      SUBCASE 1


 MAXIMUM SINGLE POINT CONSTRAINT FORCE MAGNITUDE  =  3.080210E+02  AT GRID 16
 MAXIMUM SINGLE POINT CONSTRAINT MOMENT MAGNITUDE =  4.860379E+01  AT GRID 17




INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
24
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                                          L O A D   V E C T O R   R E S U L T A N T

  SUBCASE         T1            T2            T3            R1            R2            R3
    ID
     1       0.000000E+00  6.000000E+01  0.000000E+00  0.000000E+00  0.000000E+00  6.000000E+02
```

Listing 2-3. Example Model Results Output File. (Continued)

```
INSTALLATION TEST CASE                                                01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
25
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                          S I N G L E   P O I N T   C O N S T R A I N T   V E C T O R   R E S U L T A N T

  SUBCASE         T1            T2            T3            R1            R2            R3
    ID
      1     -2.614797E-12 -6.000000E+01  0.000000E+00  0.000000E+00  0.000000E+00 -6.000000E+02




INSTALLATION TEST CASE                                                01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
26
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                                           M A X I M U M   A P P L I E D   L O A D S

  SUBCASE         T1            T2            T3            R1            R2            R3
    ID
      1      0.000000E+00  3.000000E+01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00




INSTALLATION TEST CASE                                                01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
27
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                                           M A X I M U M   D I S P L A C E M E N T S

  SUBCASE         T1            T2            T3            R1            R2            R3
    ID
      1      4.514572E-03  3.047095E-02  0.000000E+00  0.000000E+00  0.000000E+00  4.332973E-03




INSTALLATION TEST CASE                                                01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
28
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                          M A X I M U M   F O R C E S   O F   S I N G L E   P O I N T   C O N S T R A I N T

  SUBCASE         T1            T2            T3            R1            R2            R3
    ID
      1      3.000000E+02  7.967025E+01  0.000000E+00  0.000000E+00  0.000000E+00  4.860380E+01
```

Listing 2-3. Example Model Results Output File. (Continued)

```
INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
29
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH

60 LB EDGE LOAD IN Y-DIRECTION                                      SUBCASE 1


                         S T R E S S E S   I N   Q U A D   E L E M E N T S   O N   S U R F A C E   1

SURFACE COORDINATE ID = 0  X-AXIS = X  NORMAL = Z

  ELEMENT     GRID        FIBER       STRESSES IN SURFACE COORDINATE SYSTEM       PRINCIPAL STRESSES (ZERO SHEAR)        HENCKY
    ID         ID       DISTANCE       NORMAL-X     NORMAL-Y     SHEAR-XY         ANGLE        MAJOR        MINOR       VON MISES

      1      CENTER   -5.00000E-02    4.07969E+03  2.80702E+02  3.00000E+02      4.4875    4.10324E+03  2.57158E+02    3.98089E+03
                       5.00000E-02    4.07969E+03  2.80702E+02  3.00000E+02      4.4875    4.10324E+03  2.57158E+02    3.98089E+03

      2      CENTER   -5.00000E-02    3.12228E+03  5.78723E+01  3.00000E+02      5.5391    3.15137E+03  2.87790E+01    3.13708E+03
                       5.00000E-02    3.12228E+03  5.78723E+01  3.00000E+02      5.5391    3.15137E+03  2.87790E+01    3.13708E+03

      3      CENTER   -5.00000E-02    2.23803E+03 -3.02839E+01  3.00000E+02      7.4081    2.27703E+03 -6.92903E+01    2.31246E+03
                       5.00000E-02    2.23803E+03 -3.02839E+01  3.00000E+02      7.4081    2.27703E+03 -6.92903E+01    2.31246E+03

      4      CENTER   -5.00000E-02    1.35088E+03 -2.49630E+01  3.00000E+02     11.7809    1.41345E+03 -8.75320E+01    1.45919E+03
                       5.00000E-02    1.35088E+03 -2.49630E+01  3.00000E+02     11.7809    1.41345E+03 -8.75320E+01    1.45919E+03

      5      CENTER   -5.00000E-02    4.46982E+02 -9.15219E+00  3.00000E+02     26.3785    5.95763E+02 -1.57933E+02    6.88453E+02
                       5.00000E-02    4.46982E+02 -9.15219E+00  3.00000E+02     26.3785    5.95763E+02 -1.57933E+02    6.88453E+02

      6      CENTER   -5.00000E-02   -4.07969E+03 -2.80702E+02  3.00000E+02     85.5125   -2.57158E+02 -4.10324E+03    3.98089E+03
                       5.00000E-02   -4.07969E+03 -2.80702E+02  3.00000E+02     85.5125   -2.57158E+02 -4.10324E+03    3.98089E+03

      7      CENTER   -5.00000E-02   -3.12228E+03 -5.78723E+01  3.00000E+02     84.4609   -2.87790E+01 -3.15137E+03    3.13708E+03
                       5.00000E-02   -3.12228E+03 -5.78723E+01  3.00000E+02     84.4609   -2.87790E+01 -3.15137E+03    3.13708E+03

      8      CENTER   -5.00000E-02   -2.23803E+03  3.02839E+01  3.00000E+02     82.5919    6.92903E+01 -2.27703E+03    2.31246E+03
                       5.00000E-02   -2.23803E+03  3.02839E+01  3.00000E+02     82.5919    6.92903E+01 -2.27703E+03    2.31246E+03

      9      CENTER   -5.00000E-02   -1.35088E+03  2.49630E+01  3.00000E+02     78.2191    8.75320E+01 -1.41345E+03    1.45919E+03
                       5.00000E-02   -1.35088E+03  2.49630E+01  3.00000E+02     78.2191    8.75320E+01 -1.41345E+03    1.45919E+03

     10      CENTER   -5.00000E-02   -4.46982E+02  9.15219E+00  3.00000E+02     63.6215    1.57933E+02 -5.95763E+02    6.88453E+02
                       5.00000E-02   -4.46982E+02  9.15219E+00  3.00000E+02     63.6215    1.57933E+02 -5.95763E+02    6.88453E+02




 MAXIMUM QUAD ELEMENT PRINCIPAL STRESS =  4.103236E+03  AT ELEMENT 1
 MINIMUM QUAD ELEMENT PRINCIPAL STRESS = -4.103236E+03  AT ELEMENT 6
 MAXIMUM QUAD ELEMENT SHEAR STRESS     =  1.923039E+03  AT ELEMENT 6
 MAXIMUM QUAD ELEMENT VON MISES STRESS =  3.980892E+03  AT ELEMENT 6
```

Listing 2-3. Example Model Results Output File. (Continued)

```
INSTALLATION TEST CASE                                                  01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
30
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH

60 LB EDGE LOAD IN Y-DIRECTION                                      SUBCASE 1


                             G R I D   P O I N T   S T R E S S E S   O N   S U R F A C E   1

SURFACE COORDINATE ID = 0  X-AXIS = X  NORMAL = Z

   GRID          FIBER       STRESSES IN SURFACE COORDINATE SYSTEM       PRINCIPAL STRESSES (ZERO SHEAR)        HENCKY
    ID         DISTANCE      NORMAL-X     NORMAL-Y     SHEAR-XY         ANGLE        MAJOR        MINOR       VON MISES

      1      -5.00000E-02    4.22446E+02 -1.02593E+02  5.82048E+01      6.2506    4.28821E+02 -1.08968E+02    4.92432E+02
              5.00000E-02    4.22446E+02 -1.02593E+02  5.82048E+01      6.2506    4.28821E+02 -1.08968E+02    4.92432E+02

      2      -5.00000E-02    5.95719E-11 -3.41771E-11  5.41795E+02     45.0000    5.41795E+02 -5.41795E+02    9.38417E+02
              5.00000E-02    5.95719E-11 -3.41771E-11  5.41795E+02     45.0000    5.41795E+02 -5.41795E+02    9.38417E+02

      3      -5.00000E-02   -4.22446E+02  1.02593E+02  5.82048E+01     83.7494    1.08968E+02 -4.28821E+02    4.92432E+02
              5.00000E-02   -4.22446E+02  1.02593E+02  5.82048E+01     83.7494    1.08968E+02 -4.28821E+02    4.92432E+02

      4      -5.00000E-02    7.27173E+03 -3.20149E+02  2.83476E+02      2.1354    7.28230E+03 -3.30719E+02    7.45317E+03
              5.00000E-02    7.27173E+03 -3.20149E+02  2.83476E+02      2.1354    7.28230E+03 -3.30719E+02    7.45317E+03

      5      -5.00000E-02    4.29168E-12  3.41771E-12  3.16524E+02     45.0000    3.16524E+02 -3.16524E+02    5.48235E+02
              5.00000E-02    4.29168E-12  3.41771E-12  3.16524E+02     45.0000    3.16524E+02 -3.16524E+02    5.48235E+02

      6      -5.00000E-02   -7.27173E+03  3.20149E+02  2.83476E+02     87.8646    3.30719E+02 -7.28230E+03    7.45317E+03
              5.00000E-02   -7.27173E+03  3.20149E+02  2.83476E+02     87.8646    3.30719E+02 -7.28230E+03    7.45317E+03

      7      -5.00000E-02    3.64043E+03  6.25816E+01  7.28930E+01      1.1667    3.64192E+03  6.10971E+01    3.61176E+03
              5.00000E-02    3.64043E+03  6.25816E+01  7.28930E+01      1.1667    3.64192E+03  6.10971E+01    3.61176E+03

      8      -5.00000E-02    2.02647E-11  2.70752E-11  5.27107E+02     45.0000    5.27107E+02 -5.27107E+02    9.12976E+02
              5.00000E-02    2.02647E-11  2.70752E-11  5.27107E+02     45.0000    5.27107E+02 -5.27107E+02    9.12976E+02

      9      -5.00000E-02   -3.64043E+03 -6.25816E+01  7.28930E+01     88.8333   -6.10971E+01 -3.64192E+03    3.61176E+03
              5.00000E-02   -3.64043E+03 -6.25816E+01  7.28930E+01     88.8333   -6.10971E+01 -3.64192E+03    3.61176E+03

     10      -5.00000E-02    5.59876E+03  1.21774E+01  1.67094E+02      1.7117    5.60375E+03  7.18410E+00    5.60017E+03
              5.00000E-02    5.59876E+03  1.21774E+01  1.67094E+02      1.7117    5.60375E+03  7.18410E+00    5.60017E+03

     11      -5.00000E-02    4.29168E-12 -1.84528E-11  4.32906E+02     45.0000    4.32906E+02 -4.32906E+02    7.49816E+02
              5.00000E-02    4.29168E-12 -1.84528E-11  4.32906E+02     45.0000    4.32906E+02 -4.32906E+02    7.49816E+02

     12      -5.00000E-02   -5.59876E+03 -1.21774E+01  1.67094E+02     88.2883   -7.18410E+00 -5.60375E+03    5.60017E+03
              5.00000E-02   -5.59876E+03 -1.21774E+01  1.67094E+02     88.2883   -7.18410E+00 -5.60375E+03    5.60017E+03

     13      -5.00000E-02    1.80216E+03  3.41200E+01  5.82143E+01      1.8838    1.80407E+03  3.22053E+01    1.78819E+03
              5.00000E-02    1.80216E+03  3.41200E+01  5.82143E+01      1.8838    1.80407E+03  3.22053E+01    1.78819E+03

     14      -5.00000E-02    2.01368E-11 -6.79279E-11  5.41786E+02     45.0000    5.41786E+02 -5.41786E+02    9.38400E+02
              5.00000E-02    2.01368E-11 -6.79279E-11  5.41786E+02     45.0000    5.41786E+02 -5.41786E+02    9.38400E+02

     15      -5.00000E-02   -1.80216E+03 -3.41200E+01  5.82143E+01     88.1162   -3.22053E+01 -1.80407E+03    1.78819E+03
              5.00000E-02   -1.80216E+03 -3.41200E+01  5.82143E+01     88.1162   -3.22053E+01 -1.80407E+03    1.78819E+03

     16      -5.00000E-02    7.99996E+03  1.07349E+03  3.20327E+02      2.6422    8.01474E+03  1.05870E+03    7.54134E+03
              5.00000E-02    7.99996E+03  1.07349E+03  3.20327E+02      2.6422    8.01474E+03  1.05870E+03    7.54134E+03

     17      -5.00000E-02    6.59384E-12 -1.34150E-11  2.79673E+02     45.0000    2.79673E+02 -2.79673E+02    4.84408E+02
              5.00000E-02    6.59384E-12 -1.34150E-11  2.79673E+02     45.0000    2.79673E+02 -2.79673E+02    4.84408E+02

     18      -5.00000E-02   -7.99996E+03 -1.07349E+03  3.20327E+02     87.3578   -1.05870E+03 -8.01474E+03    7.54134E+03
              5.00000E-02   -7.99996E+03 -1.07349E+03  3.20327E+02     87.3578   -1.05870E+03 -8.01474E+03    7.54134E+03




 MAXIMUM SHELL ELEMENT PRINCIPAL STRESS =  8.014745E+03  AT GRID 16
 MINIMUM SHELL ELEMENT PRINCIPAL STRESS = -8.014745E+03  AT GRID 18
 MAXIMUM SHELL ELEMENT SHEAR STRESS     =  3.806512E+03  AT GRID 6
 MAXIMUM SHELL ELEMENT VON MISES STRESS =  7.541336E+03  AT GRID 18
```

Listing 2-3. Example Model Results Output File. (Continued)

```
INSTALLATION TEST CASE                                             01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
31
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH



                                 I N T E R N A L   L O A D   V E C T O R   R E S U L T A N T

  SUBCASE         T1            T2            T3            R1            R2            R3
    ID
      1      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00




INSTALLATION TEST CASE                                             01/16/15  12:12  ADS NASTRAN VERSION 10.3.0.716  PAGE
32
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH




 MODEL ANALYSIS TIME SUMMARY

 TOTAL CPU TIME = 0.9 SECONDS
 WALLCLOCK TIME = 1.0 SECONDS

 EXECUTION TERMINATED NORMALLY

 TOTAL WARNINGS     = 0
 TOTAL FATAL ERRORS = 0
```

2.2.2 Bulk Data Output File

The Bulk Data Output File is a complete NASTRAN input file generated from the Model Database whenever the Model Initialization directive `TRSLBULKDATA` is set to `ON`. This file can also be included in the Model Results Output File by setting the directive, `BULKDATAOUT`, to `ON`. Sorting of the Bulk Data Section of this file is controlled with the `BULKDATASORT` directive. Note that if model translation terminates in a fatal error, the Bulk Data Output File will be a duplicate of the Model Input File with the appropriate error messages. A translated Bulk Data Output File is not generated if a fatal error occurs in the Model Translator Module.

Listing 2-4. Example Bulk Data Output File.

```
$
$
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = 2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH
SET 1 = ALL
SURFACE 1, SET 1, SYSTEM BASIC, AXIS X, NORMAL Z
DISPLACEMENT  = ALL
SPCFORCES  = ALL
ELFORCE(CORNER) = ALL
ELSTRESS(CORNER) = ALL
SUBCASE 1
 LABEL = 60 LB EDGE LOAD IN Y-DIRECTION
 SPC = 1
 LOAD = 2
BEGIN BULK
$
$------1-------2-------3-------4-------5-------6-------7-------8-------9-------0
GRID           1       0 10.0000      0.      0.       0
GRID           2       0 10.0000 1.00000      0.       0
GRID           3       0 10.0000 2.00000      0.       0
GRID           4       0 2.00000      0.      0.       0
GRID           5       0 2.00000 1.00000      0.       0
GRID           6       0 2.00000 2.00000      0.       0
GRID           7       0 6.00000      0.      0.       0
GRID           8       0 6.00000 1.00000      0.       0
GRID           9       0 6.00000 2.00000      0.       0
GRID          10       0 4.00000      0.      0.       0
GRID          11       0 4.00000 1.00000      0.       0
GRID          12       0 4.00000 2.00000      0.       0
GRID          13       0 8.00000      0.      0.       0
GRID          14       0 8.00000 1.00000      0.       0
GRID          15       0 8.00000 2.00000      0.       0
GRID          16       0      0.      0.      0.       0
GRID          17       0      0. 1.00000      0.       0
GRID          18       0      0. 2.00000      0.       0
CQUADR         1      10      16       4       5      17      0.      0.
CQUADR         2      10       4      10      11       5      0.      0.
CQUADR         3      10      10       7       8      11      0.      0.
CQUADR         4      10       7      13      14       8      0.      0.
CQUADR         5      10      13       1       2      14      0.      0.
CQUADR         6      10      17       5       6      18      0.      0.
CQUADR         7      10       5      11      12       6      0.      0.
CQUADR         8      10      11       8       9      12      0.      0.
CQUADR         9      10       8      14      15       9      0.      0.
CQUADR        10      10      14       2       3      15      0.      0.
PSHELL        10     100 0.10000     100 1.00000     100 0.83333      0.
        -0.05000 0.05000
MAT1         1001.0000+73.8462+6 0.30000 0.10000      0.      0.      0.
              0.      0.      0.
SPC1           1  123456      16
SPC1           1  123456      17
SPC1           1  123456      18
FORCE          2       1       0 15.0000      0. 1.00000      0.
FORCE          2       2       0 30.0000      0. 1.00000      0.
FORCE          2       3       0 15.0000      0. 1.00000      0.
```

```
ENDDATA
```

2.2.3 System Log File

The System Log File contains solution status and results summary information generated as the program executes. All program generated error messages are written to this file.

Listing 2-5. Example System Log File.

```
ADS NASTRAN VERSION 10.3.0.716   16:27  06/14/14
MODEL DATABASE IDENTIFICATION NUMBER: 017T37


LICENSE STATUS

AVAILABLE ANALYSIS SOLUTIONS = LINEAR STATIC
                               PRESTRESS STATIC
                               NONLINEAR STATIC
                               MODAL
                               MODAL COMPLEX EIGENVALUE
                               LINEAR PRESTRESS MODAL
                               NONLINEAR PRESTRESS MODAL
                               LINEAR PRESTRESS COMPLEX EIGENVALUE
                               NONLINEAR PRESTRESS COMPLEX EIGENVALUE
                               LINEAR BUCKLING
                               NONLINEAR BUCKLING
                               MODAL TRANSIENT RESPONSE
                               DIRECT TRANSIENT RESPONSE
                               NONLINEAR TRANSIENT RESPONSE
                               LINEAR PRESTRESS TRANSIENT RESPONSE
                               NONLINEAR PRESTRESS TRANSIENT RESPONSE
                               MODAL FREQUENCY RESPONSE
                               DIRECT FREQUENCY RESPONSE
                               LINEAR PRESTRESS FREQUENCY RESPONSE
                               NONLINEAR PRESTRESS FREQUENCY RESPONSE
                               LINEAR STEADY STATE HEAT TRANSFER
                               NONLINEAR STEADY STATE HEAT TRANSFER
                               NONLINEAR TRANSIENT HEAT TRANSFER
                               MATRIX REDUCTION

REMAINING SOLUTION LICENSE TIME = 205 DAYS


AVAILABLE RESULTS TRANSLATORS = NORAN BINARY
                                NORAN ASCII
                                PATRAN BINARY
                                PATRAN ASCII
                                FEMAP BINARY
                                FEMAP ASCII
                                NASTRAN BINARY
                                NASTRAN XDB

REMAINING TRANSLATOR LICENSE TIME = 205 DAYS


AVAILABLE ANALYSIS FEATURES = MULTIAXIAL FATIGUE
                              DYNAMIC FATIGUE
                              VIBRATION FATIGUE
                              PROGRESSIVE PLY FAILURE
                              MCT PLY FAILURE

REMAINING FEATURE LICENSE TIME = 205 DAYS


MAXIMUM MODEL SIZE = NO LIMITS
```

Listing 2-5. Example System Log File. (Continued)

```
OPERATING SYSTEM = WINDOWS XP V5.01 BUILD:2600 SERVICE PACK:2
CPU TYPE         = XEON
CPU SPEED        = 2806 MHZ
INSTALLED MEMORY = 2046 MEGABYTES


VIRTUAL MEMORY =  536870911 WORDS     4096.0 MEGABYTES
REAL MEMORY    =  177991168 WORDS     1358.0 MEGABYTES

DRIVE M: DISK SPACE =   656016384 WORDS     5005.0 MEGABYTES
DRIVE L: DISK SPACE =  1982897152 WORDS    15128.3 MEGABYTES

GENERATING DATABASE:  TEST.NDB

14 CASE CONTROL COMMANDS WERE RECOGNIZED OF A TOTAL OF 14 READ IN

34 BULK DATA ENTRIES WERE RECOGNIZED OF A TOTAL OF 34 READ IN
MODEL DATABASE SIZE

SUBCASES                     =        1
GRID POINTS                  =       18
ELEMENTS                     =       10
ELEMENT PROPERTIES           =        1
MATERIAL PROPERTIES          =        1
SINGLE POINT CONSTRAINTS     =        3
GRID POINT FORCES            =        3
SETS                         =        1
SURFACES                     =        1

MODULE SEQUENCE FOR SOLUTION: LINEAR STATIC

MODEL SIZE    =        108 DEGREES OF FREEDOM
MATRIX SIZE   =       2071 WORDS        0.0 MEGABYTES
SEMIBANDWIDTH =         47 WORDS

AVAILABLE VIRTUAL MEMORY  =  536870911 WORDS     4096.0 MEGABYTES
AVAILABLE PHYSICAL MEMORY =  175752192 WORDS     1340.9 MEGABYTES

DRIVE M: DISK SPACE =   656008192 WORDS     5004.9 MEGABYTES
DRIVE L: DISK SPACE =  1980836864 WORDS    15112.6 MEGABYTES

ELEMENT GEOMETRY STATISTICS

MAXIMUM QUAD ELEMENT WARPING ANGLE =   0.00 DEGREES ON ELEMENT 10
MAXIMUM QUAD ELEMENT SKEW ANGLE    =   0.00 DEGREES ON ELEMENT 10
MAXIMUM QUAD ELEMENT TAPER RATIO   =   0.00         ON ELEMENT 10
MAXIMUM QUAD ELEMENT ASPECT RATIO  =   2.00         ON ELEMENT 10

GLOBAL STIFFNESS MATRIX ASSEMBLY STATISTICS

SPARSE MATRIX SIZE =       2071 WORDS        0.0 MEGABYTES
MEMORY ALLOCATED   =       3106 WORDS        0.0 MEGABYTES

MAXIMUM GLOBAL STIFFNESS MATRIX TERM ZEROED =  2.3686E-16
MINIMUM GLOBAL STIFFNESS MATRIX TERM ZEROED =  0.0000E+00
REDUCTION IN GLOBAL STIFFNESS MATRIX SIZE   =  59.91 PERCENT

ASSEMBLY TIME FOR 10 ELEMENTS = 0.0 SECONDS

SOLUTION SEQUENCE FOR SUBCASE 1

MAXIMUM STIFFNESS MATRIX DIAGONAL =  3.0598E+06  AT GRID 14 COMPONENT 2
MINIMUM STIFFNESS MATRIX DIAGONAL =  1.5359E+04  AT GRID 1  COMPONENT 5

AVAILABLE VIRTUAL MEMORY  =  536870911 WORDS     4096.0 MEGABYTES
AVAILABLE PHYSICAL MEMORY =  175667712 WORDS     1340.2 MEGABYTES

DRIVE M: DISK SPACE =   656008192 WORDS     5004.9 MEGABYTES
DRIVE L: DISK SPACE =  1980829696 WORDS    15112.5 MEGABYTES
```

Listing 2-5. Example System Log File. (Continued)

```
GLOBAL STIFFNESS MATRIX FACTORIZATION STATISTICS

NUMBER OF NEGATIVE TERMS ON FACTOR DIAGONAL = 0
MAXIMUM MATRIX FACTOR DIAGONAL RATIO = 3.829E+03 AT GRID 15 COMPONENT 3

REORDERING METHOD REQUESTED = AUTO
REORDERING METHOD USED      = VRM1

FACTORED SPARSE MATRIX SIZE =       2304 WORDS        0.0 MEGABYTES
ADDITIONAL MEMORY ALLOCATED =     255999 WORDS        2.0 MEGABYTES

FACTORIZATION TIME FOR 2304 WORDS = 0.0 SECONDS

MAXIMUM DISPLACEMENT MAGNITUDE =  3.072174E-02  AT GRID 1
MAXIMUM ROTATION MAGNITUDE     =  4.607089E-03  AT GRID 1

EPSILON       =  8.367656E-14
STRAIN ENERGY =  9.114923E-01

SOLUTION TIME FOR 108 DEGREES OF FREEDOM = 0.0 SECONDS

MAXIMUM SINGLE POINT CONSTRAINT FORCE MAGNITUDE  =  3.024285E+02  AT GRID 18
MAXIMUM SINGLE POINT CONSTRAINT MOMENT MAGNITUDE =  5.408843E+01  AT GRID 17

CALCULATING RESULTS FOR SUBCASE 1

CALCULATING QUAD ELEMENT RESULTS ON SURFACE 1 FOR SUBCASE 1

MAXIMUM QUAD ELEMENT PRINCIPAL STRESS =  8.116895E+03  AT ELEMENT 1
MINIMUM QUAD ELEMENT PRINCIPAL STRESS = -8.116895E+03  AT ELEMENT 6
MAXIMUM QUAD ELEMENT SHEAR STRESS     =  4.283409E+03  AT ELEMENT 6
MAXIMUM QUAD ELEMENT VON MISES STRESS =  8.350951E+03  AT ELEMENT 6

DATABASE STORAGE STATISTICS

MODEL TRANSLATION DATA        = MEMORY
GRID POINT COORDINATE DATA    = MEMORY
ELEMENT DIRECTION COSINE DATA = MEMORY
GRID POINT LOAD DATA          = MEMORY
ELEMENT LOAD DATA             = MEMORY
ELEMENT RESULTS DATA          = MEMORY
MODAL VECTOR DATA             = MEMORY

MODEL ANALYSIS TIME SUMMARY

TOTAL CPU TIME = 0.9 SECONDS
WALLCLOCK TIME = 1.0 SECONDS

EXECUTION TERMINATED NORMALLY

TOTAL WARNINGS     = 0
TOTAL FATAL ERRORS = 0
```

2.2.4 Model Status File

The Model Status File is similar to the System Log File except that it contains all solution status and results summary information. The Model Initialization directive, `MODLSTATUS`, controls this file. All real time displayed status information is written to this file when `MODLSTATUS` is set to `FILE` or `BOTH`.

Listing 2-6. Example Model Status File.

```
ADS NASTRAN VERSION 10.3.0.716   12:27  01/16/15
MODEL DATABASE IDENTIFICATION NUMBER: 017T37


INITIALIZATION PROCESSOR MODULE


LICENSE STATUS

READING IN FILE:  NASTRAN.INI

PROCESSING MODEL INITIALIZATION FILE

DELETING FILE:  TEST.FNO

VIRTUAL MEMORY =  536870911 WORDS     4096.0 MEGABYTES
REAL MEMORY    =  182401536 WORDS     1391.6 MEGABYTES

DRIVE M: DISK SPACE =   656016384 WORDS     5005.0 MEGABYTES
DRIVE L: DISK SPACE =  1981538304 WORDS    15117.9 MEGABYTES

WRITING MODEL RESULTS OUTPUT TO FILE:  TEST.OUT

WRITING OUT MODEL INITIALIZATION DIRECTIVES
PAGES WRITTEN:      2


MODEL TRANSLATOR MODULE


GENERATING DATABASE:  TEST.NDB

READING IN FILE:  TEST.NAS
LINES READ:       49

DETERMINING MODEL SIZE
PERCENT COMPLETE:  100

PROCESSING CASE CONTROL COMMANDS
PERCENT COMPLETE:  100

14 CASE CONTROL COMMANDS WERE RECOGNIZED OF A TOTAL OF 14 READ IN

PROCESSING BULK DATA ENTRIES
PERCENT COMPLETE:  100

34 BULK DATA ENTRIES WERE RECOGNIZED OF A TOTAL OF 34 READ IN

GENERATING CASE CONTROL COMMANDS
PERCENT COMPLETE:  100

GENERATING BULK DATA ENTRIES
PERCENT COMPLETE:  100

WRITING CASE CONTROL COMMANDS TO FILE:  TEST.BDF
LINES WRITTEN:       16

WRITING BULK DATA ENTRIES TO FILE:  TEST.BDF
LINES WRITTEN:       42
```

Listing 2-6. Example Model Status File. (Continued)

```
MODEL DATABASE SIZE

SUBCASES                     =        1
GRID POINTS                  =       18
ELEMENTS                     =       10
ELEMENT PROPERTIES           =        1
MATERIAL PROPERTIES          =        1
SINGLE POINT CONSTRAINTS     =        3
GRID POINT FORCES            =        3
SETS                         =        1
SURFACES                     =        1


MODULE SEQUENCE FOR SOLUTION: LINEAR STATIC


GEOMETRY PROCESSOR MODULE


INITIALIZING SPARSE STORAGE
PERCENT COMPLETE:  100

MODEL SIZE    =        108 DEGREES OF FREEDOM
MATRIX SIZE   =       2070 WORDS        0.0 MEGABYTES
SEMIBANDWIDTH =         47 WORDS

AVAILABLE VIRTUAL MEMORY  =   65494528 WORDS      499.7 MEGABYTES
AVAILABLE PHYSICAL MEMORY =   32453632 WORDS      247.6 MEGABYTES

DRIVE D: DISK SPACE =   15756288 WORDS      120.2 MEGABYTES

ASSEMBLING GLOBAL STIFFNESS MATRIX
PERCENT COMPLETE:  100

ELEMENT GEOMETRY STATISTICS

MAXIMUM QUAD ELEMENT WARPING ANGLE =   0.00 DEGREES ON ELEMENT 10
MAXIMUM QUAD ELEMENT SKEW ANGLE    =   0.00 DEGREES ON ELEMENT 10
MAXIMUM QUAD ELEMENT TAPER RATIO   =   0.00         ON ELEMENT 10
MAXIMUM QUAD ELEMENT ASPECT RATIO  =   2.00         ON ELEMENT 10

GLOBAL STIFFNESS MATRIX ASSEMBLY STATISTICS

SPARSE MATRIX SIZE =       1962 WORDS        0.0 MEGABYTES
MEMORY ALLOCATED   =       3105 WORDS        0.0 MEGABYTES

MAXIMUM GLOBAL STIFFNESS MATRIX TERM ZEROED =  9.9206E-17
MINIMUM GLOBAL STIFFNESS MATRIX TERM ZEROED =  0.0000E+00
REDUCTION IN GLOBAL STIFFNESS MATRIX SIZE   =  58.00 PERCENT

ASSEMBLY TIME FOR 10 ELEMENTS = 0.1 SECONDS


LINEAR SOLUTION PROCESSOR MODULE


SOLUTION SEQUENCE FOR SUBCASE 1

MAXIMUM STIFFNESS MATRIX DIAGONAL =  3.0578E+06  AT GRID 14 COMPONENT 2
MINIMUM STIFFNESS MATRIX DIAGONAL =  1.5359E+04  AT GRID 1  COMPONENT 5

AVAILABLE VIRTUAL MEMORY  =   66522624 WORDS      507.5 MEGABYTES
AVAILABLE PHYSICAL MEMORY =   32373248 WORDS      247.0 MEGABYTES

DRIVE D: DISK SPACE =   15748096 WORDS      120.1 MEGABYTES

FACTORIZING GLOBAL STIFFNESS MATRIX FOR SUBCASE 1
PERCENT COMPLETE:  100
```

Listing 2-6. Example Model Status File. (Continued)

```
GLOBAL STIFFNESS MATRIX FACTORIZATION STATISTICS

NUMBER OF NEGATIVE TERMS ON FACTOR DIAGONAL = 0
MAXIMUM MATRIX FACTOR DIAGONAL RATIO = 3.829E+03 AT GRID 15 COMPONENT 3

REORDERING METHOD REQUESTED = AUTO
REORDERING METHOD USED      = VRM1

FACTORED SPARSE MATRIX SIZE =       2304 WORDS        0.0 MEGABYTES
ADDITIONAL MEMORY ALLOCATED =     255999 WORDS        2.0 MEGABYTES

FACTORIZATION TIME FOR 2304 WORDS = 0.0 SECONDS

ASSEMBLING GLOBAL LOAD VECTOR FOR SUBCASE 1

ASSEMBLING GRID POINT LOADS
PERCENT COMPLETE:  100

SOLVING FOR DISPLACEMENTS FOR SUBCASE 1
PERCENT COMPLETE:  100

MAXIMUM DISPLACEMENT MAGNITUDE =  3.072174E-02  AT GRID 1
MAXIMUM ROTATION MAGNITUDE     =  4.607089E-03  AT GRID 1

EPSILON       =  8.367656E-14
STRAIN ENERGY =  9.114923E-01

SOLUTION TIME FOR 108 DEGREES OF FREEDOM = 0.0 SECONDS

WRITING OUT GRID POINT DISPLACEMENT VECTOR FOR SUBCASE 1
PAGES WRITTEN:      1

MAXIMUM SINGLE POINT CONSTRAINT FORCE MAGNITUDE  =  3.024285E+02  AT GRID 18
MAXIMUM SINGLE POINT CONSTRAINT MOMENT MAGNITUDE =  5.408843E+01  AT GRID 17

WRITING OUT FORCES OF SINGLE POINT CONSTRAINT FOR SUBCASE 1
PAGES WRITTEN:      1


RESULTS PROCESSOR MODULE


CALCULATING RESULTS FOR SUBCASE 1

WRITING GRID POINT DISPLACEMENTS FOR SUBCASE 1 TO FILE: TEST.DIS

WRITING GRID POINT FORCES FOR SUBCASE 1 TO FILE: TEST.GPF

CALCULATING QUAD ELEMENT RESULTS ON SURFACE 1 FOR SUBCASE 1
PERCENT COMPLETE:  100

WRITING OUT FORCES IN QUAD ELEMENTS ON SURFACE 1 FOR SUBCASE 1
PAGES WRITTEN:      1

MAXIMUM QUAD ELEMENT PRINCIPAL STRESS =  8.116895E+03  AT ELEMENT 1
MINIMUM QUAD ELEMENT PRINCIPAL STRESS = -8.116895E+03  AT ELEMENT 6
MAXIMUM QUAD ELEMENT SHEAR STRESS     =  4.283409E+03  AT ELEMENT 6
MAXIMUM QUAD ELEMENT VON MISES STRESS =  8.350951E+03  AT ELEMENT 6

WRITING OUT STRESSES IN QUAD ELEMENTS ON SURFACE 1 FOR SUBCASE 1
PAGES WRITTEN:      3

WRITING ELEMENT RESULTS FOR SUBCASE 1 TO FILE: TEST.ELS

DELETING FILE:  TEST.NDB

GENERATING RESULTS NEUTRAL FILE
PERCENT COMPLETE:  100

DELETING FILE:  TEST.GPF
```

Listing 2-6. Example Model Status File. (Continued)

```
MODEL ANALYSIS TIME SUMMARY

TOTAL CPU TIME = 0.9 SECONDS
WALLCLOCK TIME = 1.0 SECONDS

EXECUTION TERMINATED NORMALLY

TOTAL WARNINGS     = 0
TOTAL FATAL ERRORS = 0
```

2.2.5 Results Neutral Files

The result neutral file system is the primary interface for graphical processing of model results data. The file system is also used for:

- Source of expanded model results output.
- Input file for results limits search via the `RESULTLIMITS` Case Control command.
- Input file for automated `SET` entry generation via the `SETGENERATE` Case Control command.

The results neutral file system consists of eight types of files, each generated by the Results Processor. A specific Model Initialization directive as shown below controls output of each type:

File Type	Model Initialization Directive	Default Neutral Filename
Grid Point Displacement Vector	DISPFILE = [d:] [path] filename[.ext]	*model output filename.DIS*
Grid Point Force Vector	FORCFILE = [d:] [path] filename[.ext]	*model output filename.GPF*
Element Internal Load Vector	LOADFILE = [d:] [path] filename[.ext]	*model output filename.ELF*
Element Results	ELEMFILE = [d:] [path] filename[.ext]	*model output filename.ELS*
Grid Point Results	GRIDFILE = [d:] [path] filename[.ext]	*model output filename.GPS*
FEMAP Results	Defined by DISPFILE	*model output filename.NEU* *model output filename.FNO*
NASTRAN OP2 Results	Defined by DISPFILE	*model output filename.OP2*
NASTRAN XDB Results	Defined by DISPFILE	*model output filename.XDB*
NASTRAN ASCII Results	Defined by MODLOUTFILE	*model output filename.PCH*
MS Excel ASCII Results	Defined by MODLOUTFILE	*model output filename.CSV*

The `DISPFILE`, `FORCFILE`, `LOADFILE`, `ELEMFILE`, and `GRIDFILE` directives control the filenames and whether a file is to be generated. Setting a specific directive equal to the character variable `NONE` will disable output of that neutral file type.

Another useful Model Initialization directive is RSLTFILETYPE which controls file type and format. When `RSLTFILETYPE` is set to `FEMAPASCII` or `FEMAPBINARY`, a single FEMAP® compatible results neutral file of the entire results database is generated. When `RSLTFILETYPE` is set to `PATRANASCII` or `PATRANBINARY`, multiple PATRAN 2.5 compatible results neutral files are generated. PATRAN results neutral files have a two digit subcase number added to the base of the filename to facilitate multiple subcases. When `RSLTFILETYPE` is set to `NASTRANBINARY`, a single NASTRAN Output 2 compatible results file of the entire results database is generated. When `RSLTFILETYPE` is set to `NASTRANXDB`, a single NASTRAN XDB compatible results file of the entire results database is generated.

When `RSLTFILETYPE` is set to FEMAPBINARY and the `INRCRSLTOUT` directive is set to `ON`, a separate FEMAP binary results neutral file will be generated for each load increment or time step. At the end of the analysis a single neutral file with all steps will be generated.

For a detailed description of each directive see Section 2, *Initialization*.

2.2.5.1 Grid Point Displacement Vector Neutral File

The grid point displacement vector neutral file contains the calculated displacement vector at each grid point in the basic coordinate system.

Listing 2-7. Example Grid Point Displacement Vector Neutral File.

```
SUBC 1, 60 LB EDGE LOAD IN Y-DIRECTION
      18      18   0.241367E-02     3     6
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH
INSTALLATION TEST CASE
       10.4514573E-020.3047095E-010.0000000E+000.0000000E+000.0000000E+00
0.4332973E-02
       20.0000000E+000.3045806E-010.0000000E+000.0000000E+000.0000000E+00
0.4018987E-02
       3-.4514573E-020.3047095E-010.0000000E+000.0000000E+000.0000000E+00
0.4332973E-02
       40.1584370E-020.1820868E-020.0000000E+000.0000000E+000.0000000E+00
0.1424074E-02
       50.0000000E+000.1659872E-020.0000000E+000.0000000E+000.0000000E+00
0.1507010E-02
       6-.1584370E-020.1820868E-020.0000000E+000.0000000E+000.0000000E+00
0.1424074E-02
       70.3792008E-020.1321103E-010.0000000E+000.0000000E+000.0000000E+00
0.3666624E-02
       80.0000000E+000.1315702E-010.0000000E+000.0000000E+000.0000000E+00
0.3348591E-02
       9-.3792008E-020.1321103E-010.0000000E+000.0000000E+000.0000000E+00
0.3666624E-02
      100.2877639E-020.6417065E-020.0000000E+000.0000000E+000.0000000E+00
0.2774258E-02
      110.0000000E+000.6299692E-020.0000000E+000.0000000E+000.0000000E+00
0.2549374E-02
      12-.2877639E-020.6417065E-020.0000000E+000.0000000E+000.0000000E+00
0.2774258E-02
      130.4330950E-020.2147459E-010.0000000E+000.0000000E+000.0000000E+00
0.4151929E-02
      140.0000000E+000.2145137E-010.0000000E+000.0000000E+000.0000000E+00
0.3860329E-02
      15-.4330950E-020.2147459E-010.0000000E+000.0000000E+000.0000000E+00
0.4151929E-02
      160.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
      170.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
      180.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
```

2.2.5.2 Grid Point Force Vector Neutral File

The grid point force vector neutral file contains the calculated internal, applied and reacted force vector at each grid point in the basic coordinate system. The internal force vector is the resultant of all internal forces at the grid point. For transient response analysis, acceleration and velocity is also included in this file.

Listing 2-8. Example Grid Point Force Vector Neutral File.

```
SUBC 1, 60 LB EDGE LOAD IN Y-DIRECTION
      18       0   0.000000E+00     0    36
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH
INSTALLATION TEST CASE
       10.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.1500000E+020.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.1500000E+02
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
       20.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.3000000E+020.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.3000000E+02
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
       30.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.1500000E+020.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.1500000E+02
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
       40.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
       50.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
       60.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
```

Listing 2-8. Example Grid Point Force Vector Neutral File. (Continued)

```
       70.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
       80.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
       90.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
      100.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
      110.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
      120.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
      130.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
      140.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
```

Listing 2-8. Example Grid Point Force Vector Neutral File. (Continued)

```
      150.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
      160.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+00-.3000000E+03-.6983513E+020.0000000E+00
0.0000000E+000.0000000E+000.2430190E+020.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
      170.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+00-.1138289E-100.7967025E+020.0000000E+00
0.0000000E+000.0000000E+00-.4860379E+020.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
      180.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.3000000E+03-.6983513E+020.0000000E+00
0.0000000E+000.0000000E+000.2430190E+020.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+00
```

2.2.5.3 Element Results Neutral File

The element results neutral file contains various result types calculated at requested points on the element in a user specified coordinate system. The coordinate system for shell element results is specified using the Case Control command `SURFACE` and solid element results using the Case Control command `VOLUME`. Shell and solid elements that do not have a coordinate system defined via a `SURFACE` or `VOLUME` command will not be included.

Listing 2-9. Example Element Results Neutral File.

```
SUBC 1, 60 LB EDGE LOAD IN Y-DIRECTION
   40
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH
INSTALLATION TEST CASE
       1       5
0.0000000E+000.0000000E+000.0000000E+000.4079692E+040.2807025E+030.3000000E+03
0.4487487E+010.1923039E+040.4103236E+040.2571579E+030.3980892E+04-.5000000E-01
0.4079692E+040.2807025E+030.3000000E+030.4487487E+010.1923039E+040.4103236E+04
0.2571579E+030.3980892E+040.5000000E-010.3980892E+040.1923039E+040.4103236E+04
0.2571579E+030.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.4079692E+030.2807025E+020.3000000E+020.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+00
```

Listing 2-9. Example Element Results Neutral File. (Continued)

```
       2       5
0.0000000E+000.0000000E+000.0000000E+000.3122276E+040.5787233E+020.3000000E+03
0.5539088E+010.1561295E+040.3151369E+040.2877904E+020.3137079E+04-.5000000E-01
0.3122276E+040.5787233E+020.3000000E+030.5539088E+010.1561295E+040.3151369E+04
0.2877904E+020.3137079E+040.5000000E-010.3137079E+040.1561295E+040.3151369E+04
0.2877904E+020.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.3122276E+030.5787233E+010.3000000E+020.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+00
       5       5
0.0000000E+000.0000000E+000.0000000E+000.4469817E+03-.9152188E+010.3000000E+03
0.2637853E+020.3768481E+030.5957629E+03-.1579334E+030.6884528E+03-.5000000E-01
0.4469817E+03-.9152188E+010.3000000E+030.2637853E+020.3768481E+030.5957629E+03
-.1579334E+030.6884528E+030.5000000E-010.6884528E+030.3768481E+030.5957629E+03
-.1579334E+030.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.4469817E+02-.9152189E+000.3000000E+020.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+00
       6       5
0.0000000E+000.0000000E+000.0000000E+00-.4079692E+04-.2807025E+030.3000000E+03
0.8551251E+020.1923039E+04-.2571579E+03-.4103236E+040.3980892E+04-.5000000E-01
-.4079692E+04-.2807025E+030.3000000E+030.8551251E+020.1923039E+04-.2571579E+03
-.4103236E+040.3980892E+040.5000000E-010.3980892E+040.1923039E+04-.2571579E+03
-.4103236E+040.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+00-.4079692E+03-.2807025E+020.3000000E+020.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+00
       7       5
0.0000000E+000.0000000E+000.0000000E+00-.3122276E+04-.5787233E+020.3000000E+03
0.8446091E+020.1561295E+04-.2877904E+02-.3151369E+040.3137079E+04-.5000000E-01
-.3122276E+04-.5787233E+020.3000000E+030.8446091E+020.1561295E+04-.2877904E+02
-.3151369E+040.3137079E+040.5000000E-010.3137079E+040.1561295E+04-.2877904E+02
-.3151369E+040.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+00-.3122276E+03-.5787233E+010.3000000E+020.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+00
       8       5
0.0000000E+000.0000000E+000.0000000E+00-.2238026E+040.3028394E+020.3000000E+03
0.8259190E+020.1173161E+040.6929031E+02-.2277032E+040.2312456E+04-.5000000E-01
-.2238026E+040.3028394E+020.3000000E+030.8259190E+020.1173161E+040.6929031E+02
-.2277032E+040.2312456E+040.5000000E-010.2312456E+040.1173161E+040.6929031E+02
-.2277032E+040.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+00-.2238026E+030.3028394E+010.3000000E+020.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+00
       9       5
0.0000000E+000.0000000E+000.0000000E+00-.1350881E+040.2496305E+020.3000000E+03
0.7821909E+020.7504910E+030.8753200E+02-.1413450E+040.1459187E+04-.5000000E-01
-.1350881E+040.2496305E+020.3000000E+030.7821909E+020.7504910E+030.8753200E+02
-.1413450E+040.1459187E+040.5000000E-010.1459187E+040.7504910E+030.8753200E+02
-.1413450E+040.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+00-.1350881E+030.2496305E+010.3000000E+020.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+00
      10       5
0.0000000E+000.0000000E+000.0000000E+00-.4469817E+030.9152188E+010.3000000E+03
0.6362147E+020.3768481E+030.1579334E+03-.5957629E+030.6884528E+03-.5000000E-01
-.4469817E+030.9152188E+010.3000000E+030.6362147E+020.3768481E+030.1579334E+03
-.5957629E+030.6884528E+030.5000000E-010.6884528E+030.3768481E+030.1579334E+03
-.5957629E+030.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+00-.4469817E+020.9152189E+000.3000000E+020.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+00
```

2.2.5.4 Grid Point Results Neutral File

The grid point results neutral file contains various result types calculated at the grid points in a user-specified coordinate system. The coordinate system for shell element results is specified using the Case Control command `SURFACE` and solid element results using the Case Control command `VOLUME`. Shell and solid elements that do not have a coordinate system defined via a `SURFACE` or `VOLUME` command will not be included.

Listing 2-10. Example Grid Point Results Neutral File.

```
SUBC 1, 60 LB EDGE LOAD IN Y-DIRECTION
       18         0   0.000000E+00          0       40
2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH
INSTALLATION TEST CASE
       1        5
0.0000000E+000.0000000E+000.0000000E+000.4224462E+03-.1025932E+030.5820478E+02
0.6250578E+010.2688948E+030.4288212E+03-.1089683E+030.4924324E+03-.5000000E-01
0.4224462E+03-.1025932E+030.5820478E+020.6250578E+010.2688948E+030.4288212E+03
-.1089683E+030.4924324E+030.5000000E-010.4924324E+030.2688948E+030.4288212E+03
-.1089683E+030.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+00
       2        5
0.0000000E+000.0000000E+000.0000000E+000.2131628E-10-.1230731E-090.5417952E+03
0.4500000E+020.5417952E+030.5417952E+03-.5417952E+030.9384169E+03-.5000000E-01
0.2131628E-10-.1230731E-090.5417952E+030.4500000E+020.5417952E+030.5417952E+03
-.5417952E+030.9384169E+030.5000000E-010.9384169E+030.5417952E+030.5417952E+03
-.5417952E+030.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.3504872E+030.7254437E+020.2524206E-100.2066430E+030.3504872E+03
0.7254437E+020.2524206E-100.2066430E+030.2066430E+03
       3        5
0.0000000E+000.0000000E+000.0000000E+00-.4224462E+030.1025932E+030.5820478E+02
0.8374942E+020.2688948E+030.1089683E+03-.4288212E+030.4924324E+03-.5000000E-01
-.4224462E+030.1025932E+030.5820478E+020.8374942E+020.2688948E+030.1089683E+03
-.4288212E+030.4924324E+030.5000000E-010.4924324E+030.2688948E+030.1089683E+03
-.4288212E+030.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+00
       4        5
0.0000000E+000.0000000E+000.0000000E+000.7271734E+04-.3201493E+030.2834762E+03
0.2135425E+010.3806512E+040.7282304E+04-.3307195E+030.7453169E+04-.5000000E-01
0.7271734E+04-.3201493E+030.2834762E+030.2135425E+010.3806512E+040.7282304E+04
-.3307195E+030.7453169E+040.5000000E-010.7453169E+040.3806512E+040.7282304E+04
-.3307195E+030.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.5724287E+030.1357157E+030.2605754E+020.3399865E+030.5724287E+03
0.1357157E+030.2605754E+020.3399865E+030.3399865E+03
       5        5
0.0000000E+000.0000000E+000.0000000E+000.1291554E-09-.1490719E-100.3165238E+03
0.4500000E+020.3165238E+030.3165238E+03-.3165238E+030.5482354E+03-.5000000E-01
0.1291554E-09-.1490719E-100.3165238E+030.4500000E+020.3165238E+030.3165238E+03
-.3165238E+030.5482354E+030.5000000E-010.5482354E+030.3165238E+030.3165238E+03
-.3165238E+030.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.5638306E+020.1866367E+030.1842547E+020.1130661E+030.5638306E+02
0.1866367E+030.1842547E+020.1130661E+030.1130661E+03
       6        5
0.0000000E+000.0000000E+000.0000000E+00-.7271734E+040.3201493E+030.2834762E+03
0.8786458E+020.3806512E+040.3307195E+03-.7282304E+040.7453169E+04-.5000000E-01
-.7271734E+040.3201493E+030.2834762E+030.8786458E+020.3806512E+040.3307195E+03
-.7282304E+040.7453169E+040.5000000E-010.7453169E+040.3806512E+040.3307195E+03
-.7282304E+040.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.5724287E+030.1357157E+030.2605754E+020.3399865E+030.5724287E+03
0.1357157E+030.2605754E+020.3399865E+030.3399865E+03
```

Listing 2-10. Example Grid Point Results Neutral File. (Continued)

```
       7       5
0.0000000E+000.0000000E+000.0000000E+000.3640432E+040.6258159E+020.7289297E+02
0.1166665E+010.1790409E+040.3641916E+040.6109713E+020.3611755E+04-.5000000E-01
0.3640432E+040.6258159E+020.7289297E+020.1166665E+010.1790409E+040.3641916E+04
0.6109713E+020.3611755E+040.5000000E-010.3611755E+040.1790409E+040.3641916E+04
0.6109713E+020.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.3243085E+030.6820667E+020.1037270E+020.1914295E+030.3243085E+03
0.6820667E+020.1037270E+020.1914295E+030.1914295E+03

       8       5
0.0000000E+000.0000000E+000.0000000E+000.1112426E-090.3879563E-110.5271071E+03
0.4500000E+020.5271071E+030.5271071E+03-.5271071E+030.9129761E+03-.5000000E-01
0.1112426E-090.3879563E-110.5271071E+030.4500000E+020.5271071E+030.5271071E+03
-.5271071E+030.9129761E+030.5000000E-010.9129761E+030.5271071E+030.5271071E+03
-.5271071E+030.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.2399892E+030.5749077E+020.7334604E+010.1425410E+030.2399892E+03
0.5749077E+020.7334604E+010.1425410E+030.1425410E+03
       9       5
0.0000000E+000.0000000E+000.0000000E+00-.3640432E+04-.6258159E+020.7289297E+02
0.8883334E+020.1790409E+04-.6109713E+02-.3641916E+040.3611755E+04-.5000000E-01
-.3640432E+04-.6258159E+020.7289297E+020.8883334E+020.1790409E+04-.6109713E+02
-.3641916E+040.3611755E+040.5000000E-010.3611755E+040.1790409E+04-.6109713E+02
-.3641916E+040.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.3243085E+030.6820667E+020.1037270E+020.1914295E+030.3243085E+03
0.6820667E+020.1037270E+020.1914295E+030.1914295E+03
      10       5
0.0000000E+000.0000000E+000.0000000E+000.5598761E+040.1217738E+020.1670937E+03
0.1711667E+010.2798285E+040.5603755E+040.7184099E+010.5600166E+04-.5000000E-01
0.5598761E+040.1217738E+020.1670937E+030.1711667E+010.2798285E+040.5603755E+04
0.7184099E+010.5600166E+040.5000000E-010.5600166E+040.2798285E+040.5603755E+04
0.7184099E+010.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.4595750E+030.1638972E+030.5623728E+020.2835690E+030.4595750E+03
0.1638972E+030.5623728E+020.2835690E+030.2835690E+03
      11       5
0.0000000E+000.0000000E+000.0000000E+000.1131468E-09-.8577672E-100.4329063E+03
0.4500000E+020.4329063E+030.4329063E+03-.4329063E+030.7498157E+03-.5000000E-01
0.1131468E-09-.8577672E-100.4329063E+030.4500000E+020.4329063E+030.4329063E+03
-.4329063E+030.7498157E+030.5000000E-010.7498157E+030.4329063E+030.4329063E+03
-.4329063E+030.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.1539003E+030.1160527E+030.3976577E+020.1136293E+030.1539003E+03
0.1160527E+030.3976577E+020.1136293E+030.1136293E+03
      12       5
0.0000000E+000.0000000E+000.0000000E+00-.5598761E+04-.1217738E+020.1670937E+03
0.8828833E+020.2798285E+04-.7184099E+01-.5603755E+040.5600166E+04-.5000000E-01
-.5598761E+04-.1217738E+020.1670937E+030.8828833E+020.2798285E+04-.7184099E+01
-.5603755E+040.5600166E+040.5000000E-010.5600166E+040.2798285E+04-.7184099E+01
-.5603755E+040.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.4595750E+030.1638972E+030.5623728E+020.2835690E+030.4595750E+03
0.1638972E+030.5623728E+020.2835690E+030.2835690E+03
      13       5
0.0000000E+000.0000000E+000.0000000E+000.1802156E+040.3411998E+020.5821427E+02
0.1883798E+010.8859327E+030.1804071E+040.3220530E+020.1788186E+04-.5000000E-01
0.1802156E+040.3411998E+020.5821427E+020.1883798E+010.8859327E+030.1804071E+04
0.3220530E+020.1788186E+040.5000000E-010.1788186E+040.8859327E+030.1804071E+04
0.3220530E+020.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.2917021E+030.3547475E+020.6713005E-020.1696551E+030.2917021E+03
0.3547475E+020.6713005E-020.1696551E+030.1696551E+03
      14       5
0.0000000E+000.0000000E+000.0000000E+000.9789858E-100.1101910E-090.5417857E+03
0.4500000E+020.5417857E+030.5417857E+03-.5417857E+030.9384004E+03-.5000000E-01
0.9789858E-100.1101910E-090.5417857E+030.4500000E+020.5417857E+030.5417857E+03
-.5417857E+030.9384004E+030.5000000E-010.9384004E+030.5417857E+030.5417857E+03
-.5417857E+030.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.2378958E+030.3033599E+020.4746811E-020.1384614E+030.2378958E+03
0.3033599E+020.4746811E-020.1384614E+030.1384614E+03
```

Listing 2-10. Example Grid Point Results Neutral File. (Continued)

```
     15      5
0.0000000E+000.0000000E+000.0000000E+00-.1802156E+04-.3411998E+020.5821427E+02
0.8811620E+020.8859327E+03-.3220530E+02-.1804071E+040.1788186E+04-.5000000E-01
-.1802156E+04-.3411998E+020.5821427E+020.8811620E+020.8859327E+03-.3220530E+02
-.1804071E+040.1788186E+040.5000000E-010.1788186E+040.8859327E+03-.3220530E+02
-.1804071E+040.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.2917021E+030.3547475E+020.6713005E-020.1696551E+030.2917021E+03
0.3547475E+020.6713005E-020.1696551E+030.1696551E+03
     16      5
0.0000000E+000.0000000E+000.0000000E+000.7999962E+040.1073485E+040.3203271E+03
0.2642226E+010.3478021E+040.8014745E+040.1058703E+040.7541336E+04-.5000000E-01
0.7999962E+040.1073485E+040.3203271E+030.2642226E+010.3478021E+040.8014745E+04
0.1058703E+040.7541336E+040.5000000E-010.7541336E+040.3478021E+040.8014745E+04
0.1058703E+040.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+00
      17      5
0.0000000E+000.0000000E+000.0000000E+000.1239471E-090.3637979E-100.2796729E+03
0.4500000E+020.2796729E+030.2796729E+03-.2796729E+030.4844077E+03-.5000000E-01
0.1239471E-090.3637979E-100.2796729E+030.4500000E+020.2796729E+030.2796729E+03
-.2796729E+030.4844077E+030.5000000E-010.4844077E+030.2796729E+030.2796729E+03
-.2796729E+030.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.5523426E+020.7590687E+030.4662546E-110.4394072E+030.5523426E+02
0.7590687E+030.4662546E-110.4394072E+030.4394072E+03
      18      5
0.0000000E+000.0000000E+000.0000000E+00-.7999962E+04-.1073485E+040.3203271E+03
0.8735777E+020.3478021E+04-.1058703E+04-.8014745E+040.7541336E+04-.5000000E-01
-.7999962E+04-.1073485E+040.3203271E+030.8735777E+020.3478021E+04-.1058703E+04
-.8014745E+040.7541336E+040.5000000E-010.7541336E+040.3478021E+04-.1058703E+04
-.8014745E+040.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+000.0000000E+00
0.0000000E+000.0000000E+000.0000000E+000.0000000E+00
```

3. LINEAR BUCKLING ANALYSIS

3.1 Introduction

Of principal interest in buckling analysis of structures is the critical static load or combination of loads that results in instability. Buckling occurs when a member or structure under an applied loading converts membrane strain energy into strain energy of bending. At this critical load, the structure will continue to deflect without an increase in the magnitude of the loading. The magnitude of the critical load generally depends on the geometric dimensions of the structure, the method in which the structure is stiffened and supported, and the bending and extensional stiffnesses of the various components.

Autodesk Nastran determines this critical load by solving the eigenvalue problem:

$$\left| [K_l] + \lambda [K_s] \right| [\phi] = 0$$

where,

$[K_l]$	is the global linear stiffness matrix
$[K_s]$	is the global differential or initial stress stiffness matrix
λ_i	is the eigenvalue for each mode that when multiplied by the applied loading gives the critical loading P_{cr}
ϕ_i	is the eigenvector for each mode that represent the buckled mode shape

In solving the above eigenvalue problem there are as many eigenvalues and corresponding eigenvectors as there are unconstrained degrees of freedom. Often, however, only the lowest buckling mode is of practical interest. This will *always* be the first mode extracted.

3.2 How to Setup a Model Input File for Linear Buckling Analysis

In Autodesk Nastran you can solve a linear buckling problem by setting `SOLUTION = BUCKLING` in the Model Initialization File or by specifying `SOL 105` or `SOL BUCKLING` above the Case Control Section in the Model Input File, and following the procedure listed below:

1. Apply static loads to the first subcase. This subcase will be treated as a static run. The applied loading will generate internal loads that are used to formulate the differential stiffness or differential stiffness matrix.
2. The second to n subcases must also reference an `EIGRL` Bulk Data entry via the `METHOD` Case Control command. Here, n is equal to the number of buckling analyses that you want to run. Each buckling subcase may call out a unique eigenvalue solution.
3. The differential stiffness matrix is automatically generated for each element that supports differential stiffness. Elements that support differential stiffness are: `CPIPE`, `CCABLE`, `CROD`, `CBAR`, `CBEAM`, `CQUAD4`, `CQUADR`, `CTRIA3`, `CTRIAR`, `CHEXA`, `CPENTA`, `CPYRA`, and `CTETRA`.
4. You must then multiply the eigenvalues obtained in Step 2 by the appropriate applied loads to obtain the buckling loads for each buckling analysis.
5. Each subcase may have a different boundary condition; however, the global differential stiffness matrix will be based on the boundary conditions specified in the first subcase.

3.3 Interpreting Results

As an example we will use the classical Euler beam buckling problem shown in Figure 3-1. It is desired to find the lowest load at which instability occurs. Listing 3-1 contains the Model Input File and Listings 3-2 and 3-3 show the extracted eigenvalues and eigenvectors from the Model Results Output File. The mode shapes are plotted in Figure 3-2.

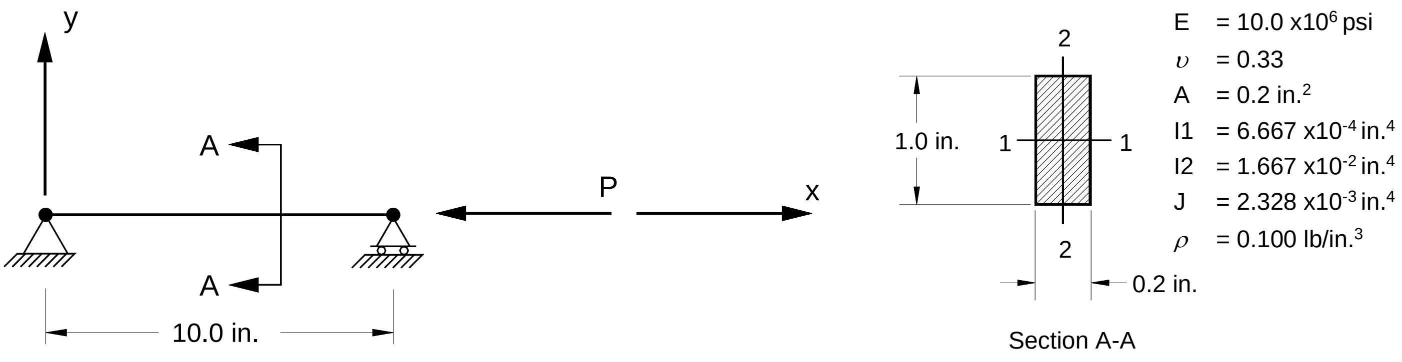


Figure 3-1. Euler Beam Example Problem -Pinned at Both Ends.

Listing 3-1. Model Input File for the Classic Euler Buckling Problem.

```
$
$ BUCKLING SOLUTION.
$
SOL BUCKLING
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = EULER BEAM BUCKLING -BAR ELEMENTS -1X5 MESH
$
DISPLACEMENT = ALL
$
SPC = 1
SUBCASE 1
 LABEL = STATIC, COMPRESSIVE LOAD IN X-DIR
 LOAD = 1
 STRESS = ALL
SUBCASE 2
 LABEL = BUCKLING, COMPRESSIVE LOAD IN X-DIR
 STRESS = NONE
 METHOD = 1
BEGIN BULK
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  5,  ,  ,
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 5 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES.
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3,
,  -0.1,  0.5,  0.1,  0.5,  -0.1,  -0.5,  0.1,  -0.5

$
$ ELEMENT MATERIAL PROPERTIES.
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ PINNED AT BOTH ENDS -ONE END FREE TO TRANSLATE IN X-DIR.
$
SPC1,  1,  123,   1
SPC1,  1,   23,  11
SPC1,  1,    4,   1,  THRU,  11
$
$ COMPRESSIVE LOAD (X-DIRECTION).
$
FORCE,  1,  11,  0,  1000.,  -1.,  0.,  0.
ENDDATA
```

The `EIGRL` entry controls the range and number of modes extracted. Here, we have requested 5 modes as shown in Listing 3-1 through 3-3, and Figure 3-2. The eigenvalues are always sorted in increasing order. Thus, the first mode is always the lowest. The eigenvalue for the first mode is equal to 0.6850, while the applied load in subcase 1 is equal to -1000 pounds. The lowest buckling load is then equal to:

$$P_{cr_1} = \lambda_1 P_a = (0.6580)(-1000) = -658.0 \text{ pounds}$$

Listing 3-2. Extracted Eigenvalues for an Euler Beam.

```
BUCKLING, COMPRESSIVE LOAD IN X-DIR, PINNED-PINNED                  SUBCASE 2

                                                 R E A L   E I G E N V A L U E S

      MODE         EIGENVALUE       RADIANS          CYCLES       GENERALIZED     GENERALIZED     ORTHOGONALITY      ERROR
     NUMBER                                                          MASS          STIFFNESS          LOSS          MEASURE
        1         6.580154E-01    8.111815E-01    1.291035E-01    4.934669E+02    3.247088E+02    0.000000E+00    2.402446E-14
        2         2.632584E+00    1.622524E+00    2.582327E-01    2.181383E+03    5.742674E+03    1.265741E-14    2.403939E-11
        3         5.928265E+00    2.434803E+00    3.875109E-01    4.431893E+03    2.627344E+04    1.039099E-15    4.084352E-09
        4         1.056186E+01    3.249901E+00    5.172378E-01    4.971708E+03    5.251047E+04    1.026956E-15    2.123925E-07
        5         1.645285E+01    4.056212E+00    6.455661E-01    4.934675E+02    8.118948E+03    6.931880E-18    8.382289E-06
```

Note that buckling will occur first in the xz-plane. The reason is that the bending stiffness (*EI*) in this plane is lower than in the xy-plane. The lowest buckling load for the xy-plane is then equal to:

$$P_{cr_5} = \lambda_5 P_a = (16.4529)(-1000) = -16452.9 \text{ pounds}$$

Table 3-1 shows a comparison between Autodesk Nastran and the theoretical result for the critical buckling load. The theoretical result is based on the Euler buckling formula for a pinned bar under axial compression. The formula is:

$$P_{cr} = \frac{\pi^2 EI}{\ell^2}$$

where,

E is Young's Modulus
I is the moment of inertia about the applicable plane
ℓ is the length of the beam

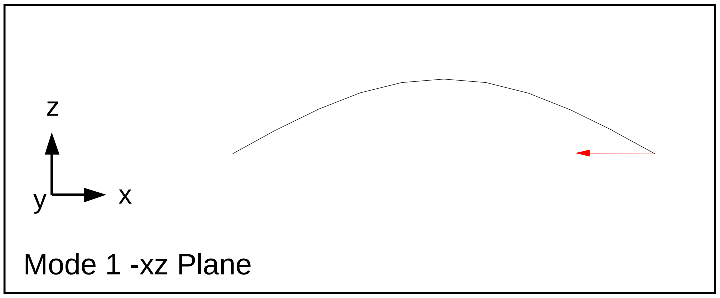
Table 3-1. Comparison of Theoretical Versus Predicted Critical Buckling Loads.

Mode Number	Theoretical (pounds)	Autodesk Nastran (pounds)	Difference (%)
1 (xz-plane)	658.0	658.0	0.0
5 (xy-plane)	16452.6	16452.9	0.0

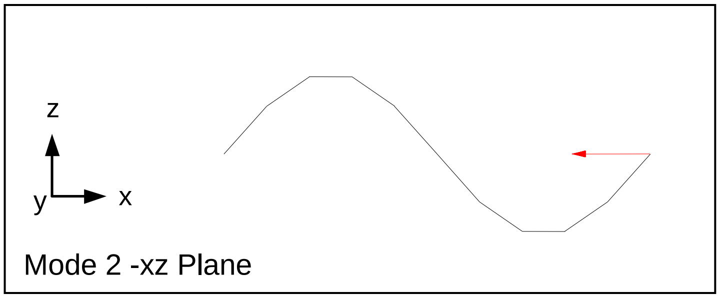
Listing 3-3. Extracted Eigenvectors for an Euler Beam.

```
EIGENVALUE = 0.65802E+00  CYCLES = 0.12910E+00                        SUBCASE 2

                                R E A L   E I G E N V E C T O R   N U M B E R   1

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID           ID
       1            0      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00 -0.314159E+00  0.000000E+00
       2            0      0.000000E+00  0.000000E+00  0.309017E+00  0.000000E+00 -0.298783E+00  0.000000E+00
       3            0      0.000000E+00  0.000000E+00  0.587785E+00  0.000000E+00 -0.254160E+00  0.000000E+00
       4            0      0.000000E+00  0.000000E+00  0.809017E+00  0.000000E+00 -0.184658E+00  0.000000E+00
       5            0      0.000000E+00  0.000000E+00  0.951057E+00  0.000000E+00 -0.970805E-01  0.000000E+00
       6            0      0.000000E+00  0.000000E+00  0.100000E+01  0.000000E+00 -0.220605E-14  0.000000E+00
       7            0      0.000000E+00  0.000000E+00  0.951057E+00  0.000000E+00  0.970805E-01  0.000000E+00
       8            0      0.000000E+00  0.000000E+00  0.809017E+00  0.000000E+00  0.184658E+00  0.000000E+00
       9            0      0.000000E+00  0.000000E+00  0.587785E+00  0.000000E+00  0.254160E+00  0.000000E+00
      10            0      0.000000E+00  0.000000E+00  0.309017E+00  0.000000E+00  0.298783E+00  0.000000E+00
      11            0      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.314159E+00  0.000000E+00

EIGENVALUE = 0.26326E+01  CYCLES = 0.25823E+00                        SUBCASE 2

                                R E A L   E I G E N V E C T O R   N U M B E R   2

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID           ID
       1            0      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00 -0.660649E+00  0.000000E+00
       2            0      0.000000E+00  0.000000E+00  0.618034E+00  0.000000E+00 -0.534476E+00  0.000000E+00
       3            0      0.000000E+00  0.000000E+00  0.100000E+01  0.000000E+00 -0.204152E+00  0.000000E+00
       4            0      0.000000E+00  0.000000E+00  0.100000E+01  0.000000E+00  0.204152E+00  0.000000E+00
       5            0      0.000000E+00  0.000000E+00  0.618034E+00  0.000000E+00  0.534476E+00  0.000000E+00
       6            0      0.000000E+00  0.000000E+00  0.273656E-12  0.000000E+00  0.660649E+00  0.000000E+00
       7            0      0.000000E+00  0.000000E+00 -0.618034E+00  0.000000E+00  0.534476E+00  0.000000E+00
       8            0      0.000000E+00  0.000000E+00 -0.100000E+01  0.000000E+00  0.204152E+00  0.000000E+00
       9            0      0.000000E+00  0.000000E+00 -0.100000E+01  0.000000E+00 -0.204152E+00  0.000000E+00
      10            0      0.000000E+00  0.000000E+00 -0.618034E+00  0.000000E+00 -0.534476E+00  0.000000E+00
      11            0      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00 -0.660649E+00  0.000000E+00

EIGENVALUE = 0.59283E+01  CYCLES = 0.38751E+00                        SUBCASE 2

                                R E A L   E I G E N V E C T O R   N U M B E R   3

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID           ID
       1            0      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.942404E+00  0.000000E+00
       2            0      0.000000E+00  0.000000E+00 -0.809017E+00  0.000000E+00  0.553931E+00  0.000000E+00
       3            0      0.000000E+00  0.000000E+00 -0.951057E+00  0.000000E+00 -0.291219E+00  0.000000E+00
       4            0      0.000000E+00  0.000000E+00 -0.309017E+00  0.000000E+00 -0.896279E+00  0.000000E+00
       5            0      0.000000E+00  0.000000E+00  0.587785E+00  0.000000E+00 -0.762421E+00  0.000000E+00
       6            0      0.000000E+00  0.000000E+00  0.100000E+01  0.000000E+00 -0.761357E-14  0.000000E+00
       7            0      0.000000E+00  0.000000E+00  0.587785E+00  0.000000E+00  0.762421E+00  0.000000E+00
       8            0      0.000000E+00  0.000000E+00 -0.309017E+00  0.000000E+00  0.896279E+00  0.000000E+00
       9            0      0.000000E+00  0.000000E+00 -0.951057E+00  0.000000E+00  0.291219E+00  0.000000E+00
      10            0      0.000000E+00  0.000000E+00 -0.809017E+00  0.000000E+00 -0.553931E+00  0.000000E+00
      11            0      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00 -0.942404E+00  0.000000E+00

EIGENVALUE = 0.10562E+02  CYCLES = 0.51724E+00                        SUBCASE 2

                                R E A L   E I G E N V E C T O R   N U M B E R   4

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID           ID
       1            0      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.100000E+01  0.000000E+00
       2            0      0.000000E+00  0.000000E+00 -0.757180E+00  0.000000E+00  0.309017E+00  0.000000E+00
       3            0      0.000000E+00  0.000000E+00 -0.467963E+00  0.000000E+00 -0.809017E+00  0.000000E+00
       4            0      0.000000E+00  0.000000E+00  0.467963E+00  0.000000E+00 -0.809017E+00  0.000000E+00
       5            0      0.000000E+00  0.000000E+00  0.757180E+00  0.000000E+00  0.309017E+00  0.000000E+00
       6            0      0.000000E+00  0.000000E+00 -0.702493E-08  0.000000E+00  0.100000E+01  0.000000E+00
       7            0      0.000000E+00  0.000000E+00 -0.757180E+00  0.000000E+00  0.309017E+00  0.000000E+00
       8            0      0.000000E+00  0.000000E+00 -0.467963E+00  0.000000E+00 -0.809017E+00  0.000000E+00
       9            0      0.000000E+00  0.000000E+00  0.467963E+00  0.000000E+00 -0.809017E+00  0.000000E+00
      10            0      0.000000E+00  0.000000E+00  0.757180E+00  0.000000E+00  0.309017E+00  0.000000E+00
      11            0      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.100000E+01  0.000000E+00

EIGENVALUE = 0.16453E+02  CYCLES = 0.64557E+00                        SUBCASE 2

                                R E A L   E I G E N V E C T O R   N U M B E R   5

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID           ID
       1            0      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.779608E-08  0.314160E+00
       2            0      0.000000E+00  0.309018E+00 -0.264147E-08  0.000000E+00  0.766531E-08  0.298784E+00
       3            0      0.000000E+00  0.587786E+00 -0.502559E-08  0.000000E+00  0.728555E-08  0.254160E+00
       4            0      0.000000E+00  0.809018E+00 -0.691976E-08  0.000000E+00  0.669333E-08  0.184658E+00
       5            0      0.000000E+00  0.951057E+00 -0.813855E-08  0.000000E+00  0.594580E-08  0.970802E-01
       6            0      0.000000E+00  0.100000E+01 -0.856191E-08  0.000000E+00  0.511549E-08  0.382391E-14
       7            0      0.000000E+00  0.951057E+00 -0.814717E-08  0.000000E+00  0.428341E-08 -0.970802E-01
       8            0      0.000000E+00  0.809018E+00 -0.693370E-08  0.000000E+00  0.353128E-08 -0.184658E+00
       9            0      0.000000E+00  0.587786E+00 -0.503954E-08  0.000000E+00  0.293336E-08 -0.254160E+00
      10            0      0.000000E+00  0.309018E+00 -0.265008E-08  0.000000E+00  0.254900E-08 -0.298784E+00
      11            0      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.241647E-08 -0.314160E+00
```

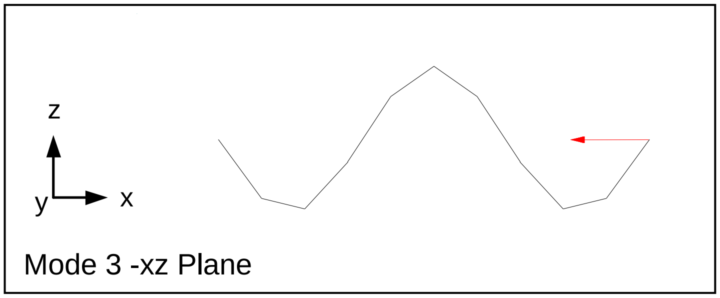
Figure 3-2. Buckled Mode Shapes of an Euler Beam.



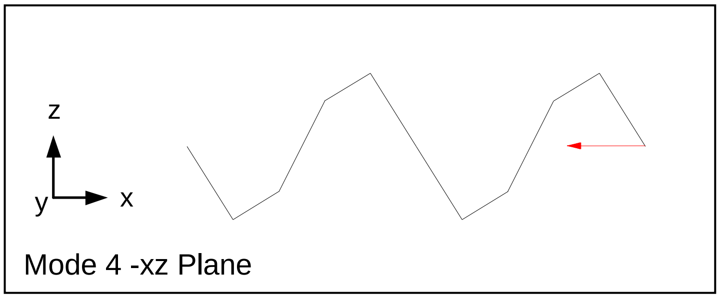
Mode 1 -xz Plane



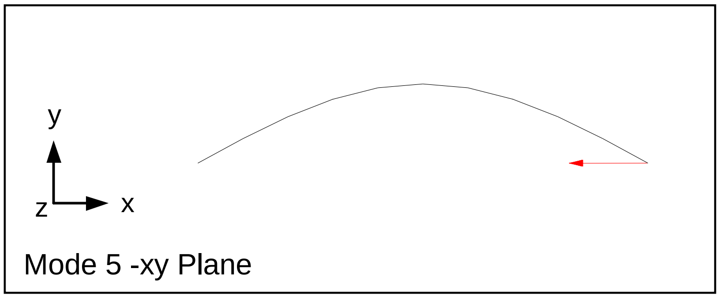
Mode 2 -xz Plane



Mode 3 -xz Plane



Mode 4 -xz Plane



Mode 5 -xy Plane

In most applications only one static and one buckling analysis is performed per run. If we wanted, however, to analyze the two other boundary conditions shown in Figures 3-3a and 3-3b, the Model Input File would look as shown in Listing 3-4. Listing 3-5 shows the extracted eigenvalues from the Model Results Output File.

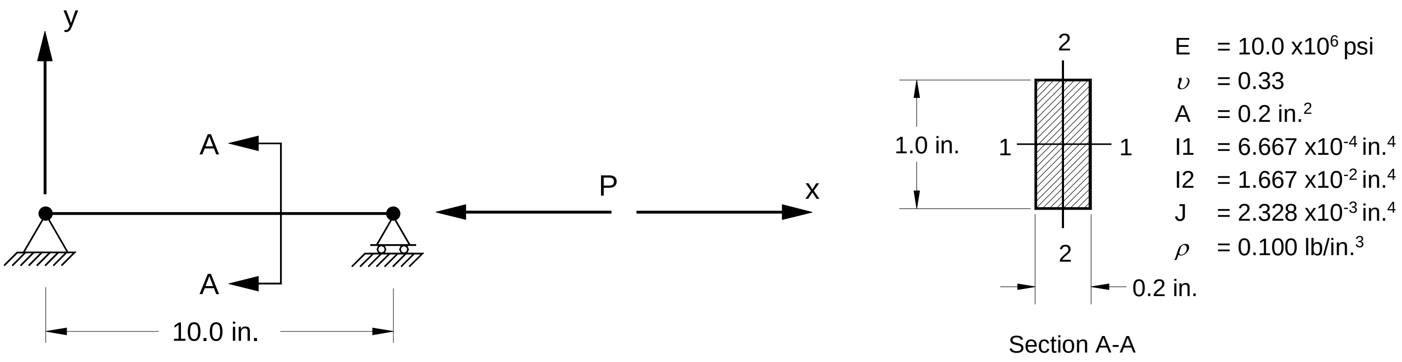


Figure 3-3a. Euler Beam Example Problem -Fixed at One End, Pinned at Other End.

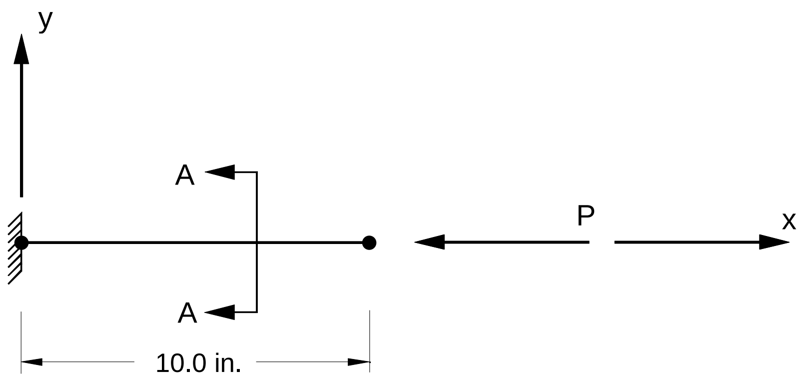


Figure 3-3b. Euler Beam Example Problem -Fixed at One End, Free at Other End.

Listing 3-4. Model Input File for an Euler Beam with Multiple Boundary Conditions.

```
$
$ BUCKLING SOLUTION.
$
SOL BUCKLING
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = EULER BEAM BUCKLING WITH MULTIPLE BOUNDARY CONDITIONS
$
DISPLACEMENT = ALL
$
SUBCASE 1
 LABEL = STATIC, COMPRESSIVE LOAD IN X-DIR
 LOAD = 1
 STRESS = ALL
 SPC = 2
SUBCASE 2
 LABEL = BUCKLING, COMPRESSIVE LOAD IN X-DIR, FIXED-PINNED
 STRESS = NONE
 METHOD = 1
 SPC = 2
SUBCASE 3
 LABEL = BUCKLING, COMPRESSIVE LOAD IN X-DIR, FIXED-FREE
 STRESS = NONE
 METHOD = 1
 SPC = 3
$
BEGIN BULK
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  5
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0” X 0.2” CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3,
,  -0.1,  0.5,  0.1,  0.5,  -0.1,  -0.5,  0.1,  -0.5
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
```

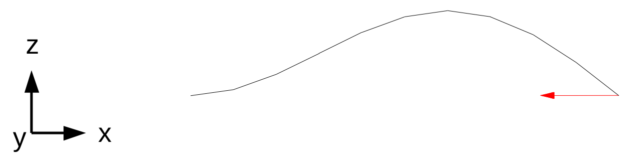
Listing 3-4. Model Input File for an Euler Beam with Multiple Boundary Conditions. (Continued)

```
$
$ FIXED AT ONE END, PINNED AT OTHER -ONE END FREE TO TRANSLATE IN X-DIR.
$
SPC1,  2,  123456,   1
SPC1,  2,      23,  11
SPC1,  2,       4,   1,  THRU,  11
$
$ FIXED AT ONE END, FREE AT OTHER.
$
SPC1,  3,  123456,   1
SPC1,  3,       4,   1,  THRU,  11
$
$ COMPRESSIVE LOAD (X-DIRECTION).
$
FORCE,  1,  11,  0,  1000.,  -1.,  0.,  0.
ENDDATA
```

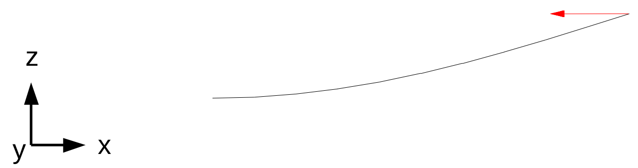
Listing 3-5. Extracted Eigenvectors for an Euler Beam with Multiple Boundary Conditions.

```
BUCKLING, COMPRESSIVE LOAD IN X-DIR, FIXED-PINNED                   SUBCASE 3

                                                  R E A L   E I G E N V A L U E S

     MODE         EIGENVALUE       RADIANS          CYCLES        GENERALIZED      GENERALIZED      ORTHOGONALITY       ERROR
    NUMBER                                                           MASS           STIFFNESS           LOSS           MEASURE
       1        1.346191E+00    1.160255E+00    1.846603E-01    5.162777E+02    6.950085E+02    0.000000E+00    1.114939E-14
       2        3.980739E+00    1.995179E+00    3.175426E-01    1.156760E+03    4.604759E+03    9.797718E-15    6.452137E-12
       3        7.941700E+00    2.818102E+00    4.485149E-01    2.099060E+03    1.667011E+04    7.355228E-16    6.042417E-10
       4        1.325625E+01    3.640914E+00    5.794694E-01    3.127143E+03    4.145419E+04    3.018422E-16    6.956877E-08
       5        1.998106E+01    4.470018E+00    7.114255E-01    4.380083E+03    8.751872E+04    6.455984E-16    2.385576E-06


BUCKLING, COMPRESSIVE LOAD IN X-DIR, FIXED-FREE                     SUBCASE 4

                                                  R E A L   E I G E N V A L U E S

     MODE         EIGENVALUE       RADIANS          CYCLES        GENERALIZED      GENERALIZED      ORTHOGONALITY       ERROR
    NUMBER                                                           MASS           STIFFNESS           LOSS           MEASURE
       1        1.645018E-01    4.055882E-01    6.455136E-02    1.233698E+02    2.029456E+01    0.000000E+00    1.445645E-13
       2        1.480615E+00    1.216805E+00    1.936606E-01    2.809938E+02    4.160436E+02    1.742391E-13    1.796533E-08
       3        4.113161E+00    2.028093E+00    3.227810E-01    1.233698E+02    5.074401E+02    3.731604E-17    1.521355E-06
       4        4.114647E+00    2.028459E+00    3.228393E-01    7.702665E+02    3.169375E+03    9.055332E-16    2.029742E-06
       5        8.075991E+00    2.841829E+00    4.522911E-01    1.524099E+03    1.230861E+04    1.290036E-15    1.207591E-05
```

Figure 3-4. Buckled Mode Shapes of an Euler Beam with Multiple Boundary Conditions.



Subcase 3, Mode 1, Fixed-Pinned Boundary Condition



Subcase 4, Mode 1, Fixed-Free Boundary Condition

3.4 Assumptions and Limitations of Linear Buckling

The following assumptions and limitations apply to linear buckling analysis:

1. The deflections are small.
2. The element stresses are elastic.
3. A minimum of five grid points per half sine wave (buckled mode shape) is recommended.
4. The distribution of the internal element forces due to the applied loads remains constant.
5. The follower force effect is not included in the generation of differential stiffness (i.e., the directions and magnitudes of the applied forces are assumed to remain constant). Follower force effects can be included by using a `NONLINEAR STATIC` solution (see Section 9, *Nonlinear Static Analysis*).
6. The tangent stiffness term due to follower force effect is not included.
7. Offsets should not be used in bar, beam, and shell elements.
8. For curved shell structures modeled with shell elements it is recommended that you use `CQUADR` and `CTRIAR` elements. These elements include vertex rotation stiffness and give significantly better results.

4. NORMAL MODES ANALYSIS

4.1 Introduction

Problems in structural dynamics can be divided into two broad areas. In one, the objective is to determine natural frequencies of vibration and the corresponding mode shapes. In the other, the objective is to determine how the structure moves with time under an applied set of loads. In this section we examine the former, natural frequencies of vibration, with damping and applied loading both set to zero. Vibration of structures under initial stress is discussed in Section 11, *Linear Prestress Modal Analysis*.

Autodesk Nastran determines natural frequency by solving the eigenvalue problem:

$$\left| [K] - \lambda[M] \right| [\phi] = 0$$

$$\lambda_i = \omega_i^2$$

$$f_i = \frac{\omega_i}{2\pi}$$

where,

$[K]$	is the global linear stiffness matrix
$[M]$	is the global mass matrix
λ_i	is the eigenvalue for each mode that yields the natural frequency
ϕ_i	is the eigenvector for each mode that represents the natural mode shape
ω_i	is the circular frequency (radians per second)
f_i	is the cyclic frequency (hertz)

In solving the above eigenvalue problem there are as many eigenvalues and corresponding eigenvectors as there are unconstrained degrees of freedom. Often, however, only the lowest natural frequency is of practical interest. This frequency will *always* be the first mode extracted.

4.2 How to Setup a Model Input File for Normal Modes Analysis

In Autodesk Nastran you can perform normal modes analysis by setting `SOLUTION = MODAL` in the Model Initialization File or by specifying `SOL 103` or `SOL MODAL` above the Case Control Section in the Model Input File. Multiple subcases can be specified, each requesting a different constraint or output set. Each subcase must also reference an `EIGRL` Bulk Data entry via the `METHOD` Case Control command.

4.3 Interpreting Results

As an example we will use the cantilever beam shown in Figure 4-1. It is desired to find the lowest natural frequency and the corresponding mode shape. Listing 4-1 contains the Model Input File and Listings 4-2 and 4-3 show the extracted frequencies and eigenvectors from the Model Results Output File. The mode shapes are plotted in Figure 4-2.

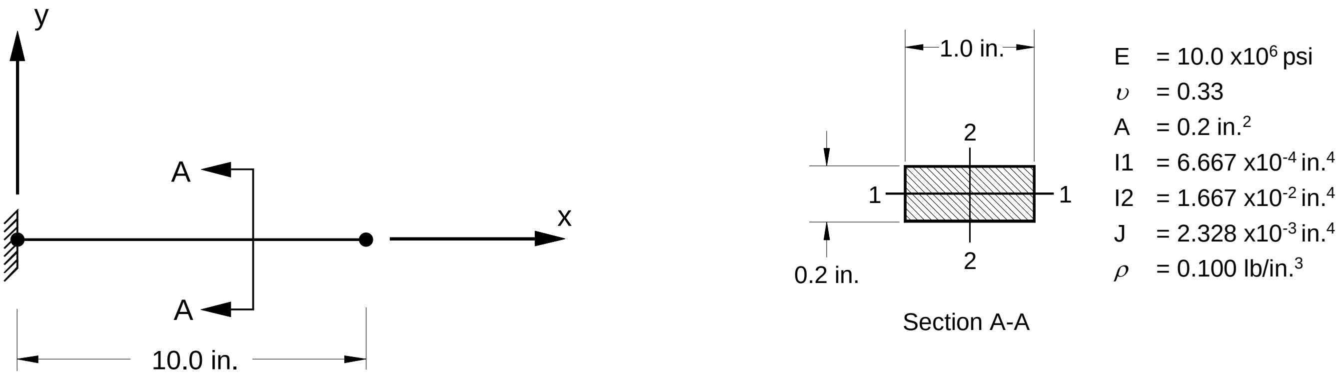


Figure 4-1. 2-D Cantilever Beam Example Problem.

Listing 4-1. Model Input File for the 2-D Cantilever Beam Problem.

```
$
$ MODAL SOLUTION.
$
SOL MODAL
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = VIBRATION OF A 2-D CANTILEVER BEAM
$
DISPLACEMENT = ALL
$
SUBCASE 1
 LABEL = NORMAL MODES
 SPC = 1
 METHOD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  5
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  1.,  0.
CBAR,  2,  10,   2,   3,  0.,  1.,  0.
CBAR,  3,  10,   3,   4,  0.,  1.,  0.
CBAR,  4,  10,   4,   5,  0.,  1.,  0.
CBAR,  5,  10,   5,   6,  0.,  1.,  0.
CBAR,  6,  10,   6,   7,  0.,  1.,  0.
CBAR,  7,  10,   7,   8,  0.,  1.,  0.
CBAR,  8,  10,   8,   9,  0.,  1.,  0.
CBAR,  9,  10,   9,  10,  0.,  1.,  0.
CBAR, 10,  10,  10,  11,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3,
,  -0.1,  0.5,  0.1,  0.5,  -0.1,  -0.5,  0.1,  -0.5
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ FIXED AT ONE END, MOVEMENT CONSTRAINED TO X-Y PLANE ONLY.
$
SPC1,  1,  123456,  1
SPC1,  1,     345,  1,  THRU,  11
ENDDATA
```

The `EIGRL` entry controls the range and number of modes extracted. Here, we have requested 5 modes as shown in Listings 4-2 and 4-3. The eigenvalues and frequencies are always sorted in increasing order. Thus, the first mode is always the lowest. The cyclic frequency for the first mode is equal to 63.50 Hz.

Listing 4-2. Extracted Eigenvalues for a 2-D Cantilever Beam.

```
MODAL ANALYSIS,  COUPLED MASS MATRIX FORMULATION                    SUBCASE 1

                                                  R E A L   E I G E N V A L U E S

     MODE          EIGENVALUE        RADIANS          CYCLES        GENERALIZED      GENERALIZED      ORTHOGONALITY       ERROR
    NUMBER                                                              MASS          STIFFNESS           LOSS           MEASURE
       1         1.592103E+05     3.990117E+02     6.350468E+01     1.000000E+00     1.592103E+05     0.000000E+00     1.815924E-13
       2         6.247437E+06     2.499487E+03     3.978058E+02     1.000000E+00     6.247437E+06     4.182852E-15     1.957341E-12
       3         4.892945E+07     6.994959E+03     1.113282E+03     1.000000E+00     4.892945E+07     1.099815E-15     1.281154E-10
       4         1.877429E+08     1.370193E+04     2.180730E+03     1.000000E+00     1.877429E+08     1.200429E-15     1.115000E-08
       5         5.131852E+08     2.265359E+04     3.605431E+03     1.000000E+00     5.131852E+08     7.494005E-16     1.739917E-06
```

The theoretical result is based on the following formula from Reference 11 for the natural frequency of a uniform cantilever beam:

$$f_i = \frac{K_i}{2\pi}\sqrt{\frac{EIg}{\rho A \ell^4}}$$

Mode	K_i
1	3.52
2	22.0
3	61.7
4	121
5	200

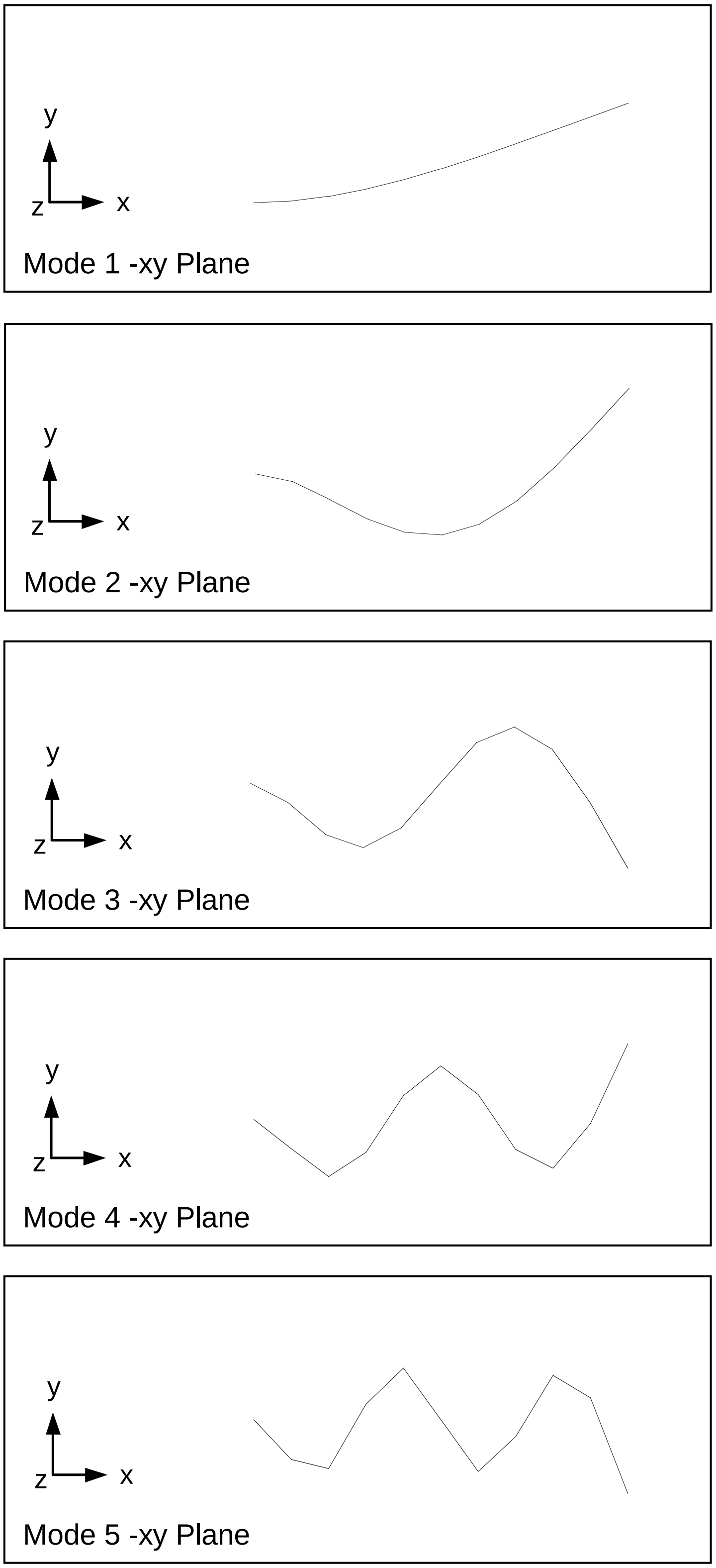
where,

f_i	are the natural frequencies (hertz) corresponding to the i-th mode shape
K_i	are constants corresponding to the i-th mode shape
E	is Young's Modulus
I	is the moment of inertia about the applicable plane
A	is the cross-sectional area
ρ	is the material density
g	is the gravitational acceleration (units consistent with length dimensions)
ℓ	is the length of the beam

Listing 4-3. Extracted Eigenvectors for a 2-D Cantilever Beam.

```
MODE = 1  EIGENVALUE = 1.592103E+05  CYCLES = 6.350468E+01          SUBCASE 1

                               R E A L   E I G E N V E C T O R   N U M B E R   1

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID          ID
        2           0     -1.224745E-13  1.474379E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.877884E+00
        3           0     -2.182180E-13  5.614241E+00  0.000000E+00  0.000000E+00  0.000000E+00  5.331303E+00
        4           0     -2.663481E-13  1.199688E+01  0.000000E+00  0.000000E+00  0.000000E+00  7.364628E+00
        5           0     -2.563841E-13  2.020699E+01  0.000000E+00  0.000000E+00  0.000000E+00  8.988734E+00
        6           0     -1.905220E-13  2.984445E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.022352E+01
        7           0     -8.314485E-14  4.053442E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.109991E+01
        8           0      4.232528E-14  5.193915E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.166139E+01
        9           0      1.585461E-13  6.377113E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.196513E+01
       10           0      2.402153E-13  7.580715E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.208262E+01
       11           0      2.695660E-13  8.790287E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.210013E+01

MODE = 2  EIGENVALUE = 6.247437E+06  CYCLES = 3.978058E+02          SUBCASE 1

                               R E A L   E I G E N V E C T O R   N U M B E R   2

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID          ID
        2           0      1.521171E-12 -8.137403E+00  0.000000E+00  0.000000E+00  0.000000E+00 -1.473772E+01
        3           0      2.710335E-12 -2.644908E+01  0.000000E+00  0.000000E+00  0.000000E+00 -2.041953E+01
        4           0      3.308125E-12 -4.622633E+01  0.000000E+00  0.000000E+00  0.000000E+00 -1.788336E+01
        5           0      3.184369E-12 -6.005469E+01  0.000000E+00  0.000000E+00  0.000000E+00 -8.892313E+00
        6           0      2.366339E-12 -6.271438E+01  0.000000E+00  0.000000E+00  0.000000E+00  3.974372E+00
        7           0      1.032680E-12 -5.180948E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.773693E+01
        8           0     -5.256994E-13 -2.787804E+01  0.000000E+00  0.000000E+00  0.000000E+00  2.961544E+01
        9           0     -1.969199E-12  6.132611E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.767520E+01
       10           0     -2.983557E-12  4.600300E+01  0.000000E+00  0.000000E+00  0.000000E+00  4.138747E+01
       11           0     -3.348103E-12  8.785747E+01  0.000000E+00  0.000000E+00  0.000000E+00  4.201618E+01

MODE = 3  EIGENVALUE = 4.892945E+07  CYCLES = 1.113282E+03          SUBCASE 1

                               R E A L   E I G E N V E C T O R   N U M B E R   3

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID          ID
        2           0      1.642381E-10 -2.002652E+01  0.000000E+00  0.000000E+00  0.000000E+00 -3.307044E+01
        3           0      2.926299E-10 -5.309707E+01  0.000000E+00  0.000000E+00  0.000000E+00 -2.740570E+01
        4           0      3.571719E-10 -6.645183E+01  0.000000E+00  0.000000E+00  0.000000E+00  3.084871E+00
        5           0      3.438096E-10 -4.625482E+01  0.000000E+00  0.000000E+00  0.000000E+00  3.563308E+01
        6           0      2.554876E-10 -1.802980E+00  0.000000E+00  0.000000E+00  0.000000E+00  4.876882E+01
        7           0      1.114939E-10  4.156061E+01  0.000000E+00  0.000000E+00  0.000000E+00  3.333828E+01
        8           0     -5.676249E-11  5.774125E+01  0.000000E+00  0.000000E+00  0.000000E+00 -3.079682E+00
        9           0     -2.126154E-10  3.472771E+01  0.000000E+00  0.000000E+00  0.000000E+00 -4.155832E+01
       10           0     -3.221344E-10 -2.001330E+01  0.000000E+00  0.000000E+00  0.000000E+00 -6.446719E+01
       11           0     -3.614939E-10 -8.780607E+01  0.000000E+00  0.000000E+00  0.000000E+00 -6.897868E+01

MODE = 4  EIGENVALUE = 1.877429E+08  CYCLES = 2.180730E+03          SUBCASE 1

                               R E A L   E I G E N V E C T O R   N U M B E R   4

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID          ID
        2           0     -2.533925E-08 -3.381413E+01  0.000000E+00  0.000000E+00  0.000000E+00 -4.888631E+01
        3           0     -4.514793E-08 -6.626123E+01  0.000000E+00  0.000000E+00  0.000000E+00 -5.398795E+00
        4           0     -5.510550E-08 -3.822775E+01  0.000000E+00  0.000000E+00  0.000000E+00  5.702071E+01
        5           0     -5.304356E-08  2.762323E+01  0.000000E+00  0.000000E+00  0.000000E+00  6.159572E+01
        6           0     -3.941644E-08  6.214470E+01  0.000000E+00  0.000000E+00  0.000000E+00  5.441405E-01
        7           0     -1.719998E-08  2.882745E+01  0.000000E+00  0.000000E+00  0.000000E+00 -6.013256E+01
        8           0      8.759806E-09 -3.478329E+01  0.000000E+00  0.000000E+00  0.000000E+00 -5.355421E+01
        9           0      3.280586E-08 -5.651087E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.588596E+01
       10           0      4.970318E-08 -4.698040E+00  0.000000E+00  0.000000E+00  0.000000E+00  8.102303E+01
       11           0      5.577583E-08  8.778828E+01  0.000000E+00  0.000000E+00  0.000000E+00  9.672126E+01

MODE = 5  EIGENVALUE = 5.131852E+08  CYCLES = 3.605431E+03          SUBCASE 1

                               R E A L   E I G E N V E C T O R   N U M B E R   5

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID          ID
        2           0      6.609671E-06  4.726964E+01  0.000000E+00  0.000000E+00  0.000000E+00  5.609961E+01
        3           0      1.177662E-05  5.817921E+01  0.000000E+00  0.000000E+00  0.000000E+00 -4.345606E+01
        4           0      1.437377E-05 -1.841751E+01  0.000000E+00  0.000000E+00  0.000000E+00 -8.465672E+01
        5           0      1.383547E-05 -6.135489E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.326372E+01
        6           0      1.028026E-05 -1.563481E-01  0.000000E+00  0.000000E+00  0.000000E+00  8.797794E+01
        7           0      4.484434E-06  6.161589E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.433282E+01
        8           0     -2.287838E-06  2.015540E+01  0.000000E+00  0.000000E+00  0.000000E+00 -8.264287E+01
        9           0     -8.560816E-06 -5.269228E+01  0.000000E+00  0.000000E+00  0.000000E+00 -3.661835E+01
       10           0     -1.296886E-05 -2.608125E+01  0.000000E+00  0.000000E+00  0.000000E+00  8.608213E+01
       11           0     -1.455305E-05  8.786180E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.246716E+02
```

Figure 4-2. Mode Shapes of a 2-D Cantilever Beam.



Note that Listings 4-1 and 4-2 are for a model run with `PARAM,COUPMASS,ON`, which requests coupled mass matrix formulation. Listing 4-4 shows the output for the same model in Listing 4-1, but with `PARAM,COUPMASS,OFF`, which requests diagonal mass matrix formulation. While the diagonal mass formulation is slightly faster, the coupled mass formulation is usually more accurate. Table 4-1 shows a comparison between Autodesk Nastran and the theoretical natural frequency.

Listing 4-4. Extracted Eigenvalues for a Cantilever Beam Using the Diagonal Mass Formulation.

```
MODAL ANALYSIS,  DIAGONAL MASS MATRIX FORMULTAION                 SUBCASE 1

                                          R E A L   E I G E N V A L U E S

     MODE        EIGENVALUE       RADIANS          CYCLES       GENERALIZED     GENERALIZED     ORTHOGONALITY      ERROR
    NUMBER                                                          MASS         STIFFNESS          LOSS          MEASURE
       1        1.576987E+05    3.971130E+02    6.320250E+01    1.000000E+00    1.576987E+05    0.000000E+00    6.465823E-14
       2        6.037523E+06    2.457137E+03    3.910655E+02    1.000000E+00    6.037523E+06    6.585986E-16    9.583959E-13
       3        4.616942E+07    6.794808E+03    1.081427E+03    1.000000E+00    4.616942E+07    1.477117E-15    1.168944E-10
       4        1.724843E+08    1.313333E+04    2.090234E+03    1.000000E+00    1.724843E+08    4.310788E-16    9.207484E-09
       5        4.574813E+08    2.138881E+04    3.404135E+03    1.000000E+00    4.574813E+08    7.797582E-16    1.559786E-06
```

Listings 4-2, 4-4 and Table 4-1 also show that as the mode number increases the accuracy decreases. The accuracy achieved is determined by a number of factors and can be controlled using settings on the `EIRGL` entry.

The accuracy of the eigensolution is measured using both orthogonality loss and error measure. The stiffness orthogonality loss for each mode is defined using:

$$\delta_i = \{\phi\}_{i-1}^T [K] \{\phi\}_i$$

And mass orthogonality loss is defined using:

$$\delta_i = \{\phi\}_{i-1}^T [M] \{\phi\}_i$$

The value shown in Listing 4-4 is the maximum of the stiffness and mass orthogonality loss. Error measure in Listing 4-4 is determined using:

$$\varepsilon_i = \frac{\left| [K]\{\phi\}_i - \lambda_i [M]\{\phi\}_i \right|}{\left| [K]\{\phi\}_i \right|}$$

Table 4-1. Comparison of Theoretical Versus Predicted Natural Frequency for a Cantilever Beam.

	Theoretical	Autodesk Nastran Diagonal Mass Formulation		Autodesk Nastran Coupled Mass Formulation	
Mode Number	Natural Frequency (Hz)	Natural Frequency (Hz)	Difference (%)	Natural Frequency (Hz)	Difference (%)
1	63.6	63.2	0.6	63.5	0.1
2	397.4	391.8	1.4	397.8	0.1
3	1114.5	1086.0	2.6	1113.3	0.1
4	2185.6	2106.0	3.6	2180.7	0.2
5	3612.6	3442.6	4.7	3605.4	0.2

4.4 Rigid-Body Modes

In the example problem of Figure 4.1 the beam is completely constrained at one end. If the beam was unconstrained it would displace without developing any internal loads or stresses. These stress-free displacements are referred to as rigid-body modes or mechanism modes.

Rigid-body modes occur in a completely unconstrained structure such as a rocket or aircraft in flight. For 3-dimensional problems that are completely unconstrained, there are six rigid body modes. These modes are referred to as T1, T2, T3, R1, R2, and R3 and will be extracted as modes one through six. Mode seven is then referred to as the first flexible mode and will not be a zero energy mode. For 2-dimensional problems there will be three rigid body modes T1, T2, and R3 and the first flexible mode will be mode four.

Mechanism modes occur in an insufficiently constrained structure where a portion of the structure displaces as a rigid body. An example would be a flat plate on a hinge or a ball joint. A mechanism mode can also occur when two parts of a structure are not connected properly. A common example of this is a bar connected to a solid element.

Rigid-body and mechanism modes are indicated by zero or near zero frequency eigenvalues. For most structures near zero should be on the order of 1.0E-3 Hz or less and may be negative.

As an example we will use the beam shown in Figure 4-1 with only 2-dimensional constraints specified (end constraint removed). Listing 4-5 contains the Model Input File and Listing 4-6 shows the extracted frequencies and eigenvectors from the Model Results Output File. The mode shapes are plotted in Figure 4-3.

Listing 4-5. Model Input File for the 2-D Unconstrained Beam Problem.

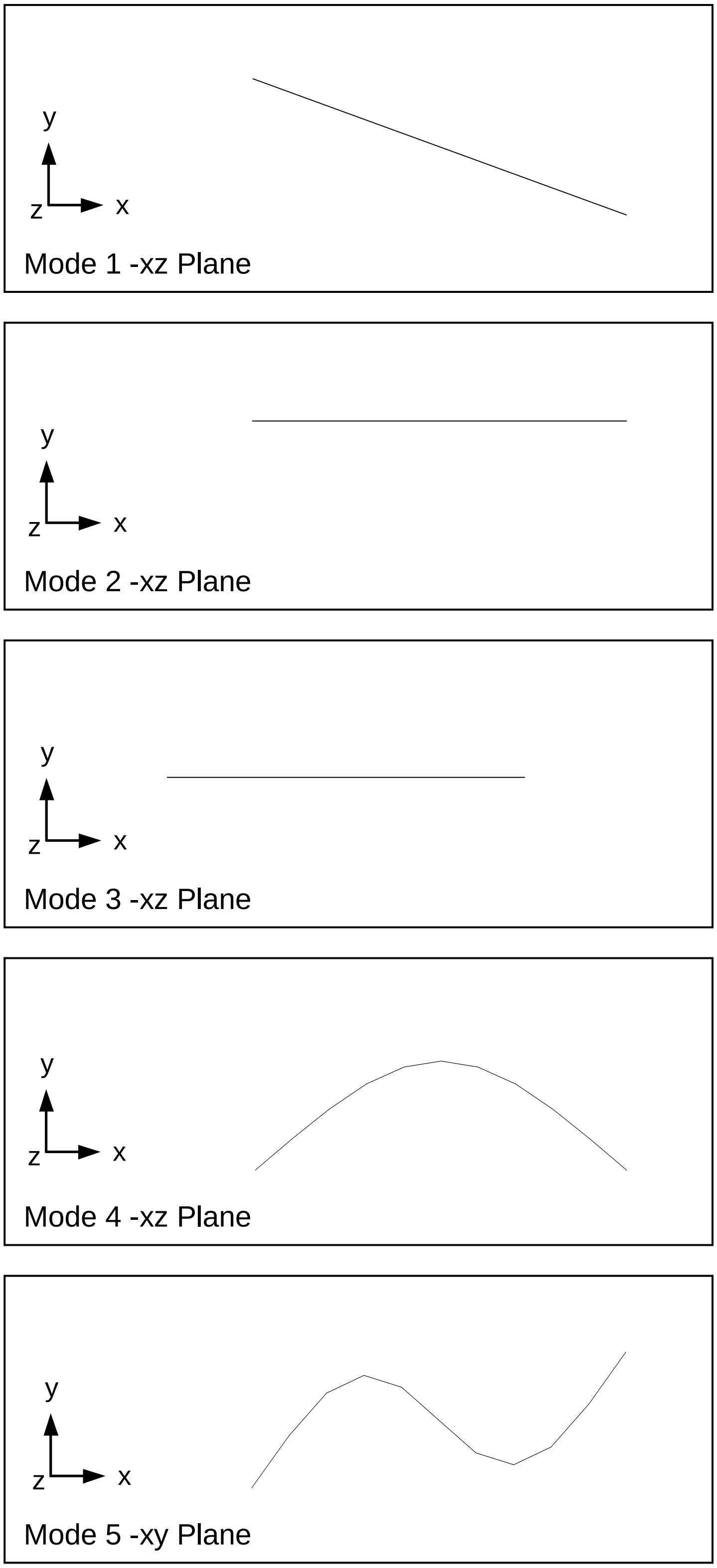
```
$
$ MODAL SOLUTION.
$
SOL MODAL
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = RIGID BODY MODES OF A 2-D UNCONSTRAINED BEAM
$
DISPLACEMENT = ALL
$
SUBCASE 1
 LABEL = NORMAL MODES
 SPC = 1
 METHOD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  5
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  1.,  0.
CBAR,  2,  10,   2,   3,  0.,  1.,  0.
CBAR,  3,  10,   3,   4,  0.,  1.,  0.
CBAR,  4,  10,   4,   5,  0.,  1.,  0.
CBAR,  5,  10,   5,   6,  0.,  1.,  0.
CBAR,  6,  10,   6,   7,  0.,  1.,  0.
CBAR,  7,  10,   7,   8,  0.,  1.,  0.
CBAR,  8,  10,   8,   9,  0.,  1.,  0.
CBAR,  9,  10,   9,  10,  0.,  1.,  0.
CBAR, 10,  10,  10,  11,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3,
,  -0.1,  0.5,  0.1,  0.5,  -0.1,  -0.5,  0.1,  -0.5
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ MOVEMENT CONSTRAINED TO X-Y PLANE ONLY.
$
SPC1,  1,  345,  1,  THRU,  11
ENDDATA
```

As expected, the first 3 frequencies are near zero and the fourth non-zero. Figure 4-3 depicts classical rigid body modes whereby each mode contains motion in only one degree of freedom. It is important to note however that any linear combination of these displacement shapes also comprises a valid set of rigid body modes.

Listing 4-6. Extracted Eigenvalues for a 2-D Unconstrained Beam.

```
MODAL ANALYSIS,  COUPLED MASS MATRIX FORMULATION                    SUBCASE 1

                                                  R E A L   E I G E N V A L U E S

     MODE         EIGENVALUE       RADIANS          CYCLES        GENERALIZED     GENERALIZED     ORTHOGONALITY       ERROR
    NUMBER                                                           MASS          STIFFNESS          LOSS           MEASURE
        1        5.587935E-08    2.363881E-04    3.762234E-05    1.000000E+00    5.587935E-08    0.000000E+00    9.465381E-14
        2        1.229346E-07    3.506203E-04    5.580295E-05    1.000000E+00    1.229346E-07    2.649432E-15    1.976175E-13
        3       -4.060566E-07    6.372257E-04    1.014176E-04    1.000000E+00   -4.060566E-07    4.128210E-18    6.873596E-13
        4        6.437379E+06    2.537199E+03    4.038078E+02    1.000000E+00    6.437379E+06    7.482689E-18    4.289748E-11
        5        4.883871E+07    6.988470E+03    1.112250E+03    1.000000E+00    4.883871E+07    1.268083E-15    3.360955E-09
```

Figure 4-3. Mode Shapes of a 2-D Unconstrained Beam.



4.5 Direct Matrix Export and Import

The direct matrix support in Autodesk Nastran provides a common interface for importing and exporting global stiffness, mass, and damping matrixes using the `DMIG` Bulk Data entry. Autodesk Nastran provides program control directives for exporting global matrixes at various stages of execution. Case Control commands and Bulk Data entries are provided for importing matrixes for use in all available solutions.

The following examples demonstrate how to setup a model for export to `DMIG` and how to import `DMIG` into a separate model. This interface is particularly useful for sharing confidential and sensitive information where the details of a design are not to be disclosed. For example, an engine contractor may need stiffness and mass data of an aircraft to perform an overall dynamic analysis. The aircraft contractor may not want the details of their design disclosed so a stiffness and mass matrix are provided which yield equivalent results.

As an example of `DMIG` we will use the beam shown in Figure 4-1. First, we will treat the last two elements (9, 10) as the portion of the model that is desired to be exported. Listing 4-7 contains the Model Input File. Note that a `MODAL` solution is selected to enable the generation of a full mass matrix.

Listing 4-7. Model Input File for the Direct Matrix Export of a 2-D Cantilever Beam Segment.

```
$
$ MODAL SOLUTION.
$
SOL MODAL
CEND
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = EXPORT STIFFNESS AND MASS MATRIX OF A CANTILEVER BEAM
$
BEGIN BULK
$
$ REQUEST A CHECKOUT RUN WHICH TERMINATES AFTER STIFFNESS AND MASS MATRIX ASSEMBLY.
$
PARAM, CHECKOUT, ON
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ GEOMETRY DEFINITION (2" BEAM SEGMENT DIVIDED INTO 2 ELEMENTS).
$
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  9,  10,   9,  10,  0.,  1.,  0.
CBAR, 10,  10,  10,  11,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
ENDDATA
```

Before the model is run, the `TRSLDMIDATA` Model Initialization directive must be set to `ON` (defaulted to `OFF`). Note that `PARAM, CHECKOUT, ON` has been added to terminate execution after the stiffness and mass matrixes are generated. Listing 4-8 shows the generated `DMIG` Bulk Data entries that are written to the Bulk Data Output File. The `DMIG` name format is

AXXXXXX

where,

A is the matrix type using one of the following symbols:
K stiffness or conductivity matrix output
M mass or capacitance matrix output
B damping matrix output
R follows the matrix type and indicates the matrix is reduced

and,

XXXXXXX is the subcase number.

Listing 4-8. Model Input File for the Direct Matrix Export of a 2-D Cantilever Beam Segment.

```
$
$ OUTPUT PRODUCED BY ADS NASTRAN VERSION 10.3.0.716   12:28    01/16/15
$
DMIG    K1             0       6       2
DMIG    K1             9       1               9       1 2.00000000D+006+C    1A
+C    1A      10       1-2.00000000D+006
DMIG    K1             9       2               9       2 8.00040000D+004+C    2A
+C    2A       9       6 4.00020000D+004      10       2-8.00040000D+004+C    3A
+C    3A      10       6 4.00020000D+004
DMIG    K1             9       3               9       3 2.00040000D+006+C    4A
+C    4A       9       5-1.00020000D+006      10       3-2.00040000D+006+C    5A
+C    5A      10       5-1.00020000D+006
DMIG    K1             9       4               9       4 8.75187970D+003+C    6A
+C    6A      10       4-8.75187970D+003
DMIG    K1             9       5               9       5 6.66800000D+005+C    7A
+C    7A      10       3 1.00020000D+006      10       5 3.33400000D+005
DMIG    K1             9       6               9       6 2.66680000D+004+C    8A
+C    8A      10       2-4.00020000D+004      10       6 1.33340000D+004
DMIG    K1            10       1              10       1 4.00000000D+006+C    9A
+C    9A      11       1-2.00000000D+006
DMIG    K1            10       2              10       2 1.60008000D+005+C   10A
+C   10A      11       2-8.00040000D+004      11       6 4.00020000D+004
DMIG    K1            10       3              10       3 4.00080000D+006+C   11A
+C   11A      11       3-2.00040000D+006      11       5-1.00020000D+006
DMIG    K1            10       4              10       4 1.75037594D+004+C   12A
+C   12A      11       4-8.75187970D+003
DMIG    K1            10       5              10       5 1.33360000D+006+C   13A
+C   13A      11       3 1.00020000D+006      11       5 3.33400000D+005
DMIG    K1            10       6              10       6 5.33360000D+004+C   14A
+C   14A      11       2-4.00020000D+004      11       6 1.33340000D+004
DMIG    K1            11       1              11       1 2.00000000D+006
DMIG    K1            11       2              11       2 8.00040000D+004+C   15A
+C   15A      11       6-4.00020000D+004
DMIG    K1            11       3              11       3 2.00040000D+006+C   16A
+C   16A      11       5 1.00020000D+006
DMIG    K1            11       4              11       4 8.75187970D+003
DMIG    K1            11       5              11       5 6.66800000D+005
DMIG    K1            11       6              11       6 2.66680000D+004
```

Listing 4-8. Model Input File for the Direct Matrix Export of a 2-D Cantilever Beam Segment. (Continued)

```
DMIG    M1             0       6       2
DMIG    M1             9       1               9       1 1.72533333D-005+C   17A
+C   17A      10       1 8.62666667D-006
DMIG    M1             9       2               9       2 1.72533333D-005+C   18A
+C   18A       9       6 2.72849229D-006      10       2 8.62666667D-006+C   19A
+C   19A      10       6-1.58484104D-006
DMIG    M1             9       3               9       3 1.72533333D-005+C   20A
+C   20A       9       5-3.14265770D-006      10       3 8.62666667D-006+C   21A
+C   21A      10       5 1.17067564D-006
DMIG    M1             9       4               9       4 2.00828800D-007+C   22A
+C   22A      10       4 1.00414400D-007
DMIG    M1             9       5               9       5 1.06817851D-006+C   23A
+C   23A      10       3-1.17067564D-006      10       5-5.13520819D-007
DMIG    M1             9       6               9       6 5.15957976D-007+C   24A
+C   24A      10       2 1.58484104D-006      10       6-3.75465684D-007
DMIG    M1            10       1              10       1 3.45066667D-005+C   25A
+C   25A      11       1 8.62666667D-006
DMIG    M1            10       2              10       2 3.45066667D-005+C   26A
+C   26A      11       2 8.62666667D-006      11       6-1.58484104D-006
DMIG    M1            10       3              10       3 3.45066667D-005+C   27A
+C   27A      11       3 8.62666667D-006      11       5 1.17067564D-006
DMIG    M1            10       4              10       4 4.01657600D-007+C   28A
+C   28A      11       4 1.00414400D-007
DMIG    M1            10       5              10       5 2.13635703D-006+C   29A
+C   29A      11       3-1.17067564D-006      11       5-5.13520819D-007
DMIG    M1            10       6              10       6 1.03191595D-006+C   30A
+C   30A      11       2 1.58484104D-006      11       6-3.75465684D-007
DMIG    M1            11       1              11       1 1.72533333D-005
DMIG    M1            11       2              11       2 1.72533333D-005+C   31A
+C   31A      11       6-2.72849229D-006
DMIG    M1            11       3              11       3 1.72533333D-005+C   32A
+C   32A      11       5 3.14265770D-006
DMIG    M1            11       4              11       4 2.00828800D-007
DMIG    M1            11       5              11       5 1.06817851D-006
DMIG    M1            11       6              11       6 5.15957976D-007
```

Next, we will discuss direct matrix import for the model shown in Figure 4-4 using the `DMIG` Bulk Data file generated in the previous example. Listing 4-9 contains the Model Input File.

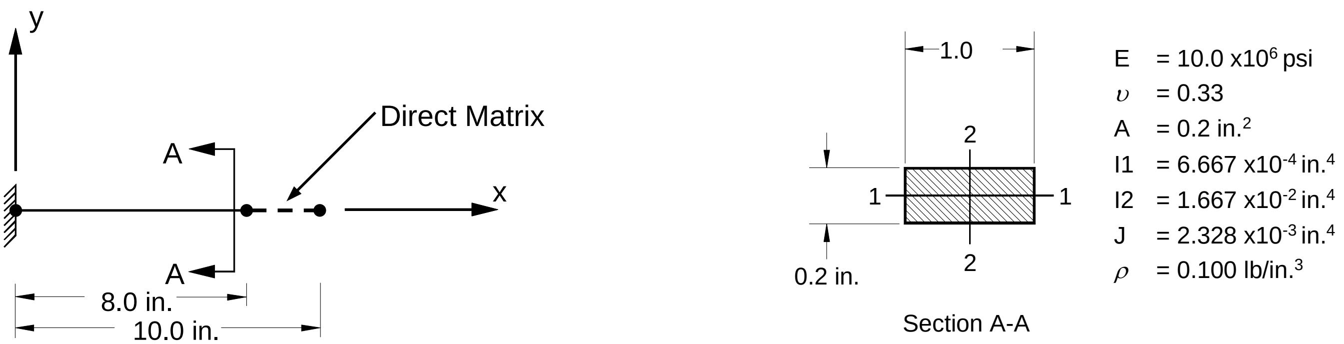


Figure 4-4. 2-D Cantilever Beam Example Problem with Direct Matrix Input.

The stiffness and mass matrixes are imported using the Case Control commands `K2GG` and `M2GG` respectively. Note that the stiffness and mass terms imported are in addition to any existing terms at the specified degree of freedom and are not replacements. Also, the mass matrix terms are not scaled by `PARAM`, `WTMASS` and must be in mass and not weight units.

Listing 4-9. Model Input File a 2-D Cantilever Beam with Direct Matrix Input.

```
$
$ MODAL SOLUTION.
$
SOL MODAL
CEND
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = VIBRATION OF A 2-D CANTILEVER BEAM
$
$ SPECIFY PREVIOUSLY GENERATED STIFFNESS AND MASS MATRIXES.
$
K2GG = K1
M2GG = M1
$
DISPLACEMENT = ALL
$
SUBCASE 1
 LABEL = NORMAL MODES ANALYSIS WITH DIRECT MATRIX INPUT
 SPC = 1
 METHOD = 1
```

Listing 4-9. Model Input File a 2-D Cantilever Beam with Direct Matrix Input. (Continued)

```
BEGIN BULK
$
$ INSERT DIRECT INPUT MATRIX DATA.
$
INCLUDE 'DMIGBGEN.BDF'
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  5
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  1.,  0.
CBAR,  2,  10,   2,   3,  0.,  1.,  0.
CBAR,  3,  10,   3,   4,  0.,  1.,  0.
CBAR,  4,  10,   4,   5,  0.,  1.,  0.
CBAR,  5,  10,   5,   6,  0.,  1.,  0.
CBAR,  6,  10,   6,   7,  0.,  1.,  0.
CBAR,  7,  10,   7,   8,  0.,  1.,  0.
CBAR,  8,  10,   8,   9,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ FIXED AT ONE END, MOVEMENT CONSTRAINED TO X-Y PLANE ONLY.
$
SPC1,  1,  123456,  1
SPC1,  1,     345,  1,  THRU,  11
ENDDATA
```

As expected this example yields results equivalent to the model shown in Figure 4-1. The models are equivalent with the only difference being elements 9 and 10 are represented directly using `DMIG` input data. The extracted frequencies are given in Listing 4-10.

Listing 4-10. Extracted Eigenvalues for a 2-D Cantilever Beam with Direct Matrix Input.

```
NORMAL MODES ANALYSIS WITH DIRECT MATRIX INPUT                      SUBCASE 1

                                                 R E A L   E I G E N V A L U E S

      MODE        EIGENVALUE       RADIANS          CYCLES        GENERALIZED      GENERALIZED      ORTHOGONALITY       ERROR
     NUMBER                                                           MASS          STIFFNESS           LOSS           MEASURE
        1        1.592103E+05    3.990117E+02    6.350468E+01    1.000000E+00    1.592103E+05    0.000000E+00    4.652420E-13
        2        6.247437E+06    2.499487E+03    3.978058E+02    1.000000E+00    6.247437E+06    1.005879E-14    1.069259E-12
        3        4.892945E+07    6.994959E+03    1.113282E+03    1.000000E+00    4.892945E+07    1.072059E-15    5.093375E-11
        4        1.877429E+08    1.370193E+04    2.180730E+03    1.000000E+00    1.877429E+08    4.822531E-16    1.670033E-09
        5        5.131852E+08    2.265359E+04    3.605431E+03    1.000000E+00    5.131852E+08    9.471590E-16    2.304874E-07
```

4.6 Model Reduction Using ASET

Model reduction provides a means for reducing model size by employing static condensation and the Guyan reduction methods. These can be especially useful when performing eigenvalue extraction especially when a large number of modes is required. The methods are approximate but can yield accurate results if used properly.

The basic dynamic equation before reduction and after `DMIG` import and single and multipoint constraints are applied is given by

$$\begin{bmatrix} M_{aa} & M_{ao} \\ M_{oa} & M_{oo} \end{bmatrix} \begin{Bmatrix} \ddot{u}_a \\ \ddot{u}_o \end{Bmatrix} + \begin{bmatrix} B_{aa} & B_{ao} \\ B_{oa} & B_{oo} \end{bmatrix} \begin{Bmatrix} \dot{u}_a \\ \dot{u}_o \end{Bmatrix} + \begin{bmatrix} K_{aa} & K_{ao} \\ K_{oa} & K_{oo} \end{bmatrix} \begin{Bmatrix} u_a \\ u_o \end{Bmatrix} = \begin{Bmatrix} P_a \\ P_o \end{Bmatrix}$$

where,

$\ddot{u}_a, \dot{u}_a, u_a$	are the displacements, velocities, and accelerations of the analysis set (a-set) to be retained.
$\ddot{u}_o, \dot{u}_o, u_o$	are the displacements, velocities, and accelerations of the omit set (o-set) to be eliminated.
M, B, K	are the mass, damping, and stiffness matrixes.
P_a, P_o	are the applied loads.

Note that all free-body motions must be included in the u_a partition otherwise K_{oo} will be singular.

The Guyan matrix G_o is given by:

$$[G_o] = -[K_{oo}]^{-1}[K_{oa}]$$

Using this we then can write the reduced global matrixes as:

$$[\overline{K}_{aa}] = [K_{aa}] + [K_{ao}][G_o]$$

$$[\overline{M}_{aa}] = [M_{aa}] + [M_{ao}][G_o] + [G_o]^T[M_{oa}] + [G_o]^T[M_{oo}][G_o]$$

Note that the reduced stiffness matrix is exact. The reduced mass matrix is approximated however. The omitted displacements are recovered using:

$$[u_o] = [G_o]\{u_a\}$$

As an example of `ASET` reduction we will use the beam shown in Figure 4-1 with the end constraint removed. We will retain only the degrees of freedom for the center and end grid points. Listing 4-11 contains the Model Input File. The `ASET` Bulk Data entry is used to specify which degrees of freedom are to be retained. Alternatively the `OMIT` entry may be used to specify which degrees of freedom are to be omitted.

Listing 4-11. Model Input File for the 2-D Cantilever Beam Problem with `ASET` Reduction.

```
$
$ MODAL SOLUTION.
$
SOL MODAL
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = RIGID BODY MODES OF AN ASET REDUCED 2-D UNCONSTRAINED BEAM
$
DISPLACEMENT = ALL
$
SUBCASE 1
 LABEL = NORMAL MODES
 SPC = 1
 METHOD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  5
$
$ ASET DEFINITION.  INCLUDE ALL DEGREES OF FREEDOM AT GRID POINTS 1, 6,
$ AND 11.
$
ASET,  1,  123456,  6,  123456,  11,  123456
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  1.,  0.
CBAR,  2,  10,   2,   3,  0.,  1.,  0.
CBAR,  3,  10,   3,   4,  0.,  1.,  0.
CBAR,  4,  10,   4,   5,  0.,  1.,  0.
CBAR,  5,  10,   5,   6,  0.,  1.,  0.
CBAR,  6,  10,   6,   7,  0.,  1.,  0.
CBAR,  7,  10,   7,   8,  0.,  1.,  0.
CBAR,  8,  10,   8,   9,  0.,  1.,  0.
CBAR,  9,  10,   9,  10,  0.,  1.,  0.
CBAR, 10,  10,  10,  11,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3,
,  -0.1,  0.5,  0.1,  0.5,  -0.1,  -0.5,  0.1,  -0.5
```

Listing 4-11. Model Input File for the 2-D Cantilever Beam Problem with `ASET` Reduction. (Continued)

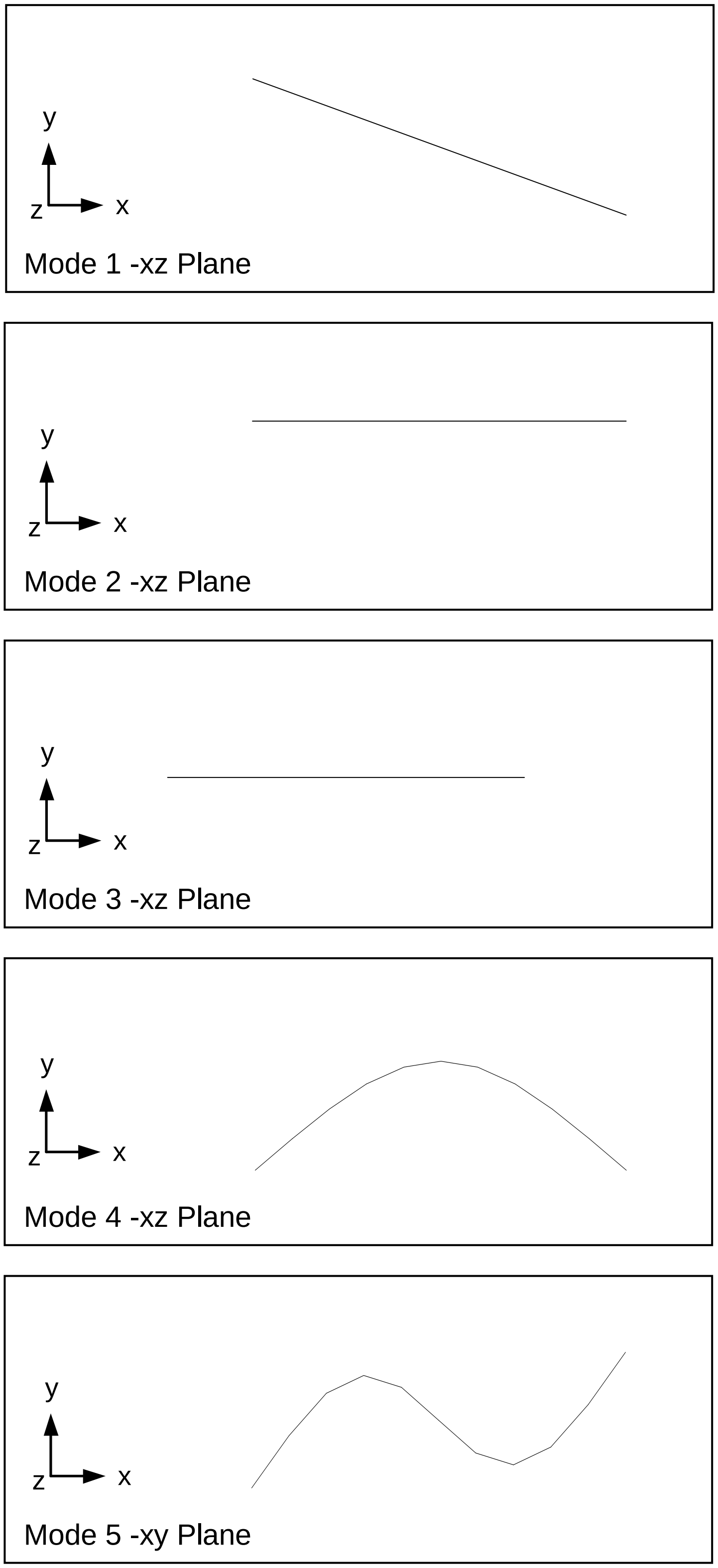
```
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ MOVEMENT CONSTRAINED TO X-Y PLANE ONLY.
$
SPC1,  1,  345,  1,  THRU,  11
ENDDATA
```

Listing 4-12 shows the extracted frequencies from the Model Results Output File. The mode shapes are plotted in Figure 4-5. As expected the results compare well to those of Section 4.4 for the unreduced model. The first 3 modes are rigid body and near zero in frequency. The forth mode is within 0.2 % of the full model and the fifth mode is within 13%. The same comparison using the diagonal mass formulation (`PARAM`, `COUPMASS`, `OFF`) yields similar differences.

Listing 4-12. Extracted Eigenvalues for a 2-D Unconstrained Beam with `ASET` Reduction.

```
NORMAL MODES                                                SUBCASE 1

                                        R E A L   E I G E N V A L U E S

     MODE        EIGENVALUE      RADIANS         CYCLES       GENERALIZED     GENERALIZED     ORTHOGONALITY      ERROR
    NUMBER                                                       MASS          STIFFNESS          LOSS          MEASURE
       1        8.032657E-08    2.834194E-04    4.510760E-05    1.000000E+00    8.032657E-08    0.000000E+00    1.462279E-15
       2        1.108274E-07    3.329075E-04    5.298387E-05    1.000000E+00    1.108274E-07    4.407430E-16    2.325812E-15
       3        2.777204E-06    1.666494E-03    2.652308E-04    1.000000E+00    2.777204E-06    6.553321E-18    5.254527E-14
       4        6.465693E+06    2.542773E+03    4.046949E+02    1.000000E+00    6.465693E+06    1.250137E-17    1.886853E-17
       5        6.318934E+07    7.949172E+03    1.265150E+03    1.000000E+00    6.318934E+07    2.220446E-16    4.428712E-17
```

Figure 4-5. Mode Shapes of a 2-D Unconstrained Beam with `ASET` Reduction.



4.7 Model Reduction Using Component Mode Synthesis (Craig-Bampton Reduction)

In Component Mode Synthesis (CMS) a structure is subdivided into components or substructures. Each component can be reduced independently and then combined in a separate analysis. Component modes synthesis can be regarded as an alternative to Guyan reduction, but unlike Guyan reduction accounts for both mass and stiffness making it more accurate.

The component is fixed at an interface and a normal modes analysis is performed where the constrained eigenvectors are given by

$$[\Phi] = [\Phi_1, \Phi_2, \Phi_3 \ldots \Phi_s]$$

CMS assumes that the dynamic behavior of the internal degrees of freedom can be correctly represented by a linear combination of the component modes.

$$\{u_{oo}\} = \sum_{i=1}^{s} \{\Phi_i\} q_i$$

where,

$$q = [q_1, q_2, q_3, \ldots q_s]^T$$

are the generalized degrees of freedom. The component displacements can then be represented by

$$u = \begin{Bmatrix} u_a \\ u_o \end{Bmatrix} = \begin{bmatrix} I & 0 \\ G_o & \Phi \end{bmatrix} \begin{Bmatrix} u_a \\ q \end{Bmatrix} = [H] \begin{Bmatrix} u_a \\ q \end{Bmatrix}$$

where $[H]$ is the Craig-Bampton transfer matrix. Substituting into the equation of motion we have

$$\begin{bmatrix} M_{aa} & M_{aq} \\ M_{qa} & M_{qq} \end{bmatrix} \begin{Bmatrix} \ddot{u}_a \\ \ddot{q} \end{Bmatrix} + \begin{bmatrix} B_{aa} & B_{aq} \\ B_{qa} & B_{qq} \end{bmatrix} \begin{Bmatrix} \dot{u}_a \\ \dot{q} \end{Bmatrix} + \begin{bmatrix} K_{aa} & K_{aq} \\ K_{qa} & K_{qq} \end{bmatrix} \begin{Bmatrix} u_a \\ q \end{Bmatrix} = \begin{Bmatrix} P_a \\ P_q \end{Bmatrix}$$

where,

$\ddot{u}_a, \dot{u}_a, u_a$	are the displacements, velocities, and accelerations of the analysis set (a-set) to be retained.
$\ddot{q}, \dot{q}, q$	are the generalized displacements, velocities, and accelerations of the modal degrees of freedom.
M, B, K	are the mass, damping, and stiffness matrixes.
P_a, P_q	are the applied loads.

The stiffness reduction is the same procedure as with the Guyan reduction shown in Section 4.6 with the addition of the modal degree of freedom terms which are simply the generalized stiffnesses for each component mode.

$$\left[\overline{K}_{aa}\right] = \left[K_{aa}\right] + \left[K_{ao}\right]\left[G_o\right]$$

$$\left[\overline{K}_{aq}\right] = \left[\overline{K}_{qa}\right] = 0$$

$$\left[\overline{K}_{qq}\right] = \begin{bmatrix} \ddots & & \\ & \omega_i^2 & \\ & & \ddots \end{bmatrix}$$

The mass and damping reduction procedure includes the off-diagonal modal degree of freedom terms

$$\left[\overline{M}_{aa}\right] = \left[M_{aa}\right] + \left[M_{ao}\right]\left[G_o\right] + \left[G_o\right]^T\left[M_{oa}\right] + \left[G_o\right]^T\left[M_{oo}\right]\left[G_o\right]$$

$$\left[\overline{M}_{aq}\right] = \left[\overline{M}_{qa}\right] = \left[M_{ao}\right]\left[\Phi\right] + \left[G_o\right]^T\left[M_{oo}\right]\left[\Phi\right]$$

$$\left[\overline{M}_{qq}\right] = \left[\Phi\right]^T\left[M_{oo}\right]\left[\Phi\right]$$

$$\left[\overline{B}_{aa}\right] = \left[B_{aa}\right] + \left[B_{ao}\right]\left[G_o\right] + \left[G_o\right]^T\left[B_{oa}\right] + \left[G_o\right]^T\left[B_{oo}\right]\left[G_o\right]$$

$$\left[\overline{B}_{aq}\right] = \left[\overline{B}_{qa}\right] = \left[B_{ao}\right]\left[\Phi\right] + \left[G_o\right]^T\left[B_{oo}\right]\left[\Phi\right]$$

$$\left[\overline{B}_{qq}\right] = \left[\Phi\right]^T\left[B_{oo}\right]\left[\Phi\right]$$

As an example of Craig-Bampton reduction we will use the beam shown in Figure 4-1 with only 2-dimensional constraints specified (end constraint removed). We will generate Craig-Bampton mass and stiffness matrixes in `DMIG` form for half the beam. We will then use these matrixes in a modal analysis which contains the other beam half and compare the results to full model and one using `ASET` reduction.

Listing 4-13 contains the Model Input File. The `ASET` Bulk Data entry is used to specify which degrees of freedom are on the component boundary. These degrees of freedom are fixed for the modal analysis phase of the reduction process. The `QSET` and `SPOINT` Bulk Data entries are used to define the number of component modes desired. Generally the more modes specified the better the accuracy at the cost of increased computation time. It is the presence of the `QSET` entry in the model that initiates the Craig-Bampton reduction sequence.

Listing 4-13. Model Input File for a 2-D Cantilever Beam with Craig-Bampton Reduction.

```
$
$ MODAL SOLUTION.
$
SOL MODAL
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = CRAIG-BAMPTON REDUCTION OF AN UNCONSTRAINED BEAM
$
DISPLACEMENT = ALL
$
$ EXPORT MASS AND STIFFNESS MATRIXES TO DMIG BULK DATA ENTRIES.
$
EXTSEOUT(DMIGBDF)
$
SUBCASE 1
 LABEL = CONSTRAINED COMPONENT NORMAL MODES
 METHOD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  5
$
$ ASET DEFINITION.  INCLUDE ALL DEGREES OF FREEDOM AT GRID POINT 6.
$
ASET,  6,  123456
$
$ QSET DEFINITION.  5 MODES REQUIRES 5 SCALAR POINTS.
$
SPOINT,  101,  THRU,  105
QSET1,  1,  101,  THRU,  105
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS, 5" SECTION
$ AND 5 ELEMENTS SPECIFIED FOR MATRIX REDUCTION).
$
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  6,  10,   6,   7,  0.,  1.,  0.
CBAR,  7,  10,   7,   8,  0.,  1.,  0.
CBAR,  8,  10,   8,   9,  0.,  1.,  0.
CBAR,  9,  10,   9,  10,  0.,  1.,  0.
CBAR, 10,  10,  10,  11,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
ENDDATA
```

Listing 4-14 shows the extracted constrained frequencies from the Model Results Output File. The `EXTSEOUT(DMIGBDF)` Case Control command exports the generated reduced stiffness and mass matrixes in `DMIG` format to the Bulk Data Output File. These matrixes are shown in Listing 4-15.

Listing 4-14. Extracted Constrained Eigenvalues for a 2-D Cantilever Beam with Craig-Bampton Reduction.

```
NORMAL MODES                                                              SUBCASE 1

                                                R E A L   E I G E N V A L U E S

    MODE          EIGENVALUE        RADIANS           CYCLES         GENERALIZED       GENERALIZED      ORTHOGONALITY       ERROR
   NUMBER                                                               MASS            STIFFNESS           LOSS           MEASURE
      1          2.546246E+06     1.595696E+03     2.539629E+02     1.000000E+00     2.546246E+06     0.000000E+00     5.291234E-14
      2          6.273196E+07     7.920351E+03     1.260563E+03     1.000000E+00     6.273196E+07     6.335606E-15     9.966381E-14
      3          9.972882E+07     9.986432E+03     1.589390E+03     1.000000E+00     9.972882E+07     1.521717E-17     2.728045E-15
      4          7.820284E+08     2.796477E+04     4.450731E+03     1.000000E+00     7.820284E+08     8.743006E-16     1.165661E-15
      5          1.445515E+09     3.801993E+04     6.051060E+03     1.000000E+00     1.445515E+09     1.659244E-23     2.314001E-15
```

Listing 4-15. Reduced Stiffness and Mass Matrix Export of a 2-D Cantilever Beam Segment.

```
$
$ OUTPUT PRODUCED BY ADS NASTRAN VERSION 10.3.0.716   12:35  01/16/15
$
DMIG    KR1            0       6       2
DMIG    KR1            6       1               6       1-9.31322575D-010
DMIG    KR1            6       2               6       2-2.91038305D-011
               6       6-8.73114914D-011
DMIG    KR1            6       3               6       3-2.32830644D-010
               6       5 5.82076609D-010
DMIG    KR1            6       4               6       4-1.81898940D-012
DMIG    KR1            6       5               6       5-1.86264515D-009
DMIG    KR1            6       6               6       6-2.40106601D-010
DMIG    KR1          101       1             101       1 2.54624579D+006
DMIG    KR1          102       1             102       1 6.27319636D+007
DMIG    KR1          103       1             103       1 9.97288152D+007
DMIG    KR1          104       1             104       1 7.82028398D+008
DMIG    KR1          105       1             105       1 1.44551530D+009
DMIG    MR1            0       6       2
DMIG    MR1            6       1               6       1 2.58800000D-004
             105       1 7.76365158D-019
DMIG    MR1            6       2               6       2 2.58800000D-004
               6       6 6.47000000D-004     101       1 1.25921130D-002
             102       1-6.30189647D-018     103       1-6.97650473D-003
             104       1 4.08126816D-003     105       1-4.62283039D-019
DMIG    MR1            6       3               6       3 2.58800000D-004
               6       5-6.47000000D-004     101       1-5.19640254D-018
             102       1-1.24952315D-002     103       1 3.05776715D-019
             104       1-6.24551630D-018
DMIG    MR1            6       4               6       4 3.01243200D-006
             105       1-1.56254658D-003
DMIG    MR1            6       5               6       5 2.17823765D-003
             101       1 1.90674625D-017     102       1 4.59311417D-002
             104       1 2.22252987D-017
DMIG    MR1            6       6               6       6 2.15752938D-003
             101       1 4.57613271D-002     102       1-2.13630605D-017
             103       1-7.30869802D-003     104       1 2.61355407D-003
DMIG    MR1          101       1             101       1 1.00000000D+000
DMIG    MR1          102       1             102       1 1.00000000D+000
DMIG    MR1          103       1             103       1 1.00000000D+000
DMIG    MR1          104       1             104       1 1.00000000D+000
DMIG    MR1          105       1             105       1 1.00000000D+000
```

Next, we will use direct matrix import for the model shown in Figure 4-6 using the `DMIG` Bulk Data file generated in the previous example. Listing 4-16 contains the Model Input File.

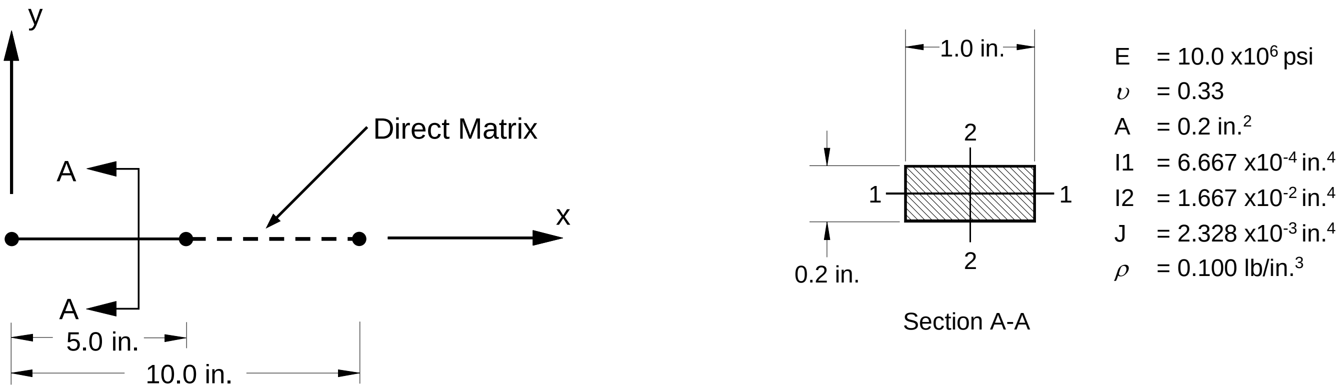


Figure 4-6. 2-D Cantilever Beam Example Problem with Craig-Bampton Direct Matrix Input.

The reduced stiffness and mass matrixes are imported using the Case Control commands `K2GG` and `M2GG` respectively.

Listing 4-16. Model Input File a 2-D Cantilever Beam with Craig-Bampton Direct Matrix Input.

```
$
$ MODAL SOLUTION.
$
SOL MODAL
CEND
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = RIGID BODY MODES OF A 2-D UNCONSTRAINED BEAM
$
$ SPECIFY PREVIOUSLY GENERATED STIFFNESS AND MASS MATRIXES.
$
K2GG = KR1
M2GG = MR1
$
DISPLACEMENT = ALL
$
SUBCASE 1
 LABEL = NORMAL MODES ANALYSIS WITH CRAIG-BAMPTON DIRECT MATRIX INPUT
 SPC = 1
 METHOD = 1
```

Listing 4-16. Model Input File a 2-D Cantilever Beam with Craig-Bampton Direct Matrix Input. (Continued)

```
BEGIN BULK
$
$ INSERT DIRECT INPUT MATRIX DATA.
$
INCLUDE 'CBDMIGGN.BDF'
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  5
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 5 ELEMENTS AND A CRAIG-BAMPTON
$ MASS AND STIFFNESS MATRIX).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
SPOINT,  101,  THRU,  105
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,  2,  0.,  1.,  0.
CBAR,  2,  10,   2,  3,  0.,  1.,  0.
CBAR,  3,  10,   3,  4,  0.,  1.,  0.
CBAR,  4,  10,   4,  5,  0.,  1.,  0.
CBAR,  5,  10,   5,  6,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ MOVEMENT CONSTRAINED TO X-Y PLANE ONLY.
$
SPC1,  1,  345,  1,  THRU,  6
ENDDATA
```

Listing 4-17 shows the extracted frequencies from the Model Results Output File. The results using 5 component modes are essentially identical to the full model with the only difference being elements 6 through 10 are represented directly using `DMIG` input data. The same comparison using the diagonal mass formulation (`PARAM`, `COUPMASS`, `OFF`) yields similar results.

Table 4-2 shows the effect on accuracy of the number of component modes specified. For this problem 3 modes would have provided acceptable accuracy.

Listing 4-17. Extracted Eigenvalues for a 2-D Cantilever Beam with Craig-Bampton Direct Matrix Input.

```
NORMAL MODES ANALYSIS WITH CRAIG-BAMPTON DIRECT MATRIX INPUT        SUBCASE 1

                                            R E A L   E I G E N V A L U E S

     MODE          EIGENVALUE        RADIANS           CYCLES         GENERALIZED      GENERALIZED      ORTHOGONALITY       ERROR
    NUMBER                                                                MASS          STIFFNESS           LOSS           MEASURE
       1          6.571645E-08    2.563522E-04    4.079972E-05    1.000000E+00    6.571645E-08    0.000000E+00    2.679496E-13
       2          2.839952E-07    5.329120E-04    8.481558E-05    1.000000E+00    2.839952E-07    7.397487E-15    1.419973E-13
       3          2.127956E-06    1.458751E-03    2.321675E-04    1.000000E+00    2.127956E-06    3.362438E-13    2.899843E-13
       4          6.437793E+06    2.537281E+03    4.038207E+02    1.000000E+00    6.437793E+06    4.250672E-14    3.341944E-14
       5          4.884240E+07    6.988734E+03    1.112291E+03    1.000000E+00    4.884240E+07    7.011752E-15    1.204301E-14
```

Table 4-2. Effect of Number of Component Modes on Craig-Bampton Matrix Accuracy for a Cantilever Beam.

	Mode 4 (Full Model = 403.8Hz)		Mode 5 (Full Model = 1112.2Hz)	
Number of Component Modes	Natural Frequency (Hz)	Difference (%)	Natural Frequency (Hz)	Difference (%)
1	405.9	0.5	1234.3	11.0
2	405.9	0.5	1234.3	11.0
3	403.9	0.0	1112.9	0.1
4	403.8	0.0	1112.3	0.0
5	403.8	0.0	1112.3	0.0

4.8 Model Reduction Using Superelements

The process defined in Section 4.7 can be completely automated using superelement analysis. This includes the generation of Craig-Bampton `DMIG` matrixes, their assembly, solution, and expansion of results data. The basic flow of a typical superelement analysis is shown in Figure 4-7.

The Initialization phase is carried out in the Model Translator module and consists of converting user defined superelement groups into `ASET` and `QSET` lists assigned to each superelement. In Autodesk Nastran superelements are defined either by specifying elements or their associated grid points. The following Case Control commands and Bulk Data entries can be used for this purpose.

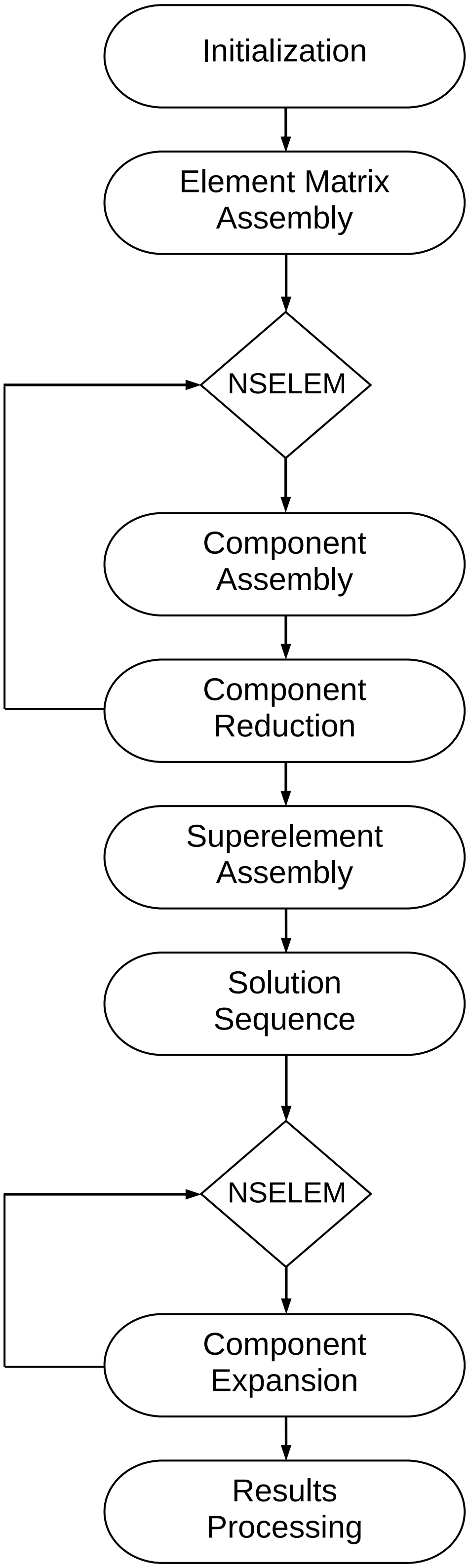
`SELEMGENERATE`	Case Control command to generate a superelement using either a grid point or element `SET` command
`SESET`	Bulk Data entry to define superelement interior grid points (field 9, SEID, of the `GRID` Bulk Data entry may be used for this same purpose)
`SEELT`	Bulk Data entry to define superelement interior elements
`SELABEL`	Bulk Data entry to define a label or name to be displayed in superelement output headings
`PARAM, NCBMODE, n`	Model parameter to specify the number of component (Craig-Bampton) modes for component reduction

See *Nastran Solver Reference Guide*, Section 3, *Case Control* and Section 4, *Bulk Data,* for more information.

The Element Matrix Assembly phase is carried out in the Geometry Processor module which generates the individual element stiffness and mass matrixes for all elements and stores them on disk. The Component Assembly phase is carried out in the Component Assembly Processor module which indexes the individual stiffness and mass matrixes into full component size. Component mass property output from the Grid Point Weight Generator is handled in this phase. The Component Reduction phase is carried out in the Matrix Reduction Processor module which reduces the full component using the methods described in Section 4-7 into a superelement. The Component Assembly and Component Reduction phases are repeated in succession for each superelement defined. Any `DMIG` matrixes specified are assigned to the last superelement and are not reduced. Each superelement reduced stiffness and mass matrix can be exported via the `EXTSEOUT(DMIGBDF)` Case Control command as they are generated. The Superelement Assembly phase is carried out in the Superelement Assembly Processor module which indexes the residual element stiffness and mass matrixes (elements not a member of any superelement) and the previously generated superelement stiffness and mass matrixes into full model size. The Solution Sequence phase is based on the `SOLUTION` specified. The solution sequences currently available for superelement analysis are:

Solution Character Variable	Solution Number
LINEAR STATIC	101
MODAL	103
MODAL FREQUENCY RESPONSE	111
MODAL TRANSIENT RESPONSE	112

Figure 4-7. Flow Diagram for a Typical Superelement Solution Sequence.



In the `LINEAR STATIC` solution sequence all superelement load vectors are assembled and reduced during the Component Reduction phase. The Component Expansion phase occurs in either the Solution Processor module (linear static solutions) or the Eigenvalue Processor module (modal solutions). In this phase internal superelement displacements are recovered to full model size enabling complete recovery of all vector and element results. The Results Processing phase is carried out in the Results Processor module in the same manner as non-superelement solution sequences using the expanded results data.

As an example of a superelement modal solution we will use the beam shown in Figure 4-1 with only 2-dimensional constraints specified (end constraint removed). We will generate two superelements by putting elements 1 through 3 in superelement 1 and elements 4 through 6 in superelement 2. The residual set will contain elements 7 through 10. We will then run a superelement modal analysis and compare the results to the full model and one using Craig-Bampton reduction.

Listing 4-18 contains the Model Input File. The `SELEMGENERATE` Case Control command is used to generate each superelements. Alternatively, we could have used the `SEELT` Bulk Data entry.

Listing 4-18. Model Input Model Input File for a 2-D Cantilever Beam with Superelement Reduction.

```
$
$ MODAL SOLUTION.
$
SOL MODAL
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = SUPERELEMENT REDUCTION OF AN UNCONSTRAINED BEAM
$
$ SUPERELEMENT DEFINITION.
$
SET 1 = 1, 2, 3
SELEMGENERATE, 1, ELEM, 1
$
SET 2 = 4, 5, 6
SELEMGENERATE, 2, ELEM, 2
$
DISPLACEMENT = ALL
$
$ EXPORT MASS AND STIFFNESS MATRIXES TO DMIG BULK DATA ENTRIES.
$
EXTSEOUT(DMIGBDF)
$
SUBCASE 1
 LABEL = SUPERELEMENT NORMAL MODES ANALYSIS
 SPC = 1
 METHOD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ NUMBER OF CRAIG-BAMPTON MODES.
$
PARAM, NCBMODE, 3
```

Listing 4-18. Model Input Model Input File for a 2-D Cantilever Beam with Superelement Reduction. (Continued)

```
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  8
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS, 3" SECTION
$ AND 3 ELEMENTS IN EACH SUPERELEMENT).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  1.,  0.
CBAR,  2,  10,   2,   3,  0.,  1.,  0.
CBAR,  3,  10,   3,   4,  0.,  1.,  0.
CBAR,  4,  10,   4,   5,  0.,  1.,  0.
CBAR,  5,  10,   5,   6,  0.,  1.,  0.
CBAR,  6,  10,   6,   7,  0.,  1.,  0.
CBAR,  7,  10,   7,   8,  0.,  1.,  0.
CBAR,  8,  10,   8,   9,  0.,  1.,  0.
CBAR,  9,  10,   9,  10,  0.,  1.,  0.
CBAR, 10,  10,  10,  11,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ MOVEMENT CONSTRAINED TO X-Y PLANE ONLY.
$
SPC1,  1,  345,  1,  THRU,  11
ENDDATA
```

Listing 4-19 shows the automatic `ASET` and `QSET` generation. The boundary for superelement 1 is grid point 4 and the boundary for superelement 2 is grid point 4 and 7. Each generates an `ASET` for all 6 degrees of freedom as shown. Grid points 1 through 3 in superelement 1 and 5 through 6 in superelement 2 are moved into the `OMIT` set. A `QSET` component is generated for each Craig-Bampton mode requested (`PARAM`, `NCBMODE`, `3`) for each superelement. Note that multipoint constraint equations, rigid elements, and interpolation elements must not cross a superelement boundary (i.e., reference grid points in different superelements).

Listing 4-20 shows the extracted constrained frequencies for each superelement from the Model Results Output File. The `EXTSEOUT(DMIGBDF)` Case Control command exports the generated reduced stiffness and mass matrixes in `DMIG` format to the Bulk Data Output File. These matrixes are shown in Listing 4-21.

Listing 4-19. `ASET` and `QSET` Generation for a 2-D Cantilever Beam with Superelement Reduction.

```
                         A N A L Y S I S   D E G R E E   O F   F R E E D O M   S E T   D E F I N I T I O N

     GRID        COMPONENT   SUPERELEMENT
      ID          NUMBER          ID
        4           1              1
        4           1              2
        4           2              1
        4           2              2
        4           3              1
        4           3              2
        4           4              1
        4           4              2
        4           5              1
        4           5              2
        4           6              1
        4           6              2
        7           1              2
        7           2              2
        7           3              2
        7           4              2
        7           5              2
        7           6              2

                      G E N E R A L I Z E D   D E G R E E   O F   F R E E D O M   S E T   D E F I N I T I O N

     GRID        COMPONENT   SUPERELEMENT
      ID          NUMBER          ID
       12           1              1
       13           1              1
       14           1              1
       15           1              2
       16           1              2
       17           1              2
```

Listing 4-20a. Extracted Constrained Eigenvalues for a 2-D Cantilever Beam with Superelement Reduction –Superelement 1.

```
 SUPERELEMENT 1

                                                R E A L   E I G E N V A L U E S

     MODE         EIGENVALUE       RADIANS          CYCLES         GENERALIZED      GENERALIZED      ORTHOGONALITY       ERROR
     NUMBER                                                            MASS          STIFFNESS           LOSS           MEASURE
        1        1.962878E+07    4.430438E+03    7.051260E+02    1.000000E+00    1.962878E+07    0.000000E+00    9.786391E-14
        2        4.712834E+08    2.170906E+04    3.455105E+03    1.000000E+00    4.712834E+08    2.744160E-29    5.509215E-15
        3        7.678557E+08    2.771021E+04    4.410217E+03    1.000000E+00    7.678557E+08    8.343960E-29    1.700471E-14
```

Listing 4-20b. Extracted Constrained Eigenvalues for a 2-D Cantilever Beam with Superelement Reduction –Superelement 2.

```
 SUPERELEMENT 2

                                                R E A L   E I G E N V A L U E S

     MODE         EIGENVALUE       RADIANS          CYCLES         GENERALIZED      GENERALIZED      ORTHOGONALITY       ERROR
     NUMBER                                                            MASS          STIFFNESS           LOSS           MEASURE
        1        7.988699E+08    2.826429E+04    4.498401E+03    1.000000E+00    7.988699E+08    0.000000E+00    9.886231E-16
        2        6.182106E+09    7.862637E+04    1.251378E+04    1.000000E+00    6.182106E+09    2.220446E-16    2.322149E-16
        3        1.743152E+10    1.320285E+05    2.101299E+04    1.000000E+00    1.743152E+10    7.642692E-16    1.752250E-16
```

Listing 4-21a. Reduced Stiffness and Mass Matrix Export of a 2-D Cantilever Beam Segment –Superelement 1.

```
$
$ OUTPUT PRODUCED BY ADS NASTARAN VERSION 10.3.0.716   02:02  01/16/15
$ SUPERELEMENT 1
$
$-------2-------3-------4-------5-------6-------7-------8-------9-------0-------
DMIG    KR1            0       6       2
DMIG    KR1            4       2               4       2 1.45519152D-011
               4       6-2.91038305D-011
DMIG    KR1            4       4               4       4 1.81898940D-012
DMIG    KR1            4       6               4       6 4.36557457D-011
DMIG    KR1           12       1              12       1 1.96287773D+007


$
$ OUTPUT PRODUCED BY ADS NASTARAN VERSION 10.3.0.716   02:02  01/16/15
$ SUPERELEMENT 1
$
$-------2-------3-------4-------5-------6-------7-------8-------9-------0-------
DMIG    MR1            0       6       2
DMIG    MR1            4       1               4       1 1.55280000D-004
DMIG    MR1            4       2               4       2 1.55280000D-004
               4       6-2.32920000D-004      12       1 9.74812934D-003
              14       1-5.39328367D-003
DMIG    MR1            4       3               4       3 1.55280000D-004
               4       5 2.32920000D-004      13       1-9.54336595D-003
DMIG    MR1            4       4               4       4 1.80745920D-006
              12       1 6.97418578D-020      13       1-9.08091199D-020
              14       1 1.36976183D-020
DMIG    MR1            4       5               4       5 4.78782588D-004
              13       1-2.14921797D-002
DMIG    MR1            4       6               4       6 4.66357626D-004
              12       1-2.12737503D-002      14       1 3.40205848D-003
DMIG    MR1           12       1              12       1 1.00000000D+000
DMIG    MR1           13       1              13       1 1.00000000D+000
DMIG    MR1           14       1              14       1 1.00000000D+000
```

Listing 4-21b. Reduced Stiffness and Mass Matrix Export of a 2-D Cantilever Beam Segment –Superelement 2.

```
$
$ OUTPUT PRODUCED BY ADS NASTARAN VERSION 10.3.0.716   02:02  01/16/15
$ SUPERELEMENT 2
$
$-------2-------3-------4-------5-------6-------7-------8-------9-------0-------
DMIG    KR2            0       6       2
DMIG    KR2            4       1               4       1 6.66666667D+005
               7       1-6.66666667D+005
DMIG    KR2            4       2               4       2 2.96311111D+003
               4       6 4.44466667D+003       7       2-2.96311111D+003
               7       6 4.44466667D+003
DMIG    KR2            4       3               4       3 7.40888889D+004
               4       5-1.11133333D+005       7       3-7.40888889D+004
               7       5-1.11133333D+005
DMIG    KR2            4       4               4       4 2.91729323D+003
               7       4-2.91729323D+003
DMIG    KR2            4       5               4       5 2.22266667D+005
               7       3 1.11133333D+005       7       5 1.11133333D+005
DMIG    KR2            4       6               4       6 8.88933333D+003
               7       2-4.44466667D+003       7       6 4.44466667D+003
DMIG    KR2            7       1               7       1 6.66666667D+005
DMIG    KR2            7       2               7       2 2.96311111D+003
               7       6-4.44466667D+003
```

Listing 4-21b. Reduced Stiffness and Mass Matrix Export of a 2-D Cantilever Beam Segment –Superelement 2. (Continued)

```
DMIG    KR2            7       3               7       3 7.40888889D+004
               7       5 1.11133333D+005
DMIG    KR2            7       4               7       4 2.91729323D+003
DMIG    KR2            7       5               7       5 2.22266667D+005
DMIG    KR2            7       6               7       6 8.88933333D+003
DMIG    KR2           15       1              15       1 7.98869913D+008
DMIG    KR2           16       1              16       1 6.18210607D+009
DMIG    KR2           17       1              17       1 1.74315232D+010
$
$ OUTPUT PRODUCED BY ADS NASTARAN VERSION 10.3.0.716   02:02  01/16/15
$ SUPERELEMENT 2
$
$-------2-------3-------4-------5-------6-------7-------8-------9-------0-------
DMIG    MR2            0       6       2
DMIG    MR2            4       1               4       1 5.17600000D-005
               7       1 2.58800000D-005      16       1 6.94322301D-020
              17       1-3.91305377D-019
DMIG    MR2            4       2               4       2 5.77444454D-005
               4       6 2.44183971D-005       7       2 1.98955546D-005
               7       6-1.44016029D-005      15       1-5.16246465D-003
              16       1 3.06779597D-003      17       1-1.79208025D-018
DMIG    MR2            4       3               4       3 5.94011070D-005
               4       5-2.48325625D-005       7       3 1.82388930D-005
               7       5 1.39874375D-005
DMIG    MR2            4       4               4       4 6.02486400D-007
               7       4 3.01243200D-007      15       1 1.40537237D-019
              16       1-3.91266194D-019      17       1-6.01241915D-004
DMIG    MR2            4       5               4       5 1.50353927D-005
               7       3-1.39874375D-005       7       5-1.04137053D-005
DMIG    MR2            4       6               4       6 1.33787311D-005
               7       2 1.44016029D-005       7       6-9.99953991D-006
              15       1-3.34544804D-003      16       1 1.19752480D-003
              17       1-6.95029984D-019
DMIG    MR2            7       1               7       1 5.17600000D-005
              16       1 7.15296911D-020      17       1-3.91306754D-019
DMIG    MR2            7       2               7       2 5.77444454D-005
               7       6-2.44183971D-005      15       1-5.16246465D-003
              16       1-3.06779597D-003      17       1 1.54363565D-018
DMIG    MR2            7       3               7       3 5.94011070D-005
               7       5 2.48325625D-005
DMIG    MR2            7       4               7       4 6.02486400D-007
              15       1 1.41141810D-019      16       1-4.03045286D-019
              17       1-6.01241915D-004
DMIG    MR2            7       5               7       5 1.50353927D-005
DMIG    MR2            7       6               7       6 1.33787311D-005
              15       1 3.34544804D-003      16       1 1.19752480D-003
              17       1-6.07077604D-019
DMIG    MR2           15       1              15       1 1.00000000D+000
DMIG    MR2           16       1              16       1 1.00000000D+000
DMIG    MR2           17       1              17       1 1.00000000D+000
```

Listing 4-22 shows the extracted frequencies for the reduced model from the Model Results Output File. The results using 3 component modes and 2 superelements are essentially identical to the full model. The same comparison using the diagonal mass formulation (`PARAM`, `COUPMASS`, `OFF`) yields similar results.

Table 4-3 shows the effect on accuracy of the number of component modes specified (`PARAM`, `NCBMODE`, `n`). For this problem 3 modes provide acceptable accuracy.

Listing 4-22. Extracted Eigenvalues for a 2-D Cantilever Beam with Superelement Reduction.

```
SUPERELEMENT NORMAL MODES ANALYSIS                                  SUBCASE 1

                                                 R E A L   E I G E N V A L U E S

     MODE         EIGENVALUE       RADIANS          CYCLES       GENERALIZED     GENERALIZED     ORTHOGONALITY       ERROR
    NUMBER                                                          MASS          STIFFNESS          LOSS          MEASURE
       1         7.357448E-08    2.712462E-04    4.317017E-05    1.000000E+00    7.357448E-08    0.000000E+00    3.463960E-14
       2         9.816140E-07    9.907643E-04    1.576850E-04    1.000000E+00    9.816140E-07    1.318187E-15    6.609803E-14
       3         3.338791E-06    1.827236E-03    2.908136E-04    1.000000E+00    3.338791E-06    9.216069E-15    6.681541E-14
       4         6.437427E+06    2.537208E+03    4.038093E+02    1.000000E+00    6.437427E+06    2.997602E-15    1.624684E-14
       5         4.885240E+07    6.989449E+03    1.112405E+03    1.000000E+00    4.885240E+07    9.436896E-16    2.177672E-14
```

Table 4-3. Effect of Number of Component Modes on Accuracy for a Cantilever Beam with Superelement Reduction.

	Mode 4 (Full Model = 403.8Hz)		Mode 5 (Full Model = 1112.2Hz)	
Number of Component Modes	**Natural Frequency (Hz)**	**Difference (%)**	**Natural Frequency (Hz)**	**Difference (%)**
1	403.8	0.0	1116.2	0.3
2	403.8	0.0	1115.9	0.3
3	403.8	0.0	1112.4	0.0
4	403.8	0.0	1112.4	0.0
5	403.8	0.0	1112.4	0.0

4.9 Modal Database Storage and Retrieval

When any modal analysis is performed the modal database (i.e., eigenvalues, eigenvectors, modal participation factors, etc.) can be saved and used for subsequent modal response solutions thus saving a substantial amount of time. The modal database is deleted by default but can be stored by setting `PARAM, MODALDATABASE` to `STORE`. A file with the same base name and location as the Model Input File will be generated with an .MDB extension. When `PARAM, MODALDATABASE` is set to `FETCH`, Autodesk Nastran will skip the eigenvalue extraction phase and load the modal database with the same Model Input File base name and an .MDB extension. The procedure is as follows:

1. Set `PARAM, MODALDATABASE, STORE` in the Model Input File for the modal analysis (normal modes or modal response) that is to be stored.
2. Run the modal analysis.
3. Set `PARAM, MODALDATABASE, FETCH` in the Model Input File for the modal response analysis that will use the modal database generated in Step 1.
4. Rename the modal database base name if different from the Model Input File for the modal response analysis.
5. Run the modal response analysis.

See the *Nastran Solver Reference Guide*, Section 5, *Parameters,* for more information on `MODALDATABASE`.

4.10 Modal Correlation

Modal correlation is often used to compare analytical and test results. Analyzing experimentally obtained mode shapes and comparing them with analysis results is critical in assessing the value of an analytical model and its interpretation. Other applications include comparing results from different analytical models or the same model with different analysis settings. Autodesk Nastran contains two built in tools for assessing modal correlation: Modal Assurance Criteria (MAC) and Modal Cross-Orthogonality (MXO). MAC gives quantitatively the global closeness between experimental and analysis mode shapes ignoring the effects of the system mass. The formula for MAC is given by

$$[\mathrm{MAC}] = \frac{\left[\{\phi\}_i^T \{\phi\}_j\right]^2}{\left[\{\phi\}_i^T \{\phi\}_i\right]\left[\{\phi\}_j^T \{\phi\}_j\right]}$$

where i and j corresponds to the indices of two mode shapes that can be from the same origin (experimental or analytical) in order to check linear dependency, or mixed in order to check correlation between the two model modal bases. MAC values oscillate between 0 and 1. A unitary value means perfect correlation. In general this situation does not appear, and a value greater than 0.8 is judged acceptable. Two corresponding modes will have a high degree of correlation.

MXO is similar to MAC except that the calculation is weighted by the global mass matrix. The formula for MXO is given by

$$[\mathrm{MXO}] = \{\phi\}_i^T [M] \{\phi\}_j$$

A generally accepted requirement for the MXO is to have all diagonal terms larger than 0.9 and all the off-diagonal terms less than 0.1.

As an example we will use the beam shown in Figure 4-1. The experimental data is given in MS Excel Comma Separate Variable (.CSV) format and shown in Listing 4-23. The format is

grid1, component1, eigenvector1
.
.
.
gridn, componentn, eigenvectorn
-1, mode, eigenvalue

where,

Option	Definition	Type
grid	Grid point identification number.	Integer > 0
component	Component number of global coordinate.	1 ≤ Integer ≤ 6
eigenvector	Eigenvector value for the specified grid point and component direction.	Real
mode	Associated mode number.	Integer > 0
eigenvalue	Associated eigenvalue.	Real
-1	Last record symbol for the mode specified.	

Listing 4-23. Experimental Data Input File for a 2-D Cantilever Beam with Modal Correlation.

```
$
$ MODE 1 EIGENDATA.
$
 3,  2,  -5.578773E+00
 5,  2,  -2.008938E+01
 7,  2,  -4.032251E+01
 9,  2,  -6.348129E+01
11,  2,  -8.756870E+01
 3,  6,  -5.298854E+00
 5,  6,  -8.941852E+00
 7,  6,  -1.105478E+01
 9,  6,  -1.193366E+01
11,  6,  -1.208716E+01
-1,  1,   1.577834E+05
$
$ MODE 2 EIGENDATA.
$
 3,  2,  -2.593419E+01
 5,  2,  -5.927000E+01
 7,  2,  -5.200510E+01
 9,  2,   4.389160E+00
11,  2,   8.550123E+01
 3,  6,  -2.007332E+01
 5,  6,  -9.052572E+00
 7,  6,   1.697939E+01
 9,  6,   3.701651E+01
11,  6,   4.206841E+01
-1,  2,   6.059634E+06
$
$ MODE 3 EIGENDATA.
$
 3,  2,   5.179274E+01
 5,  2,   4.744148E+01
 7,  2,  -3.837883E+01
 9,  2,  -3.749654E+01
11,  2,   8.184914E+01
 3,  6,   2.718869E+01
 5,  6,  -3.340254E+01
 7,  6,  -3.434102E+01
 9,  6,   3.778687E+01
11,  6,   6.996753E+01
-1,  3,   4.656183E+07
$
$ MODE 4 EIGENDATA.
$
 3,  2,  -6.494734E+01
 5,  2,   2.305299E+01
 7,  2,   3.424033E+01
 9,  2,  -5.633661E+01
11,  2,   7.588218E+01
 3,  6,  -7.294765E+00
 5,  6,   6.109471E+01
 7,  6,  -5.391942E+01
 9,  6,   5.490157E+00
11,  6,   9.886335E+01
-1,  4,   1.750925E+08
$
$ MODE 5 EIGENDATA.
$
 3,  2,  -5.848987E+01
 5,  2,   6.071392E+01
 7,  2,  -5.770473E+01
 9,  2,   4.372249E+01
11,  2,  -6.737308E+01
 3,  6,   3.676077E+01
 5,  6,  -3.013181E+00
 7,  6,  -2.726321E+01
 9,  6,   5.026109E+01
11,  6,  -1.256559E+02
-1,  5,   4.678855E+08
```

Listing 4-24 contains the Model Input File. A `MODAL` solution is required for modal correlation. The `XYPLOTCSVOUT` directive requests MAC and MXO plot file generation in MS Excel .CSV format. The `DATINFILE1` and `DATINFILE2` directives are used to specify the external input data associated with the two eigendata sets to be compared. The default for the `DATINFILE1` directive is the modal database for analysis being run. The `DATINFILE2` directive is used to specify the external or experimental data input file (Listing 4-23). You can also specify a standard Autodesk Nastran modal database file by simply using an .MDB extension. `DMIG` format is also supported where the `DMIG` Bulk Data entries are specified in the Model Input File and the `DMIG` name is specified using the `DATINFILE2` directive. The `CORRELATE` Case Control command requests the modal correlation. The `ESET` defines the degrees of freedom used in the comparison. External data may only exist at a few selected points as is often the case with experimental data. In our example only the y-displacement and z-rotation at grid points 3, 5, 7, 9, and 11 and selected.

While not shown in this example, the `XSETGENERATE` command can be used to generate the `ESET` from an external set of locations not common to existing nodes in the model. The input is typically an MS Excel .CSV file similar to the one in Listing 4-23. The actual experimental data locations are referenced in the model as grid points. The `XSETGENERATE` command can then be used to interpolate from the `XSET` to the `ESET`, where the `XSET` is the set of user defined experimental data points and the `ESET` is the corresponding closest model locations. See *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information on `XSETGENERATE`.

Listing 4-24. Model Input File for a 2-D Cantilever Beam with Modal Correlation.

```
$
$ REQUEST GENERATION OF MS EXCEL COMMA SEPERATED VARIABLE FILE.
$
NASTRAN XYPLOTCSVOUT=ON
$
$ SPECIFY INPUT DATA FILE.
$
NASTRAN DATINFILE2=MACBARIN.CSV
$
$ MODAL SOLUTION.
$
SOL 103
CEND
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = VIBRATION OF A 2-D CANTILEVER BEAM
$
DISPLACEMENT = ALL
$
$ REQUEST MODAL ASSURANCE CRITERIA (MAC) ANALYSIS.
$
CORRELATE = ALL
$
SUBCASE 1
 LABEL = NORMAL MODES
 SPC = 1
 METHOD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
```

Listing 4-24. Model Input File for a 2-D Cantilever Beam with Modal Correlation. (Continued)

```
$
$ DEFINE EXPERIMENT DATA DEGREES OF FREEDOM FOR MAC ANALYSIS.
$
ESET1,  26,  3,  5,  7,  9,  11
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  5
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  1.,  0.
CBAR,  2,  10,   2,   3,  0.,  1.,  0.
CBAR,  3,  10,   3,   4,  0.,  1.,  0.
CBAR,  4,  10,   4,   5,  0.,  1.,  0.
CBAR,  5,  10,   5,   6,  0.,  1.,  0.
CBAR,  6,  10,   6,   7,  0.,  1.,  0.
CBAR,  7,  10,   7,   8,  0.,  1.,  0.
CBAR,  8,  10,   8,   9,  0.,  1.,  0.
CBAR,  9,  10,   9,  10,  0.,  1.,  0.
CBAR, 10,  10,  10,  11,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ FIXED AT ONE END, MOVEMENT CONSTRAINED TO X-Y PLANE ONLY.
$
SPC1,  1,  123456,  1
SPC1,  1,     345,  1,  THRU,  11
ENDDATA
```

Listing 4-25 shows a comparison of the `ESET` analysis data (INPUT DATA-1) and external data (INPUT DATA-2). These should be checked for every analysis to confirm the input data format and the results of `XSETEGENERATE` if used to generate `ESET` external data.

Listing 4-25. MAC and MXO Output for a 2-D Cantilever Beam with Modal Correlation.

```
MODE = 1  EIGENVALUE = 1.592103E+05  CYCLES = 6.350468E+01

                                  R E A L   E S E T   E I G E N V E C T O R   N U M B E R   1
                                                    (INPUT DATA-1/INPUT DATA-2)

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID           ID
        3            0      0.000000E+00  9.062164E+00  0.000000E+00  0.000000E+00  0.000000E+00  8.605464E+00
                            0.000000E+00 -9.004914E+00  0.000000E+00  0.000000E+00  0.000000E+00 -8.553086E+00
        5            0      0.000000E+00  3.261689E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.450906E+01
                            0.000000E+00 -3.242705E+01  0.000000E+00  0.000000E+00  0.000000E+00 -1.443339E+01
        7            0      0.000000E+00  6.542818E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.791679E+01
                            0.000000E+00 -6.508613E+01  0.000000E+00  0.000000E+00  0.000000E+00 -1.784395E+01
        9            0      0.000000E+00  1.029355E+02  0.000000E+00  0.000000E+00  0.000000E+00  1.931338E+01
                            0.000000E+00 -1.024676E+02  0.000000E+00  0.000000E+00  0.000000E+00 -1.926258E+01
       11            0      0.000000E+00  1.418874E+02  0.000000E+00  0.000000E+00  0.000000E+00  1.953129E+01
                            0.000000E+00 -1.413480E+02  0.000000E+00  0.000000E+00  0.000000E+00 -1.951035E+01

 MODE = 2  EIGENVALUE = 6.247437E+06  CYCLES = 3.978058E+02

                                  R E A L   E S E T   E I G E N V E C T O R   N U M B E R   2
                                                    (INPUT DATA-1/INPUT DATA-2)

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID           ID
        3            0      0.000000E+00 -4.149925E+01  0.000000E+00  0.000000E+00  0.000000E+00 -3.203873E+01
                            0.000000E+00 -4.069137E+01  0.000000E+00  0.000000E+00  0.000000E+00 -3.149552E+01
        5            0      0.000000E+00 -9.422725E+01  0.000000E+00  0.000000E+00  0.000000E+00 -1.395225E+01
                            0.000000E+00 -9.299605E+01  0.000000E+00  0.000000E+00  0.000000E+00 -1.420370E+01
        7            0      0.000000E+00 -8.129033E+01  0.000000E+00  0.000000E+00  0.000000E+00  2.782967E+01
                            0.000000E+00 -8.159725E+01  0.000000E+00  0.000000E+00  0.000000E+00  2.664107E+01
        9            0      0.000000E+00  9.622213E+00  0.000000E+00  0.000000E+00  0.000000E+00  5.911329E+01
                            0.000000E+00  6.886697E+00  0.000000E+00  0.000000E+00  0.000000E+00  5.807979E+01
       11            0      0.000000E+00  1.378505E+02  0.000000E+00  0.000000E+00  0.000000E+00  6.592440E+01
                            0.000000E+00  1.341535E+02  0.000000E+00  0.000000E+00  0.000000E+00  6.600634E+01

 MODE = 3  EIGENVALUE = 4.892945E+07  CYCLES = 1.113282E+03

                                  R E A L   E S E T   E I G E N V E C T O R   N U M B E R   3
                                                    (INPUT DATA-1/INPUT DATA-2)

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID           ID
        3            0      0.000000E+00 -8.088203E+01  0.000000E+00  0.000000E+00  0.000000E+00 -4.174673E+01
                            0.000000E+00  7.889517E+01  0.000000E+00  0.000000E+00  0.000000E+00  4.141616E+01
        5            0      0.000000E+00 -7.045934E+01  0.000000E+00  0.000000E+00  0.000000E+00  5.427938E+01
                            0.000000E+00  7.226695E+01  0.000000E+00  0.000000E+00  0.000000E+00 -5.088163E+01
        7            0      0.000000E+00  6.330870E+01  0.000000E+00  0.000000E+00  0.000000E+00  5.078375E+01
                            0.000000E+00 -5.846194E+01  0.000000E+00  0.000000E+00  0.000000E+00 -5.231120E+01
        9            0      0.000000E+00  5.290025E+01  0.000000E+00  0.000000E+00  0.000000E+00 -6.330521E+01
                            0.000000E+00 -5.711796E+01  0.000000E+00  0.000000E+00  0.000000E+00  5.756022E+01
       11            0      0.000000E+00 -1.337538E+02  0.000000E+00  0.000000E+00  0.000000E+00 -1.050743E+02
                            0.000000E+00  1.246797E+02  0.000000E+00  0.000000E+00  0.000000E+00  1.065806E+02

 MODE = 4  EIGENVALUE = 1.877429E+08  CYCLES = 2.180730E+03

                                  R E A L   E S E T   E I G E N V E C T O R   N U M B E R   4
                                                    (INPUT DATA-1/INPUT DATA-2)

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID           ID
        3            0      0.000000E+00  9.635623E+01  0.000000E+00  0.000000E+00  0.000000E+00  7.850859E+00
                            0.000000E+00 -9.444559E+01  0.000000E+00  0.000000E+00  0.000000E+00 -1.060795E+01
        5            0      0.000000E+00 -4.016934E+01  0.000000E+00  0.000000E+00  0.000000E+00 -8.957171E+01
                            0.000000E+00  3.352336E+01  0.000000E+00  0.000000E+00  0.000000E+00  8.884314E+01
        7            0      0.000000E+00 -4.192051E+01  0.000000E+00  0.000000E+00  0.000000E+00  8.744400E+01
                            0.000000E+00  4.979185E+01  0.000000E+00  0.000000E+00  0.000000E+00 -7.840893E+01
        9            0      0.000000E+00  8.217738E+01  0.000000E+00  0.000000E+00  0.000000E+00 -2.310117E+01
                            0.000000E+00 -8.192398E+01  0.000000E+00  0.000000E+00  0.000000E+00  7.983716E+00
       11            0      0.000000E+00 -1.276606E+02  0.000000E+00  0.000000E+00  0.000000E+00 -1.406508E+02
                            0.000000E+00  1.103469E+02  0.000000E+00  0.000000E+00  0.000000E+00  1.437658E+02

 MODE = 5  EIGENVALUE = 5.131852E+08  CYCLES = 3.605431E+03

                                  R E A L   E S E T   E I G E N V E C T O R   N U M B E R   5
                                                    (INPUT DATA-1/INPUT DATA-2)

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID           ID
        3            0      0.000000E+00  7.299322E+01  0.000000E+00  0.000000E+00  0.000000E+00 -5.452114E+01
                            0.000000E+00 -7.338302E+01  0.000000E+00  0.000000E+00  0.000000E+00  4.612109E+01
        5            0      0.000000E+00 -7.697759E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.664104E+01
                            0.000000E+00  7.617338E+01  0.000000E+00  0.000000E+00  0.000000E+00 -3.780421E+00
        7            0      0.000000E+00  7.730498E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.798233E+01
                            0.000000E+00 -7.239796E+01  0.000000E+00  0.000000E+00  0.000000E+00 -3.420518E+01
        9            0      0.000000E+00 -6.610923E+01  0.000000E+00  0.000000E+00  0.000000E+00 -4.594246E+01
                            0.000000E+00  5.485545E+01  0.000000E+00  0.000000E+00  0.000000E+00  6.305896E+01
       11            0      0.000000E+00  1.102338E+02  0.000000E+00  0.000000E+00  0.000000E+00  1.564163E+02
                            0.000000E+00 -8.452814E+01  0.000000E+00  0.000000E+00  0.000000E+00 -1.576514E+02
```

Listing 4-26 shows the modal correlation MXO and MAC output matrixes. The MXO diagonal terms are all above 0.9 and the off-diagonal terms above 0.1 indicating good MXO correlation. The MAC diagonal terms are all above 0.8 indicating good MAC correlation. Figure 4-8 shows a graphical representation of these results.

Listing 4-26. MAC and MXO Output for a 2-D Cantilever Beam with Modal Correlation.

```
                                     M A S S   C R O S S - O R T H O G O N A L I T Y   M A T R I X

   MODE-I            MODE-J        ORTHOGONALITY
INPUT DATA-1       INPUT DATA-2          LOSS

       1                 1          9.955433E-01
                         2          2.685236E-02
                         3          3.895341E-02
                         4          5.220840E-02
                         5          6.206351E-02

       2                 2          9.847571E-01
                         3          4.791300E-02
                         4          6.548172E-02
                         5          6.725372E-02

       3                 3          9.734993E-01
                         4          5.053034E-02
                         5          9.509978E-02

       4                 4          9.560986E-01
                         5          2.933552E-02

       5                 5          9.110971E-01


MAXIMUM ORTHOGONALITY LOSS (DIAGONAL) =  9.955433E-01 FOR MODE 1
MINIMUM ORTHOGONALITY LOSS (DIAGONAL) =  9.110971E-01 FOR MODE 5

MAXIMUM ORTHOGONALITY LOSS (OFF-DIAGONAL) =  9.509978E-02 FOR MODE 3
MINIMUM ORTHOGONALITY LOSS (OFF-DIAGONAL) =  2.685236E-02 FOR MODE 1


                                     M O D A L   A S S U R A N C E   C R I T E R I A   M A T R I X

   MODE-I            MODE-J                MAC
INPUT DATA-1       INPUT DATA-2

       1                 1          9.999995E-01
                         2          1.074446E-01
                         3          8.049219E-02
                         4          7.299949E-02
                         5          5.895291E-02

       2                 2          9.996946E-01
                         3          1.512839E-01
                         4          1.059641E-01
                         5          1.251350E-01

       3                 3          9.975953E-01
                         4          1.609962E-01
                         5          1.039244E-01

       4                 4          9.908904E-01
                         5          1.682945E-01

       5                 5          9.757175E-01


MAXIMUM MODAL ASSURANCE CRITERION (DIAGONAL) =  9.999995E-01 FOR MODE 1
MINIMUM MODAL ASSURANCE CRITERION (DIAGONAL) =  9.757175E-01 FOR MODE 5

MAXIMUM MODAL ASSURANCE CRITERION (OFF-DIAGONAL) =  1.682945E-01 FOR MODE 4
MINIMUM MODAL ASSURANCE CRITERION (OFF-DIAGONAL) =  5.895291E-02 FOR MODE 1
```

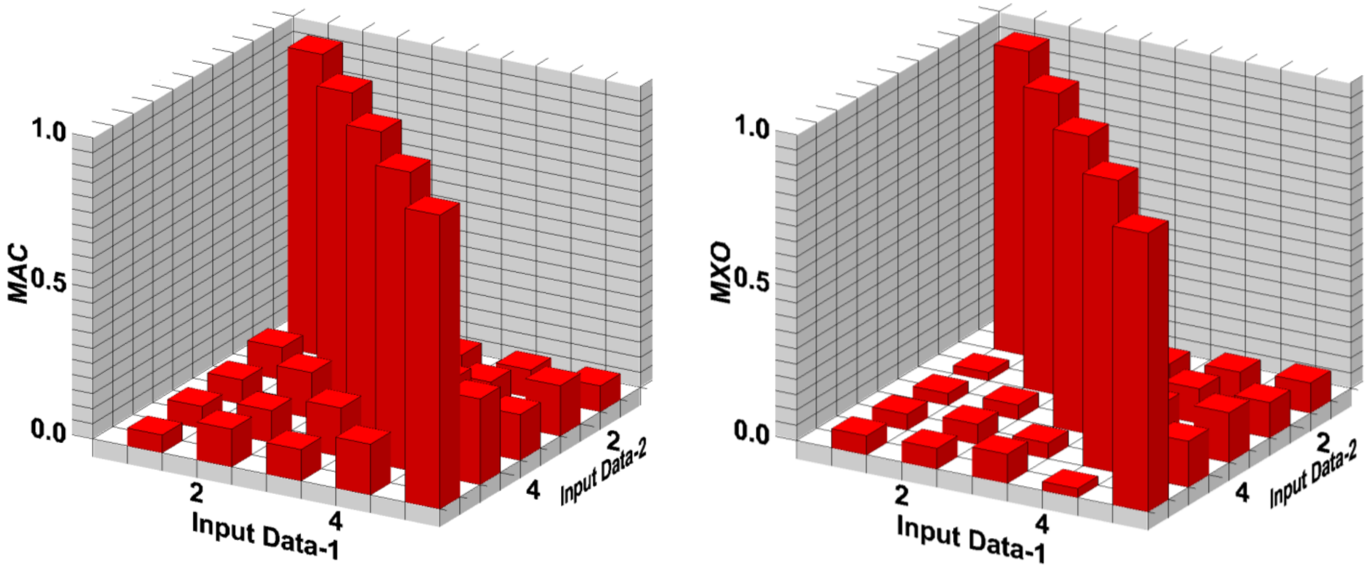


Figure 4-8. MAC and MXO 3-D Plots for a 2-D Cantilever Beam with Modal Correlation.

5. LINEAR TRANSIENT RESPONSE ANALYSIS

5.1 Introduction

Problems in structural dynamics can be divided into two broad areas. In one, the objective is to determine natural frequencies of vibration and the corresponding mode shapes. In the other, the objective is to determine how the structure behaves with time under an applied set of loads. In this section we examine the latter, which is termed transient response analysis. Unlike normal modes analysis, in transient response the applied loading and damping are not necessarily zero and loading can vary with time. Loading can be in the form of applied forces and/or enforced motions. Available grid point output includes: displacements, velocities, accelerations, and loading at each output time step. Available element output includes: energy, forces, and stresses at each output time step.

There are two methods available for performing linear transient response analysis: direct and modal. The direct method performs a numerical integration on the complete coupled equations of motion. The modal method uses the mode shapes of the structure to reduce and uncouple the equations of motion. The solution is then obtained through the summation of the individual modal responses. Generally the modal method is more efficient especially for larger models where a large number of time steps are specified. The direct method may be more efficient for models where high-frequency excitation require the extraction of a large number of modes. The direct method may also be more accurate because there are no mode truncation effects. If structural damping is used the direct method should be used.

Autodesk Nastran will also handle transient response of structures under initial stress, for example the forced vibration of a cable in tension. For more information see Section 8, *Linear Prestress Transient Response Analysis*.

5.1.1 Direct Transient Response Analysis

In direct transient response structural response is computed by solving a set of coupled equations using direct numerical integration. The method used is the same as for nonlinear transient response and allows for an adaptive time stepping algorithm. We begin with the dynamic equation of motion in matrix form:

$$[M]\{\ddot{u}(t)\}+[B]\{\dot{u}(t)\}+[K]\{u(t)\}=\{P(t)\}$$

The fundamental structural response (displacement) is solved at discrete times, typically with a fixed integration time step Δt. The damping matrix $[B]$ is used to represent energy dissipation characteristics of the structure. The damping matrix consists of several matrixes:

$$[B]=[B_1]+[B_2]+\alpha[K]+\beta[M]$$

$$[B_1]=[B_{DAMP}]+\frac{G}{W_3}[K]+\frac{1}{W_4}\sum G_{ELEM}K_{ELEM}$$

where,

$[B_1]$	damping from damping elements (`CVISC`, `CDAMPi`) and `B2GG DMIG`
$[B_2]$	damping from `B2PP DMIG`
$[K]$	global stiffness matrix
$[M]$	global mass matrix

$[K_{ELEM}]$	element stiffness matrix
G	overall structural damping coefficient (`PARAM`, `G`)
G_{ELEM}	element structural damping coefficient (`GE` on the `MATi` entry)
W_3	frequency of interest in radians per unit time (`PARAM`, `W3`) for the conversion of overall structural damping into equivalent viscous damping
W_4	frequency of interest in radians per unit time (`PARAM`, `W4`) for the conversion of element structural damping into equivalent viscous damping
α	Rayleigh damping stiffness matrix scale factor
β	Rayleigh damping mass matrix scale factor

Transient response does not permit the use of complex coefficients. Therefore, structural damping is included by means of equivalent viscous damping.

The viscous damping force is a damping force that is a function of a damping coefficient b and the velocity. It is an induced force that is represented in the equation of motion using the $[B]$ matrix and velocity vector.

$$[M]\{\ddot{u}(t)\}+[B]\{\dot{u}(t)\}+[K]\{u(t)\}=\{P(t)\}$$

The structural damping force is a displacement-dependent damping. The structural damping force is a function of a damping coefficient G and a complex component of the structural stiffness matrix.

$$[M]\{\ddot{u}(t)\}+(1+iG)[K]\{u(t)\}=\{P(t)\}$$

Assuming constant amplitude oscillatory response for a single degree of freedom system, the two damping forces are identical if

$$Gk = b\omega$$

or

$$b=\frac{Gk}{\omega}$$

Therefore, if structural damping G is to be modeled using equivalent viscous damping b, then the equality holds at only one frequency (see Figure 5-1).

Two parameters are used to convert structural damping to equivalent viscous damping. An overall structural damping coefficient can be applied to the entire system stiffness matrix using `PARAM`, `W3`, `r` where `r` is the circular frequency at which damping is made equivalent. This parameter is used along with `PARAM`, `G`. The default for `W3` is zero, which results in damping from this source to be ignored in transient analysis.

`PARAM`, `W4` is an alternate parameter used to convert element structural damping to equivalent viscous damping. `PARAM`, `W4`, `r` is used where r is the circular frequency at which damping is to be made equivalent. `PARAM`, `W4` is used along with the GE field on the `MATi` entry. The default for `W4` is zero, which results in damping from this source to be ignored in transient analysis.

Units for `PARAM`, `W3` and `PARAM`, `W4` are in radians per unit time. The choice of `W3` or `W4` is typically the dominant frequency at which damping is active. Often, the first natural frequency is selected, but isolated individual element damping can occur at different frequencies and can be handled by the appropriate data entries.

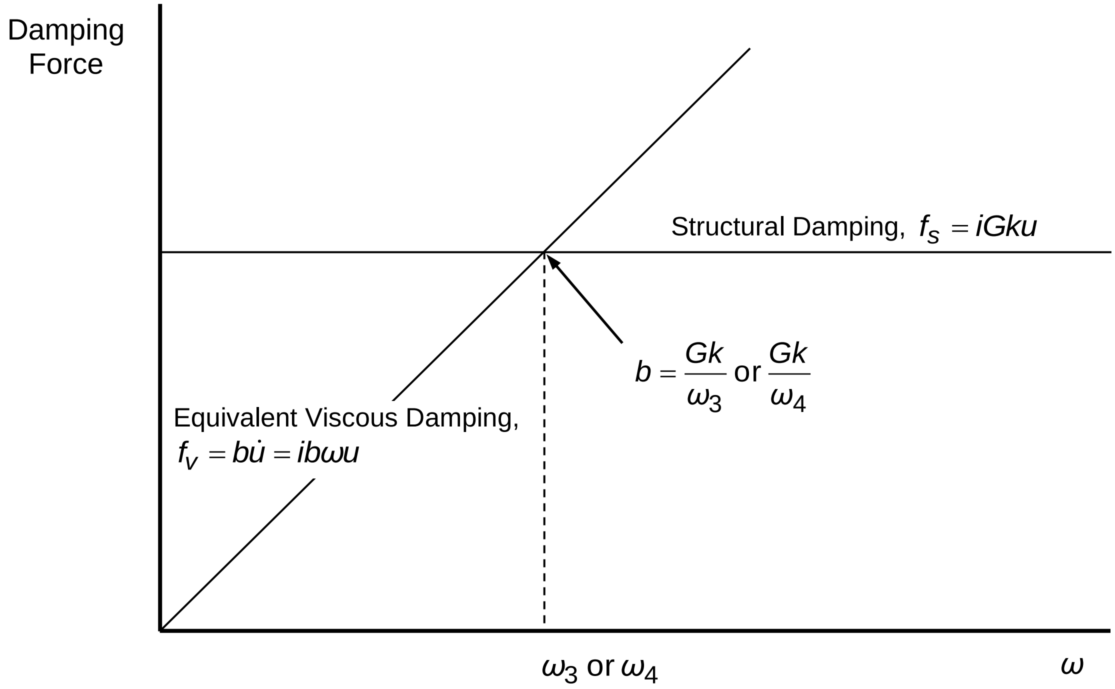


Figure 5-1. Structural Damping Versus Viscous Damping.

5.1.2 Modal Transient Response Analysis

Modal transient response analysis uses the mode shapes of the structure to reduce the size, uncouple the equations of motion, and make numerical integration more efficient. Since the mode shapes are typically computed as part of the characterization of the structure, modal transient response is a natural extension of normal modes analysis.

To outline the procedure we first look at the general equation of equilibrium for a finite element system in motion:

$$[M]\{\ddot{u}(t)\} + [B]\{\dot{u}(t)\} + [K]\{u(t)\} = \{P(t)\}$$

where,

$[K]$ is the global stiffness matrix

$[M]$ is the global mass matrix

$[B]$ is the global damping matrix

$\{P\}$ is the global load vector

$\{\ddot{u}\}$ is the global acceleration vector

$\{\dot{u}\}$ is the global velocity vector

$\{u\}$ is the global displacement vector

The transformation from physical coordinates $\{u\}$ to modal coordinates $\{\xi\}$ is given by:

$$\{u(t)\} = [\phi]\{\xi(t)\}$$

The mode shapes $[\phi]$ are used to transform the problem in terms of the behavior of the modes as opposed to the behavior of the grid points.

If we assume modal damping is used, we can rewrite the general equation of equilibrium as:

$$[M][\phi]\{\ddot{\xi}(t)\} + [B][\phi]\{\dot{\xi}(t)\} + [K][\phi]\{\xi(t)\} = P(t)$$

which is now the equation of motion in terms of modal coordinates. To uncouple the equations, premultiply by $[\phi]^T$ to obtain:

$$[\phi]^T[M][\phi]\{\ddot{\xi}(t)\} + [\phi]^T[B][\phi]\{\dot{\xi}(t)\} + [\phi]^T[K][\phi]\{\xi(t)\} = [\phi]^T P(t)$$

where

$[\phi]^T[M][\phi]$ [is the modal or generalized mass matrix

$[\phi]^T[K][\phi]$ is the modal or generalized stiffness matrix

$[\phi]^T[B][\phi]$ is the modal damping matrix

$[\phi]^T[P]$ is the modal force vector

Using the orthogonality property of the mode shapes we can formulate the equations of motion in terms of the diagonal generalized mass, stiffness, and damping (modal damping). Since these matrices do not have off-diagonal terms that couple the equations of motion, the modal equations of motion are uncoupled. The equations of motion can then be written as:

$$m_i\ddot{\xi}_i + b_i\dot{\xi}_i(t) + k_i\xi_i(t) = p_i(t)$$

where,

m_i = i-th modal mass

b_i = i-th modal damping

k_i = i-th modal stiffness

p_i = i-th modal force

ξ_i = i-th modal degree of freedom

The above equation can also be written as

$$\ddot{\xi}_i + 2\zeta_i\omega_i\dot{\xi}_i(t) + \omega_i^2\xi_i(t) = \frac{1}{m_i}p_i(t)$$

where,

$\zeta_i = b_i/(2m_i\omega_i) \equiv$ modal damping ratio

$\omega_i^2 = k_i/m_i \equiv$ modal frequency

The physical responses are then recovered from the summation of the individual modal responses using

$$\{u(t)\} = [\phi]\{\xi(t)\}$$

5.2 How to Setup a Model Input File for Transient Response Analysis

5.2.1 Direct Transient Response

In Autodesk Nastran you can perform direct transient response analysis by setting `SOLUTION = DIRECT TRANSIENT RESPONSE` in the Model Initialization File or by specifying `SOL 109` or `SOL DIRECT TRANSIENT RESPONSE` above the Case Control Section in the Model Input File. Multiple subcases can be specified, each requesting a different constraint, load, or output set. Time stepping is continuous from one subcase to the next. Adaptive time stepping is available in direct transient response solutions in Autodesk Nastran. See the `TSTEP` entry in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for more information.

5.2.2 Modal Transient Response

In Autodesk Nastran you can perform modal transient response analysis by setting `SOLUTION = MODAL TRANSIENT RESPONSE` in the Model Initialization File or by specifying `SOL 112` or `SOL MODAL TRANSIENT RESPONSE` above the Case Control Section in the Model Input File. Multiple subcases can be specified, each requesting a different load or output set. Time stepping is continuous from one subcase to the next. Only one reference to an `EIGRL` Bulk Data entry (`METHOD` Case Control command) is permitted. This request should be placed above the first subcase.

5.2.3 Transient Load Definition

Setting up a transient response analysis can be challenging due to flexibility permitted in defining the transient loading. The following Bulk Data entries are used to define the dynamic loading:

`TLOAD1`	Tabular input
`TLOAD2`	Analytical function
`DAREA`	Spatial distribution of dynamic load
`TABLEDi`	Tabular values versus time
`LSEQ`	Generates the spatial distribution of dynamic loads from static load entries
`DLOAD`	Combines dynamic load sets
`DELAY`	Time delay

See *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for more information on each entry.

Dynamic load definition can be divided into two parts, one being the location and the other being the time variation of the loading. Spatial distribution is the characteristic which defines the location of the loading. Temporal distribution is the characteristic which defines the time variation. The complete dynamic load definition consists of the product of spatial and temporal distributions. This method of defining dynamic loading allows you to combine simple loadings to create complicated loading distributions that vary in position as well as time. Figure 5-2 describes pictorially the relationships between static and dynamic load Case Control commands and Bulk Data entries.

5.2.4 Integration Time Step

The `TSTEP` Bulk Data entry controls the integration time step, the duration of the solution, and which time steps are output. The `TSTEP` entry is selected by the `TSTEP` Case Control command. The integration time step must be small enough to accurately represent the variation in the loading. Additionally, it must be small enough to represent the maximum frequency of interest. A good rule-of-thumb is to use at least 10 solution time steps per period of response for the maximum frequency of interest (cutoff frequency).

Figure 5-2a. Relationship of Dynamic and Static Loads –with `LOADSET` and `LSEQ` Reference.

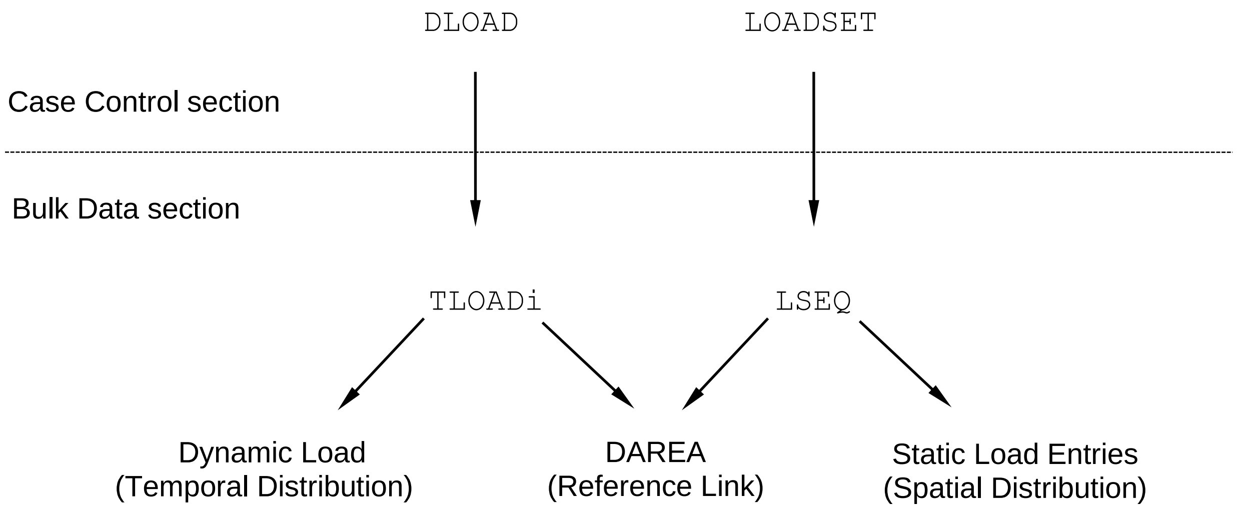
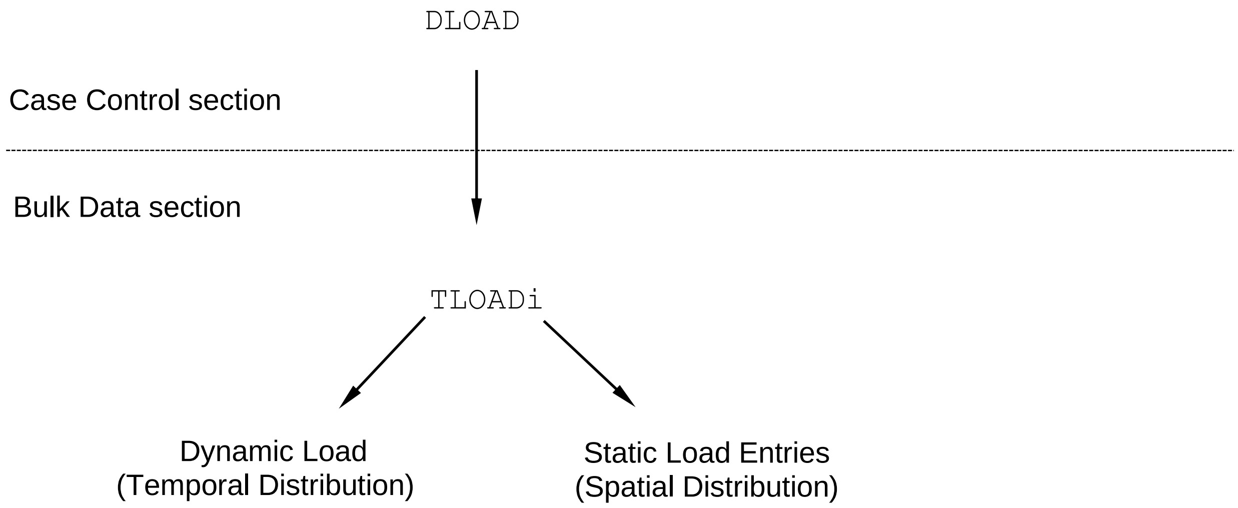


Figure 5-2b. Relationship of Dynamic and Static Loads –without `LOADSET` and `LSEQ` Reference.



5.2.5 Dynamic Data Recovery

A transient response analysis can produce very large amounts of output data since there are usually a large number of time steps involved for a given solution. There are several options available for recovering and storing this data. For data recovery, results can be calculated using one of two methods: mode displacement method and matrix method.

The mode displacement method calculates element results from the global displacement vector in physical coordinates for every time step. The number of operations is proportional to the number of time steps.

The matrix method calculates element results from the global displacement vector produced for each mode shape during eigenvalue extraction. Then the results for each time step are computed as the sum of the modal responses. The number of operations is proportional to the number of modes used.

Since the number of modes is typically much less than the number of time steps, the matrix method is usually more efficient. The `DYNRSLTMETHOD` Model Initialization directive controls these operations. The default for this directive is `AUTO`, which allows the program to choose which method is most efficient based on the number of modes versus time steps. See the *Nastran Solver Reference Guide*, Section 2, *Initialization,* for more information.

For storing and importing results into a post-processor, for example FEMAP®, it is recommended that the Model Initialization directive `RSLTFILETYPE` be set to `FEMAPBINARY`. This will produce a single, binary results neutral file which will contain all results data for each time step.

5.3 Interpreting Results

In this section we will present several examples demonstrating the features and capabilities of transient response analysis. For all examples we will use the cantilever beam shown in Figure 5-3 with a `MODAL TRANSIENT RESPONSE` solution. For the first problem, it is desired to find the response of the beam to an impulse load applied at the beam free end. Three different levels of damping are used: no damping, 5% critical damping, and 100% critical damping. Listing 5-1 contains the Model Input Files for each case.

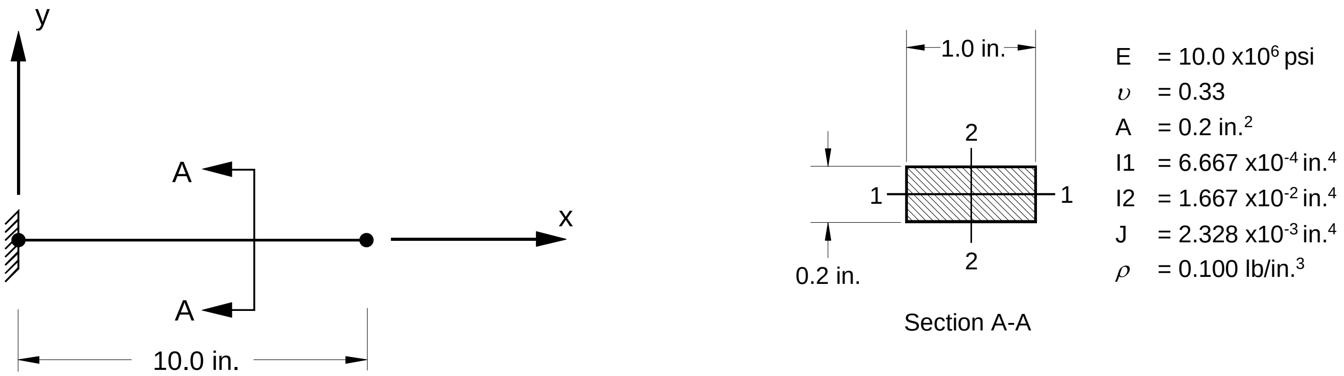


Figure 5-3. 2-D Cantilever Beam Example Problem.

Listing 5-1a. Model Input File for the 2-D Cantilever Beam Problem with a Time-Dependent Point Load at the Free End and No Damping.

```
$
$ MODAL TRANSIENT RESPONSE SOLUTION - NO DAMPING.
$
SOL MODAL TRANSIENT RESPONSE
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = FORCED DYNAMIC RESPONSE - NO DAMPING
$
DISPLACEMENT = ALL
$
TSTEP = 25
LOADSET = 10
METHOD = 1
SPC = 1
SUBCASE 1
 LABEL = 10 LB EDGE LOAD IN Z-DIRECTION
 DLOAD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  25
$
$ DEFINE 0.06 SECONDS OF RESPONSE.
$
TSTEP,  25,  600,  0.0001,  1
$
$ DEFINE LOADING.
$
DLOAD,  1,  1.,  1.,  11
$
$ DEFINE TIME-DEPENDENT LOADING.
$
TLOAD1,  11,  100,  ,  ,  10
TABLED1,  10,
,  0.,  0.,  0.001,  1.,  100.,  1.,  ENDT
$
$ 10 LB POINT LOAD IN Z-DIRECTION AT FREE END.
$
FORCE,  1,  11,  0,  10.,  0.,  0.,  1.
LSEQ,  10,  100,  1
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
```

Listing 5-1a. Model Input File for the 2-D Cantilever Beam Problem with a Time-Dependent Point Load at the Free End and No Damping. (Continued)

```
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  1.,  0.
CBAR,  2,  10,   2,   3,  0.,  1.,  0.
CBAR,  3,  10,   3,   4,  0.,  1.,  0.
CBAR,  4,  10,   4,   5,  0.,  1.,  0.
CBAR,  5,  10,   5,   6,  0.,  1.,  0.
CBAR,  6,  10,   6,   7,  0.,  1.,  0.
CBAR,  7,  10,   7,   8,  0.,  1.,  0.
CBAR,  8,  10,   8,   9,  0.,  1.,  0.
CBAR,  9,  10,   9,  10,  0.,  1.,  0.
CBAR, 10,  10,  10,  11,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3,
,  -0.1,  0.5,  0.1,  0.5,  -0.1,  -0.5,  0.1,  -0.5
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ FIXED AT ONE END, MOVEMENT CONSTRAINED TO X-Z PLANE ONLY.
$
SPC1,  1,  123456,  1
SPC1,  1,     246,  1,  THRU,  11
ENDDATA
```

Listing 5-1b. Model Input File for the 2-D Cantilever Beam Problem with a Time-Dependent Point Load at the Free End and 5% Critical Damping.

```
$
$ TRANSIENT RESPONSE SOLUTION - 5% CRITICAL DAMPING.
$
SOL MODAL TRANSIENT RESPONSE
$
DISPLACEMENT = ALL
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = FORCED DYNAMIC RESPONSE - 5% CRITICAL DAMPING
$
SPC = 1
SDAMP = 20
TSTEP = 25
LOADSET = 10
METHOD = 1
$
SUBCASE 1
 LABEL = 10 LB EDGE LOAD IN Z-DIRECTION
 DLOAD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  25
$
$ DEFINE 0.06 SECONDS OF RESPONSE.
$
TSTEP,  25,  600,  0.0001,  1
$
$ DEFINE LOADING.
$
DLOAD,  1,  1.,  1.,  11
$
$ DEFINE TIME-DEPENDENT LOADING.
$
TLOAD1,  11,  100,  ,  ,  10
TABLED1,  10,
,  0.,  0.,  0.001,  1.,  100.,  1.,  ENDT
$
$ 10 LB POINT LOAD IN Z-DIRECTION AT FREE END.
$
FORCE,  1,  11,  0,  10.,  0.,  0.,  1.
LSEQ,  10,  100,  1
$
$ 5% CRITICAL DAMPING.
$
TABDMP1,  20,  CRIT,
,  1.,  0.05,  10000.,  0.05,  ENDT
$
$ INSERT BASIC MODEL (SEE LISTING 5-1a).
$
ENDDATA
```

Listing 5-1c. Model Input File for the 2-D Cantilever Beam Problem with a Time-Dependent Point Load at the Free End and 100% Critical Damping.

```
$
$ TRANSIENT RESPONSE SOLUTION - 100% CRITICAL DAMPING.
$
SOL MODAL TRANSIENT RESPONSE
$
DISPLACEMENT = ALL
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = FORCED DYNAMIC RESPONSE - 100% CRITICAL DAMPING
$
SPC = 1
SDAMP = 20
TSTEP = 25
LOADSET = 10
METHOD = 1
$
SUBCASE 1
 LABEL = 10 LB EDGE LOAD IN Z-DIRECTION
 DLOAD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  25
$
$ DEFINE 0.06 SECONDS OF RESPONSE.
$
TSTEP,  25,  600,  0.0001,  1
$
$ DEFINE LOADING.
$
DLOAD,  1,  1.,  1.,  11
$
$ DEFINE TIME-DEPENDENT LOADING.
$
TLOAD1,  11,  100,  ,  ,  10
TABLED1,  10,
,  0.,  0.,  0.001,  1.,  100.,  1.,  ENDT
$
$ 10 LB POINT LOAD IN Z-DIRECTION AT FREE END.
$
FORCE,  1,  11,  0,  10.,  0.,  0.,  1.
LSEQ,  10,  100,  1
$
$ 100% CRITICAL DAMPING.
$
TABDMP1,  20,  CRIT,
,  1.,  1.,  10000.,  1.,  ENDT
$
$ INSERT BASIC MODEL (SEE LISTING 5-1a).
$
ENDDATA
```

In each example, the spatial definition of the dynamic load vector is defined using a static point load (`FORCE`) applied in the z-direction at the free end of the beam. This load is then referenced by a load sequence entry (`LSEQ`), which references an area factor (`DAREA`) that serves as the reference link between static and dynamic load definitions. The time-dependent dynamic load (`TLOAD1`) then references the area factor defined by the `LSEQ` entry for spatial definition (area) and a `TABLED1` for temporal definition (time). The `DLOAD` Bulk Data entry is used to combine and scale dynamic loads defined using the `TLOADi` Bulk Data entries. The `DLOAD` and `LSEQ` Bulk Data entries are called out in the Case Control Section using the `DLOAD` and `LOADSET` Case Control commands, respectively. The resulting load time history is shown graphically in Figure 5-4. Note that a `DLOAD` Case Control command can directly call out a `TLOADi` Bulk Data entry, which is not shown in these examples.

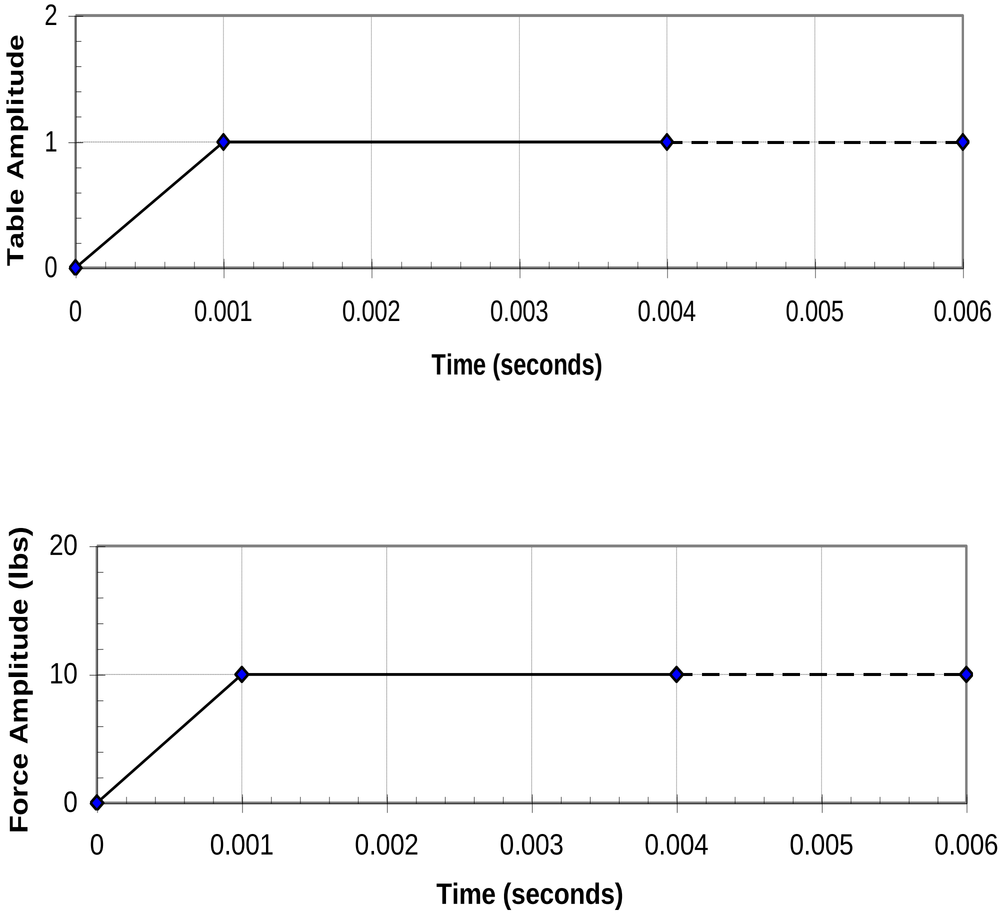


Figure 5-4. Time History from the `TABLED1` Entry (top) and Resulting Applied Load (bottom).

The `TSTEP` Bulk Data entry is used to control the duration of the solution and which time steps are output. It is called out in the Case Control Section using the `TSTEP` Bulk Data entry. In these examples we have requested 600 time steps with a time increment of 0.0001 seconds and output every time step.

Figure 5-5 shows the response at the beam free end (grid point 11) for each case. As expected for the undamped case, the beam vibrates with a constant amplitude at its resonant frequency about the deflected shape produced by the 10 pound end load. For the partially damped case, oscillation occurs with decreasing magnitude until the motion has dampened out at 0.060 seconds. Again, the beam is deflected 0.020 inches at the free end, exactly the same as predicted by beam theory. For the critically damped case, no oscillation occurs and the beam again assumes a deflected shape.

Figure 5-5a. Tip Displacement of a 2-D Cantilever Beam with No Damping.

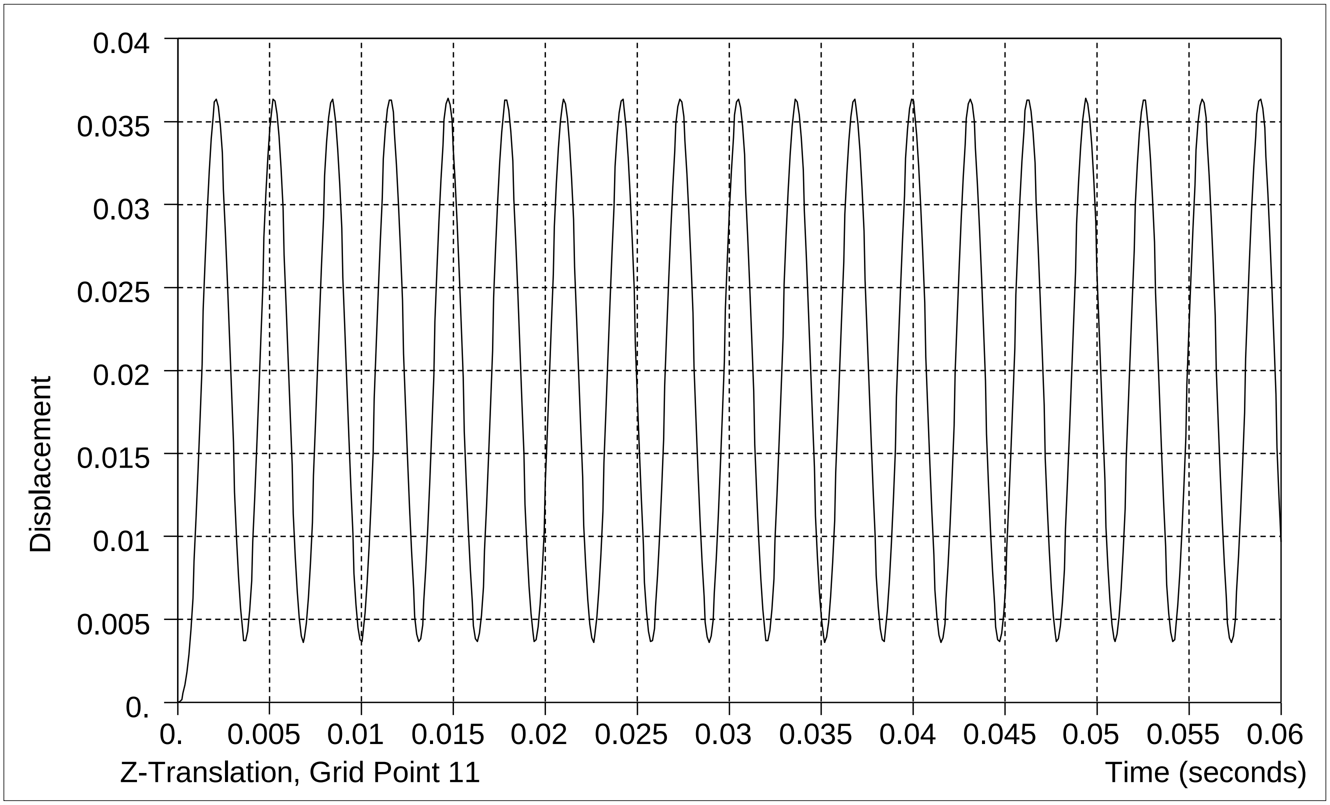


Figure 5-5b. Tip Displacement of a 2-D Cantilever Beam with 5% Critical Damping.

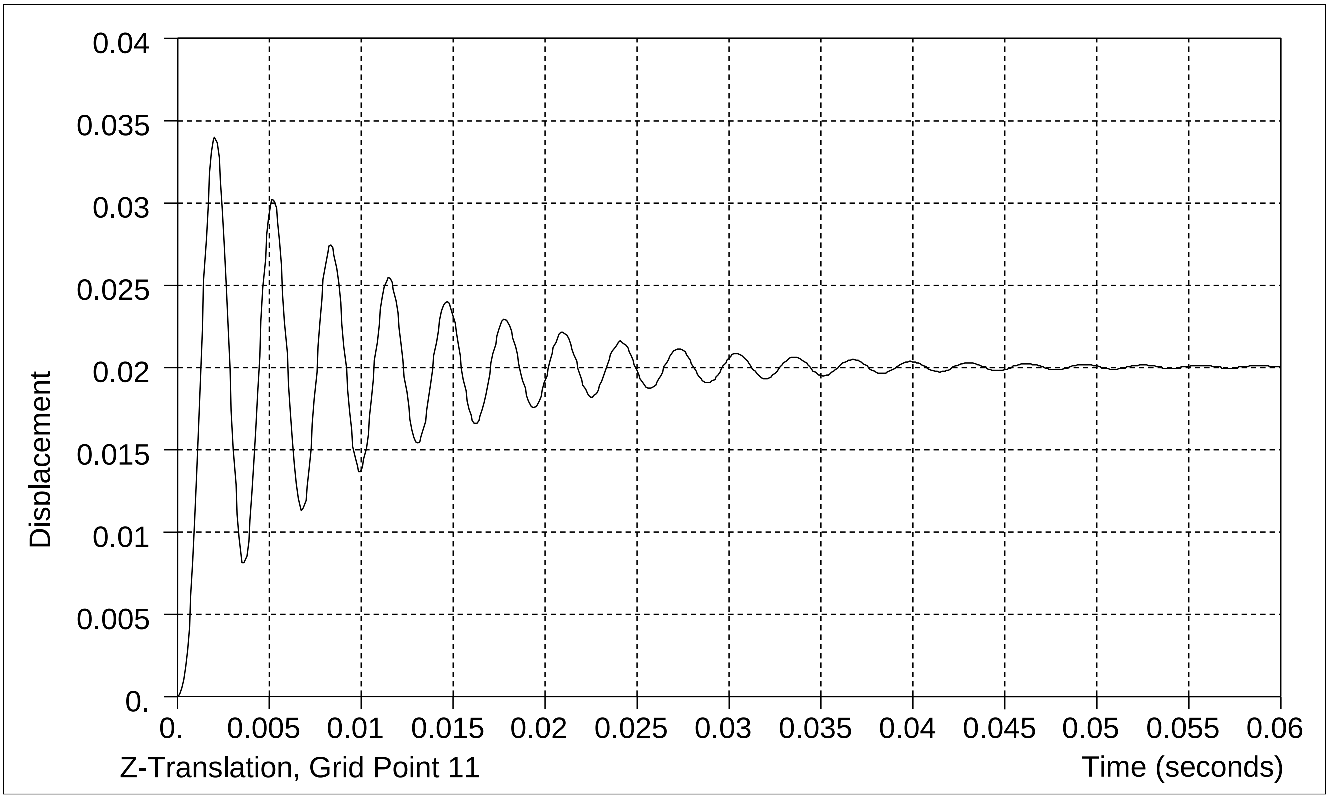
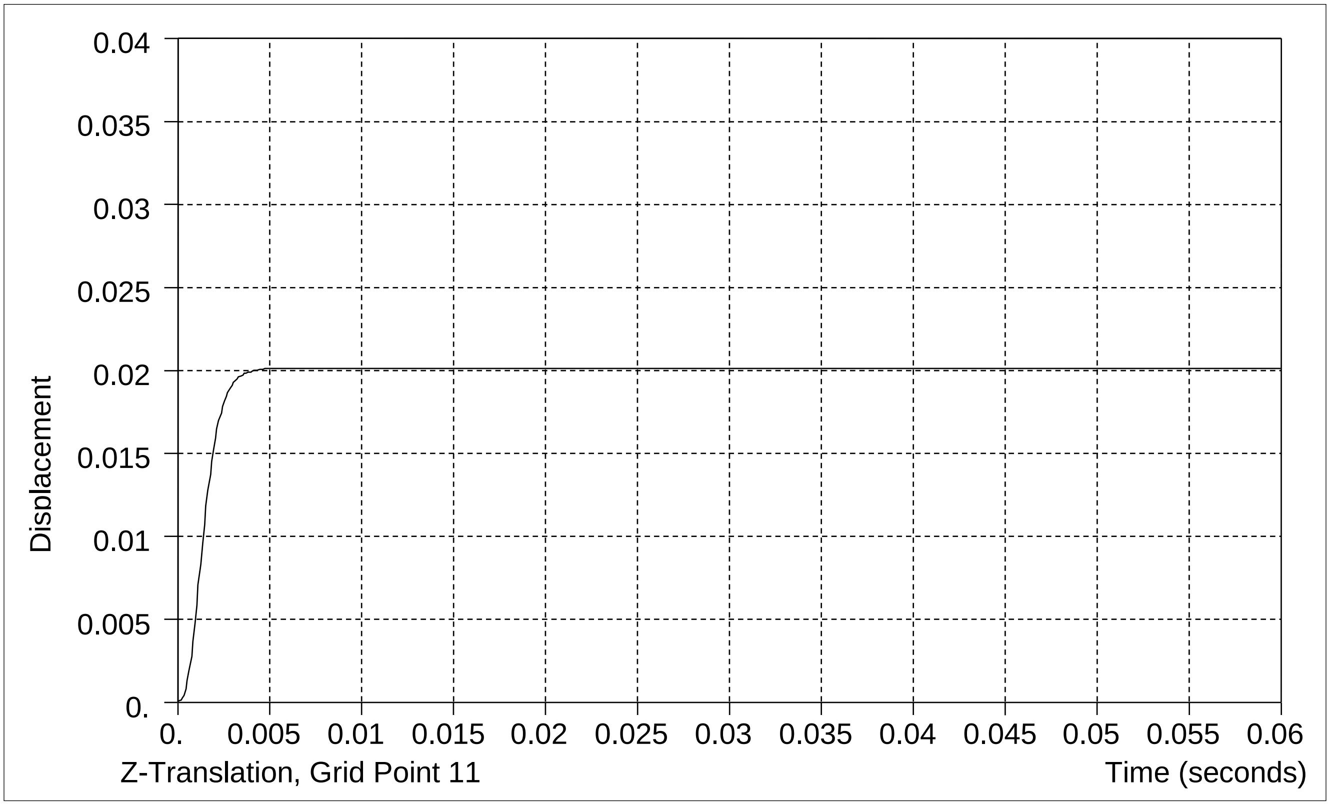


Figure 5-5c. Tip Displacement of a 2-D Cantilever Beam with 100% Critical Damping.



The examples shown in Listing 5-1 use the `TLOAD1` Bulk Data entry which defines the time-dependent loading using a table (`TABLEDi`). We can also define dynamic loads using an analytical function of the form:

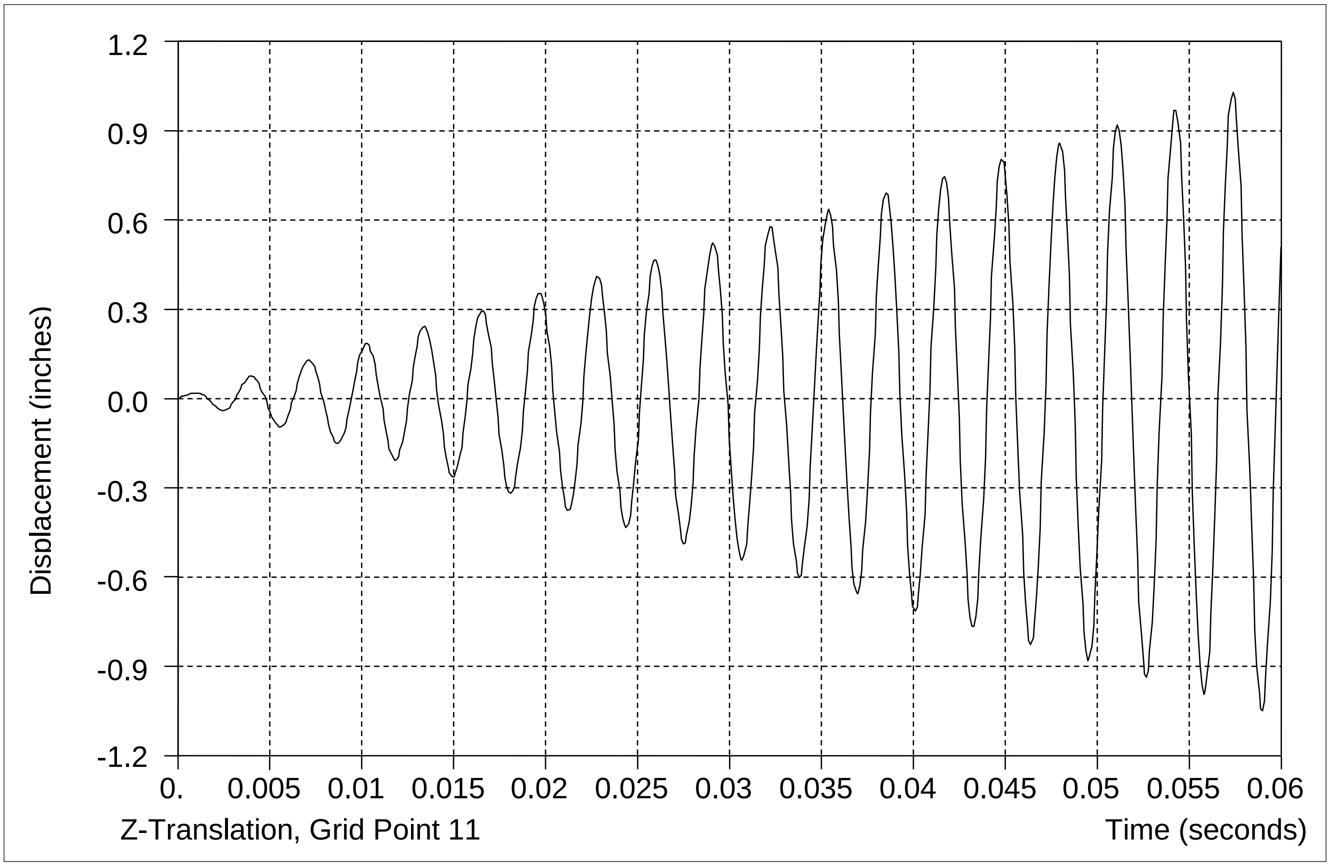
$$P(t) = \mathrm{A}t^{\mathrm{B}} e^{\mathrm{C}t} \cos(2\pi \mathrm{F}t + \mathrm{P})$$

using the `TLOAD2` Bulk Data entry. Again, we will use the cantilever beam shown in Figure 5-3. For this example, the beam will be loaded at its free end with a harmonic forcing function at its resonant frequency. Listing 5-2 contains the Model Input File and Figure 5-6 the response of the beam free end.

Listing 5-2. Model Input File for the 2-D Cantilever Beam Loaded at Resonant Frequency.

```
$
$ MODAL TRANSIENT RESPONSE SOLUTION - FORCED RESPONSE AT BEAM RESONANT
$ FREQUENCY.
$
SOL MODAL TRANSIENT RESPONSE
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = FORCED DYNAMIC RESPONSE AT BEAM RESONANT FREQUENCY
$
DISPLACEMENT = ALL
$
TSTEP = 25
LOADSET = 10
METHOD = 1
SPC = 1
SUBCASE 1
 LABEL = 10 LB EDGE LOAD IN Z-DIRECTION
 DLOAD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  25
$
$ DEFINE 0.25 SECONDS OF RESPONSE.
$
TSTEP,  25,  600,  0.0001,  1
$
$ DEFINE LOADING.
$
DLOAD,  1,  1.,  1.,  11
$
$ DEFINE TIME-DEPENDENT HARMONIC LOADING AT BEAM RESONANT FREQUENCY.
$
TLOAD2,  11,  100,  ,  ,  0.,  0.1,  317.974,  0.
$
$ 10 LB POINT LOAD IN Z-DIRECTION AT FREE END.
$
FORCE,  1,  11,  0,  10.,  0.,  0.,  1.
LSEQ,  10,  100,  1
$
$ INSERT BASIC MODEL (SEE LISTING 5-1a).
$
ENDDATA
```

Figure 5-6. Tip Displacement of a 2-D Cantilever Beam Loaded at Resonant Frequency.

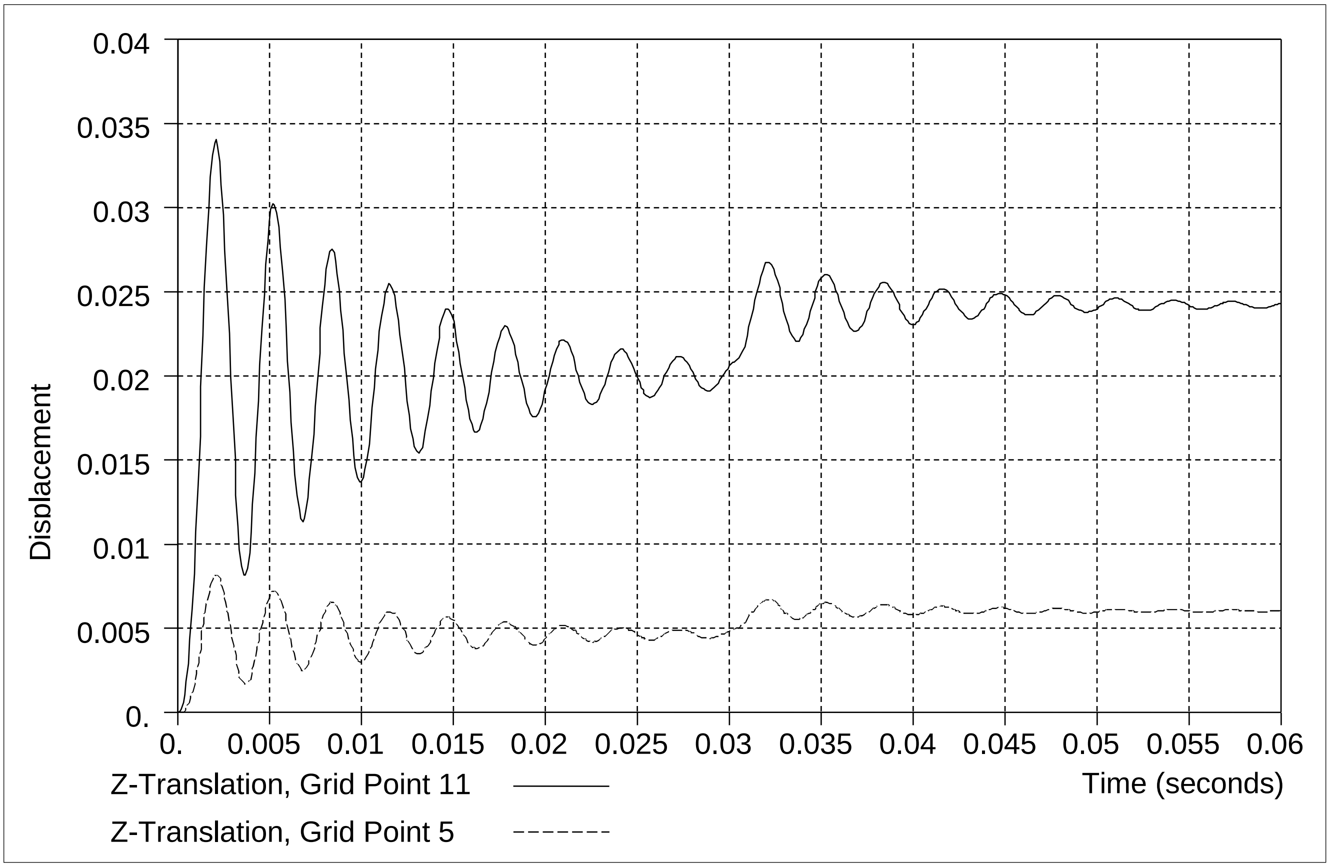


Dynamic loads on a structure can be complex, acting at different points and at different times. A time delay can be specified when it is desired to define loads acting at different points in time. For example, Listing 5-3 defines two separate loads acting at two different locations and instances in time. Again, we will use the cantilever beam shown in Figure 5-3 except for the addition of another point load at the beam mid-span. Figure 5-7 shows the response of the beam free end and mid-span.

Listing 5-3. Model Input File for the 2-D Cantilever Beam Problem with Multiple Point Loads and a Delay.

```
$
$ MODAL TRANSIENT RESPONSE SOLUTION - MULTPLE POINT LOADS WITH A DELAY.
$
SOL MODAL TRANSIENT RESPONSE
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = FORCED DYNAMIC RESPONSE - MULTIPLE POINT LOADS WITH A DELAY
$
DISPLACEMENT = ALL
$
SDAMPING = 20
TSTEP = 25
LOADSET = 10
METHOD = 1
SPC = 1
SUBCASE 1
 LABEL = 10 LB EDGE LOAD IN Z-DIRECTION
 DLOAD = 1$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  25
$
$ DEFINE 0.06 SECONDS OF RESPONSE.
$
TSTEP,  25,  600,  0.0001,  1
$
$ DEFINE LOADING.
$
DLOAD,  1,  1.,  1.,  11,  1.,  12
$
$ DEFINE TIME-DEPENDENT LOADING.
$
TLOAD1,  11,  100,  ,  ,  10
TLOAD1,  12,  200,  30,  ,  10
TABLED1,  10,
, -1.,  0.,  0.,  0.,  0.001,  1.,  100.,  1.,
,  ENDT
$
$ DEFINE A 0.03 SECOND DELAY FOR THE LOAD AT THE BEAM MID-SPAN.
$
DELAY,  30,  5,  3,  0.03
$
$ 10 LB POINT LOAD IN Z-DIRECTION AT FREE END.
$
FORCE,  1,  11,  0,  10.,  0.,  0.,  1.
FORCE,  2,  5,  0,  10.,  0.,  0.,  1.
LSEQ,  10,  100,  1
LSEQ,  10,  200,  2
$
$ 5% CRITICAL DAMPING.
$
TABDMP1,  20,  CRIT,
,  1.,  0.05,  10000.,  0.05,  ENDT
$
$ INSERT BASIC MODEL (SEE LISTING 5-1a).
$
ENDDATA
```

Figure 5-7. Tip and Midpoint Displacements of a 2-D Cantilever Beam with Multiple Point Loads and a Delay.



5.4 Enforced Motion

Enforced motion specifies the displacement, velocity, and/or acceleration at a set of grid points in transient response analysis. Enforced motion is used when base excitation is desired and can be combined with externally applied dynamic loading. A good example would be a building subjected to base motion due to an earthquake. In this case, instead of applied loads, the base is subjected to an enforced displacement or acceleration time history.

Autodesk Inventor Nastran uses the large mass method to convert applied forces to enforced motion. The idea is that if a very large mass m_0.is connected to a degree of freedom and a dynamic load p is applied to that same degree of freedom, then the acceleration of the degree of freedom is closely approximated by:

$$\ddot{u} = \frac{1}{M_0} p$$

Which can be re-written in terms of the load that produces the desired acceleration as:

$$p = m_0 \ddot{u}$$

The accuracy of this approximation improves as m_0 becomes larger in comparison to the mass of the structure. A good rule-of-thumb value for m_0 is approximately 10^6 times the mass of the entire structure for an enforced translational degree of freedom and 10^6 times the mass moment of inertia for a rotational degree of freedom.

Provisions are made on the `TLOADi` Bulk Data entries for specifying whether an enforced displacement, velocity, or acceleration is supplied (TYPE = 1, 2, 3). Autodesk Inventor Nastran will automatically differentiate a specified velocity once or a specified displacement twice to obtain acceleration. The `DYNLMDIRECTDIF` parameter controls whether this differentiation is carried out directly or empirically. See the *Nastran Solver Reference Guide*, Section 5, *Parameters,* for more information.

The following are the basic steps involved in the large mass method:

1. Remove any constraints from the enforced degrees of freedom.
2. Apply large masses m_0 with the `CMASSi` or `CONMi` Bulk Data entries to the degrees of freedom where the motion is enforced. The magnitude for m_0 should be approximately 10^6 times the mass of the entire structure for an enforced translational degree of freedom and 10^6 times the mass moment of inertia for a rotational degree of freedom.
3. Indicate in field 5 of the `TLOADi` Bulk Data entry whether the enforced motion is a displacement, velocity, or acceleration.

The following examples demonstrate the large mass method for enforced motion. Again, we will use the cantilever beam shown in Figure 5-3 except for the removal of the z-direction constraint at the fixed end. Listing 5-4 contains the Model Input File and Figure 5-8 the response of the beam free end.

Listing 5-4a. Model Input File for the 2-D Cantilever Beam Problem with Enforced Motion.

```
$
$ MODAL TRANSIENT RESPONSE SOLUTION - ENFORCED MOTION.
$
SOL MODAL TRANSIENT RESPONSE
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = FORCED DYNAMIC RESPONSE - ENFORCED DISPLACEMENT
$
DISPLACEMENT = ALL
VELOCITY = ALL
ACCELERATION = ALL
FORCE = ALL

SDAMPING = 20
TSTEP = 25
METHOD = 1
SPC = 1
SUBCASE 1
 LABEL = ENFORCED DISPLACEMENT AT CONSTRAINED END
 DLOAD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  25
$
$ DEFINE 0.06 SECONDS OF RESPONSE.
$
TSTEP,  25,  600,  0.0001,  1
$
$ DEFINE LOADING.
$
DLOAD,  1,  2.E+5,  0.002588,  11
$
$ DEFINE TIME-DEPENDENT LOADING.
$
TLOAD1,  11,  100,  ,  1,  10
TABLED1,  10,
,  0.,  0.,  0.001,  1.,  100.,  1.,  ENDT
DAREA,  100,  1,  3,  1.
$
$ LARGE MASS OF (1xE6)*BAR MASS = (1xE6)*2.0
$
CONM2,  20,  1,  ,  2.E+5
$
$ 5% CRITICAL DAMPING.
$
TABDMP1,  20,  CRIT,
,  1.,  0.05,  10000.,  0.05,  ENDT
$
$ FIXED AT ONE END EXCEPT IN Z-DIRECTION, MOVEMENT CONSTRAINED TO
$ X-Z PLANE ONLY.
$
SPC1,  1,  12456,  1
SPC1,  1,    246,  1,  THRU,  11
$
$ INSERT BASIC MODEL (SEE LISTING 5-1a).

ENDDATA
```

Listing 5-4b. Model Input File for the 2-D Cantilever Beam Problem with Enforced Motion at Resonant Frequency.

```
$
$ MODAL TRANSIENT RESPONSE SOLUTION - ENFORCED MOTION AT RESONANT FREQUENCY.
$
SOL MODAL TRANSIENT RESPONSE
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = FORCED DYNAMIC RESPONSE - ENFORCED DISPLACEMENT AT RESONANT FREQ
$
DISPLACEMENT = ALL
VELOCITY = ALL
ACCELERATION = ALL
FORCE = ALL
$
SDAMPING = 20
TSTEP = 25
METHOD = 1
SPC = 1
SUBCASE 1
 LABEL = ENFORCED DISPLACEMENT AT CONSTRAINED END
 DLOAD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  25
$
$ DEFINE 0.06 SECONDS OF RESPONSE.
$
TSTEP,  25,  600,  0.0001,  1
$
$ DEFINE LOADING.
$
DLOAD,  1,  2.E+5,  0.002588,  11
$
$ DEFINE TIME-DEPENDENT HARMONIC LOADING AT BEAM RESONANT FREQUENCY.
$
TLOAD2,  11,  100,  ,  1,  0.,  0.1,  317.974,  0.
DAREA,  100,  1,  3,  1.
$
$ LARGE MASS OF (1xE6)*BAR MASS = (1xE6)*2.0
$
CONM2,  20,  1,  ,  2.E+5
$
$ 5% CRITICAL DAMPING.
$
TABDMP1,  20,  CRIT,
,  1.,  0.05,  10000.,  0.05,  ENDT
$
$ FIXED AT ONE END EXCEPT IN Z-DIRECTION, MOVEMENT CONSTRAINED TO
$ X-Z PLANE ONLY.
$
SPC1,  1,  12456,  1
SPC1,  1,    246,  1,  THRU,  11
$
$ INSERT BASIC MODEL (SEE LISTING 5-1a).
$
ENDDATA
```

Figure 5-8a. Tip Displacement of a 2-D Cantilever Beam Subjected to a 1.0 Inch Enforced Displacement at the Constrained End.

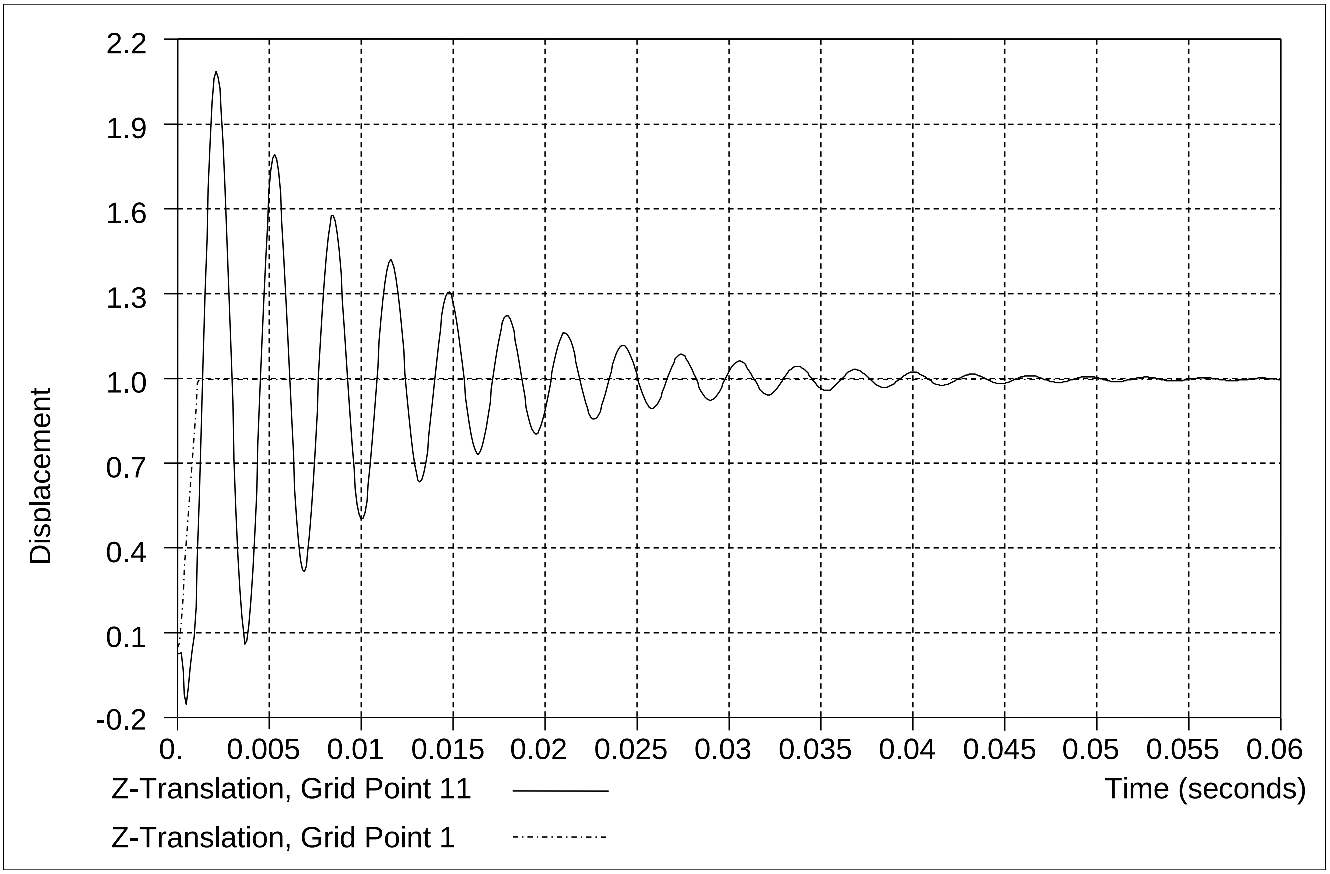
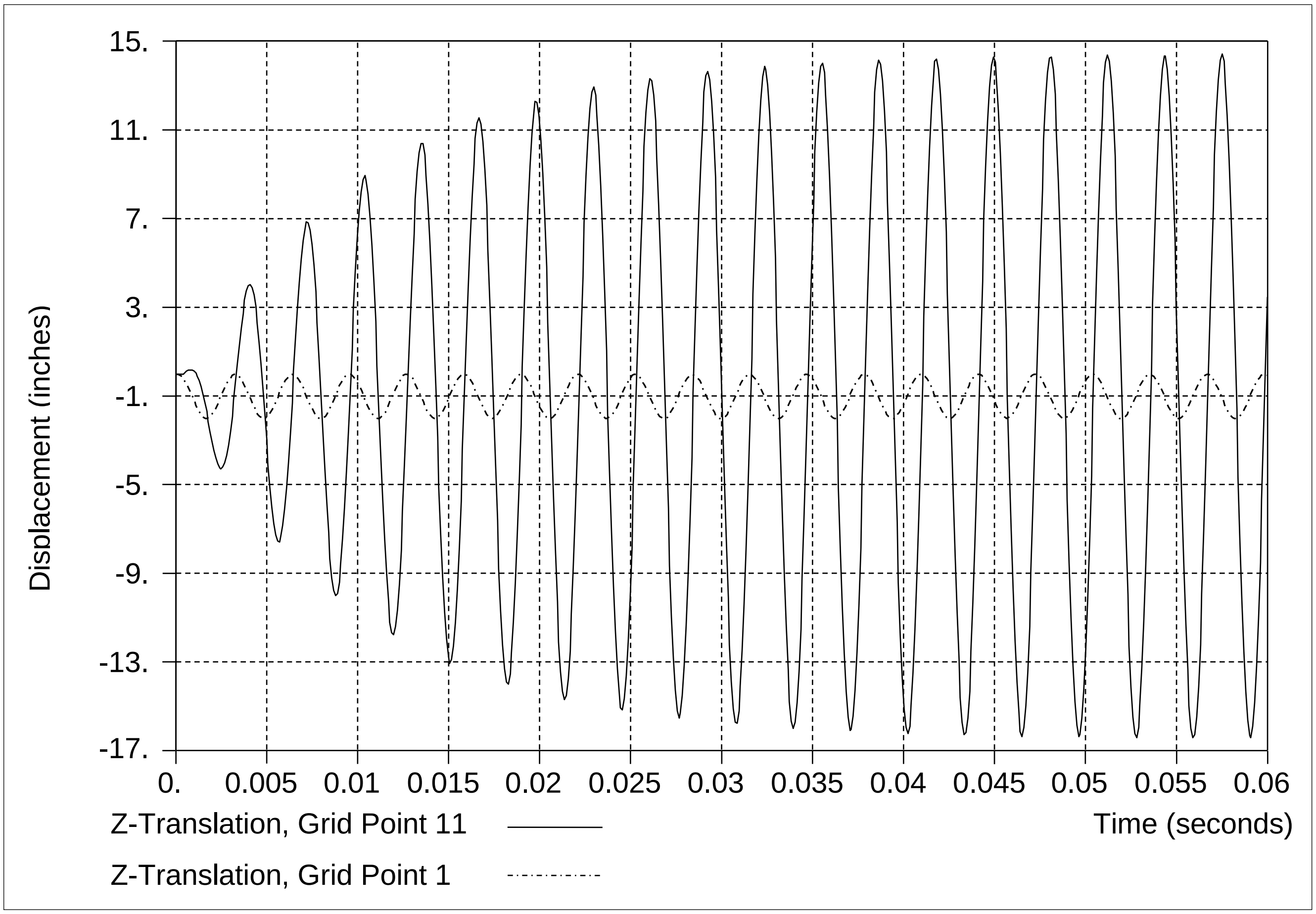


Figure 5-8b. Tip Displacement of a 2-D Cantilever Beam Subjected to an Enforced Harmonic Displacement at Resonant Frequency at the Constrained End.



6. SHOCK AND RESPONSE SPECTRUM ANALYSIS

6.1 Introduction

Shock spectrum analysis, also known as response spectrum analysis, solves for the maximum (peak) expected response (displacements, velocities, accelerations, forces, stresses, and strains) of structures subjected to complicated time dependent loads or accelerations. These loads or accelerations typically excite the base of a structure such as earthquake ground motion on a building. Other examples include an explosive shock on a small component in a ship or an impulse load due to stage separation in a spacecraft. Note that the only difference between shock and response spectra is the displacement output reference frame.

While this method is widely accepted, it is still an approximation and is often used only as a design tool. The primary advantage of this method over a conventional transient response analysis is efficiency as the only significant calculation is obtaining a sufficient number of modes to represent the entire frequency range of input excitation and resulting response. The primary disadvantage of this method is the accuracy may be questionable.

The solution is accomplished in through two separate analyses. The first analysis converts applied loads or base excitations through a modal transient response solution into a spectrum table containing the peak response magnitudes for a set of single degree of freedom oscillators (spectrum generation). The second analysis consists of a normal modes solution of the structure, data recovery, and the response calculation that combines the modal properties of the analysis model with the spectrum data of the applied loads (spectrum application).

6.2 Generating Response Spectra Data

The generation of response spectra involves converting applied loads or base excitations through a modal transient response solution into a spectrum table containing the peak response magnitudes for a set of single degree of freedom oscillators. Each oscillator consists of a scalar spring/mass/damper with a different natural frequency and damping ratio (Figure 6-1).

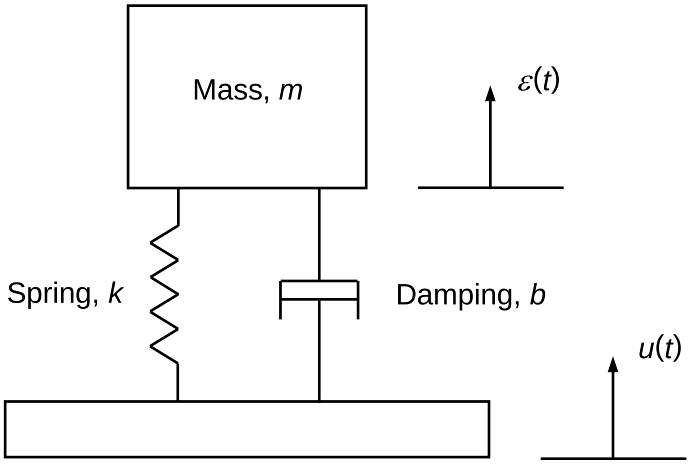


Figure 6-1. Oscillator Definition.

The equation of motion for each oscillator becomes

$$m_i \ddot{\varepsilon} + b_i \dot{\varepsilon} + k_i \varepsilon = u_b$$

where,

$m_i = 1$

$k_i = \omega_i^2$

$b_i = 2\zeta_i \sqrt{k_i}$

u_b = enforced acceleration

ε = peak modal response

The series of single degree of freedom (SDOF) oscillators all act at the same physical location at the connection to the base structure. These oscillators each have a different resonant frequency with the same specified damping ratio. The modal transient response solution is used to excite the oscillators. The peak oscillator response to the base structure excitation is stored versus oscillator frequency and is the generated response spectra table for the specified damping ratio.

6.3 How to Setup a Model Input File for Response Spectrum Analysis –Spectrum Generation

In Autodesk Inventor Nastran, the spectrum generation part of response spectrum analysis is performed as a post processing step in a transient response solution (`SOL MODAL TRANSIENT RESPONSE`, `SOL DIRECT TRANSIENT RESPONSE`, `SOL LINEAR PRESTRESS TRANSIENT RESPONSE`, and `SOL NONLINEAR PRESTRESS TRANSIENT RESPONSE`). Therefore, the first step in setting up the spectrum generation part of a response spectrum analysis is the same as for a transient response analysis. The Response Spectrum Generation module is activated by setting `PARAM, RSPECTRA` to `ON`.

6.3.1 Response Spectrum Definition

Response spectra are generated in the transient response solution sequence. Transient response input is required to apply the transient excitation to the base structure. Additional input required to generate response spectra are described below:

`XYPRINT SPECTRAL`	Case Control command to compute response spectra
`DTI, SPSEL`	Bulk Data entry to select oscillator frequencies, oscillator damping values, and grid points at which spectra will be computed
`FREQi`	Bulk Data entry to specify oscillator frequencies and damping values
`PARAM, RESPECTRA, ON`	Model parameter to invoke the Response Spectrum Generation module

See *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for more information on each entry. The `FREQi` entries are used to specify both oscillator frequencies (i.e., the frequencies for which the spectra will be computed) and oscillator damping. Note that damping for the base structure is specified using the `TABDMP1` Bulk Data entry the same as for a standard transient response solution.

The `XYPRINT` command has the following form:

Command	Response	Type	Spectrum Id	Grid Point Id	Component Symbol	Motion Type

`XYPRINT`	`ACCELERATION`
	`VELOCITY`
	`DISPLACEMENT`
		`SPECTRAL`	Integer > 0	Integer > 0	`T1, T2, T3, R1, R2, R3`	`IP` (relative)
						`RM` (absolute)

For example, to generate the relative acceleration spectra for grid point 11, component T3

```
XYPRINT ACCELERATION SPECTRAL 1 /11(T3IP)
```

Alternatively, to generate the absolute displacement spectra for grid point 6, component T2

```
XYPRINT DISPLACEMENT SPECTRAL 1 /6(T2RM)
```

Note that `XYPLOT` and `XYPUNCH` may be interchanged with `XYPRINT`, but will have the same effect. Also, the response options `ACCELERATION`, `VELOCITY`, and `DISPLACEMENT`, may be shortened to `ACCE`, `VELO`, and `DISP` respectively.

6.3.2 Response Spectrum Output

The generated spectra are written to the FEMAP Results Neutral File as data functions and to the Bulk Data Output File (*filename*.BDF) as `TABLED1` Bulk Data entries. This data may then be referenced as the spectra input to the subsequent normal modes analysis discussed later in this section. Bulk Data Output File operation is controlled by the `TRSLMODLDATA` Model Initialization directive (see *Nastran Solver Reference Guide*, Section 2, *Initialization,* for directive format). If `TRSLMODLDATA` is set to `ON`, this file will also contain the analysis model Case Control commands and Bulk Data entries.

6.4 Interpreting Results

As an example we will use the cantilever beam shown in Figure 6-2. The beam is subjected to an enforced 1 inch impulse displacement in the z-direction at the constrained end. It is desired to generate the response spectrum at the beam free end for 2% critical damping. Modal damping is used with 2% critical damping across all modes for the beam. Listing 6-1 contains the Model Input File and Figure 6-3 the response of the beam free end. The generated response spectrum is plotted in Figure 6-4. The input response spectrum `TABLED1` Bulk Data entries are shown in Listing 6-2.

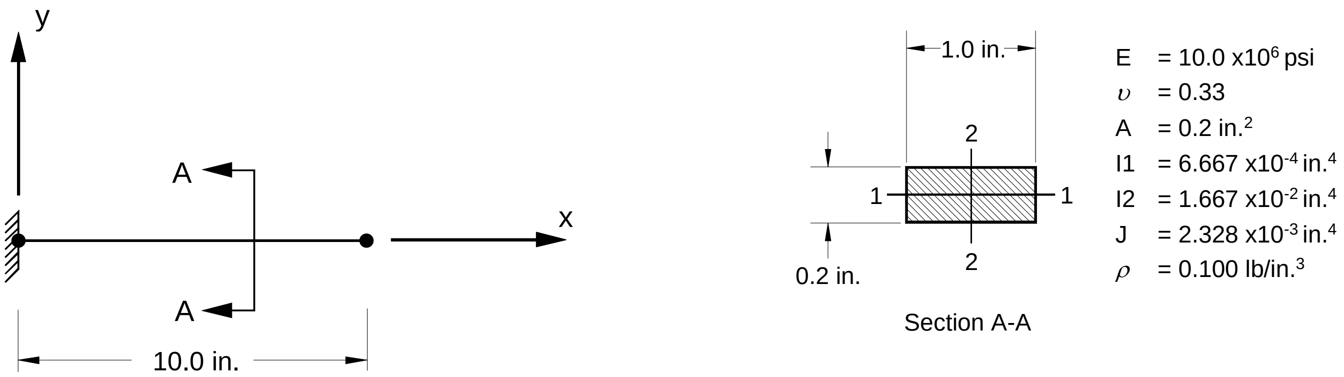


Figure 6-2. 2-D Cantilever Beam Example Problem.

Listing 6-1. Model Input File for the 2-D Cantilever Beam Problem with Enforced Motion and Response Spectrum Generation.

```
$
$ LINEAR TRANSIENT RESPONSE SOLUTION – RESPONSE SPECTRUM GENERATION.
$
SOL MODAL TRANSIENT RESPONSE
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = FORCED DYNAMIC RESPONSE - ENFORCED DISPLACEMENT
$
DISPLACEMENT = ALL
VELOCITY = ALL
ACCELERATION = ALL

SDAMPING = 20
TSTEP = 25
METHOD = 1
SPC = 1
SUBCASE 1
 LABEL = ENFORCED DISPLACEMENT AT CONSTRAINED END
 DLOAD = 1
$
$ XYPRINT COMMAND THAT GENERATES SPECTRA FOR SET ID 1, GRID POINT 11, COMPONENT 3,
$ ABSOLUTE ACCELERATION.
$
XYPRINT ACCELERATION SPECTRAL 1 / 11(T3RM)
$
BEGIN BULK
$
$ REQUEST RESPONSE SPECTRUM GENERATION.
$
PARAM, RSPECTRA, ON
$
$ DEFINE RESPONSE SPECTRUM GENERATION.
$
DTI,  SPSEL,  1,  10,  20,  11
$
$ GENERATE SPECTRA IN 5 Hz INCREMENTS.
$
FREQ1,  20,  1.,  5.,  100
$
$ 2% CRITICAL DAMPING VALUE FOR SPECTRUM GENERATION.
$
FREQ,  10,  0.02
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  25
```

Listing 6-1. Model Input File for the 2-D Cantilever Beam Problem with Enforced Motion and Response Spectrum Generation. (Continued)

```
$
$ DEFINE 0.06 SECONDS OF RESPONSE.
$
TSTEP,  25,  600,  0.0001,  1
$
$ DEFINE LOADING.
$
DLOAD,  1,  2.E+5,  0.002588,  11
$
$ DEFINE TIME-DEPENDENT LOADING.
$
TLOAD1,  11,  100,  ,  1,  10
TABLED1,  10,
,  0.,  0.,  0.001,  1.,  100.,  1.,  ENDT
DAREA,  100,  1,  3,  1.
$
$ LARGE MASS OF (1xE6)*BAR MASS = (1xE6)*2.0
$
CONM2,  20,  1,  ,  2.E+5
$
$ 2% CRITICAL DAMPING.
$
TABDMP1,  20,  CRIT,
,  1.,  0.02,  10000.,  0.02,  ENDT
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  1.,  0.
CBAR,  2,  10,   2,   3,  0.,  1.,  0.
CBAR,  3,  10,   3,   4,  0.,  1.,  0.
CBAR,  4,  10,   4,   5,  0.,  1.,  0.
CBAR,  5,  10,   5,   6,  0.,  1.,  0.
CBAR,  6,  10,   6,   7,  0.,  1.,  0.
CBAR,  7,  10,   7,   8,  0.,  1.,  0.
CBAR,  8,  10,   8,   9,  0.,  1.,  0.
CBAR,  9,  10,   9,  10,  0.,  1.,  0.
CBAR, 10,  10,  10,  11,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3,
,  -0.1,  0.5,  0.1,  0.5,  -0.1,  -0.5,  0.1,  -0.5
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ FIXED AT ONE END EXCEPT IN Z-DIRECTION, MOVEMENT CONSTRAINED TO
$ X-Z PLANE ONLY.
$
SPC1,  1,  12456,  1
SPC1,  1,     246,  1,  THRU,  11
ENDDATA
```

Figure 6-3. Tip Displacement of a 2-D Cantilever Beam Subjected to a 1.0 Inch Enforced Displacement at the Constrained End.

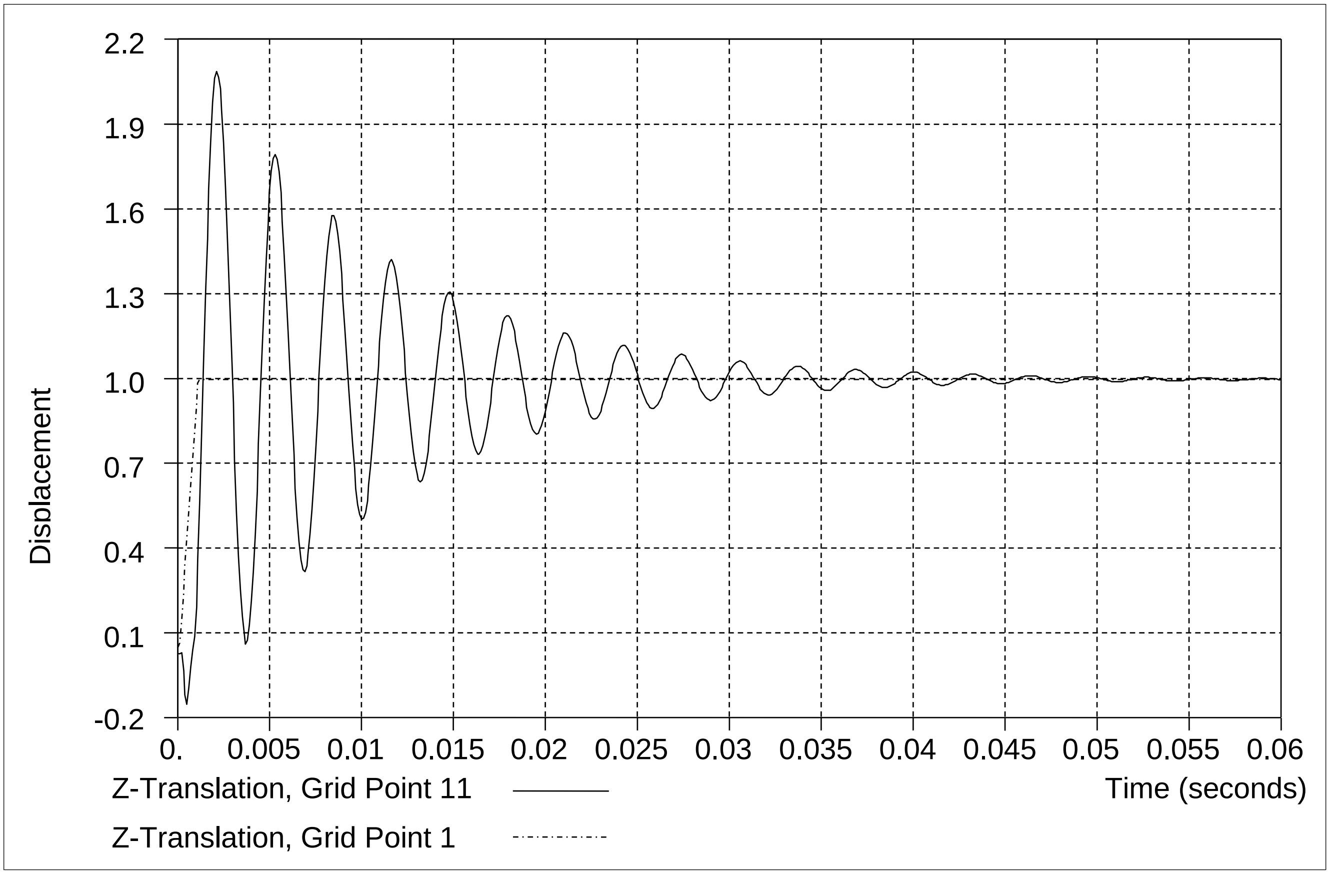
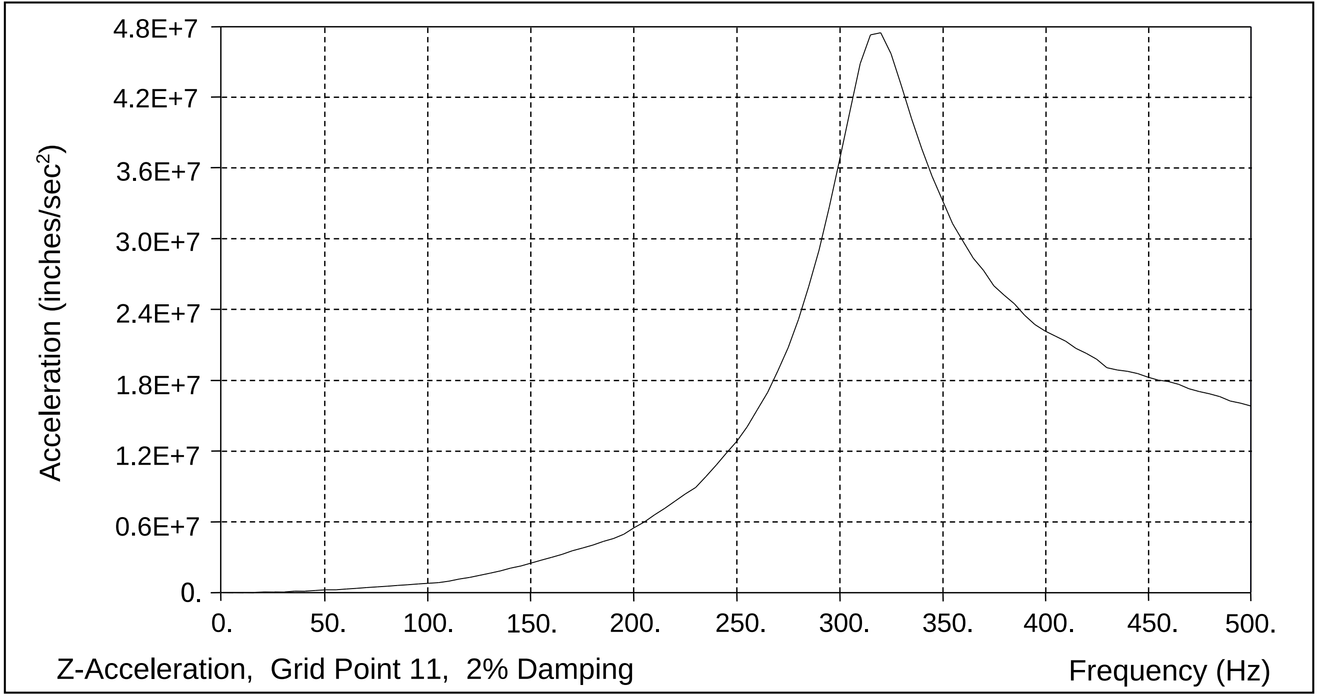


Figure 6-4. Absolute Acceleration Response Spectrum of a 2-D Cantilever Beam Tip.



Listing 6-2. Generated Response Spectrum Table.

```
$
$ OUTPUT PRODUCED BY ADS NASTARAN VERSION 10.3.0.716   02:02  01/16/15
$
BEGIN BULK
$
$-------2-------3-------4-------5-------6-------7-------8-------9-------0-------
$
$ ACCE     11 T3 0.020
$
TABLED1       21                                                        +C    1A
+C    1A 1.00000 87.4087 6.00000 3143.27 11.0000 10548.8 16.0000 22274.8+C    2A
+C    2A 21.0000 38280.8 26.0000 58515.2 31.0000 82915.6 36.0000 111408.+C    3A
+C    3A 41.0000 143908. 46.0000 180321. 51.0000 220539. 56.0000 264445.+C    4A
+C    4A 61.0000 311910. 66.0000 362795. 71.0000 416950. 76.0000 477095.+C    5A
+C    5A 81.0000 546781. 86.0000 610951. 91.0000 668883. 96.0000 732492.+C    6A
+C    6A 101.000 804690. 106.000 879314. 111.000 986341. 116.0001.1377+6+C    7A
+C    7A 121.0001.3016+6 126.0001.4776+6 131.0001.6653+6 136.0001.8639+6+C    8A
+C    8A 141.0002.0723+6 146.0002.2896+6 151.0002.5142+6 156.0002.7565+6+C    9A
+C    9A 161.0003.0061+6 166.0003.2613+6 171.0003.5201+6 176.0003.7807+6+C   10A
+C   10A 181.0004.0422+6 186.0004.3252+6 191.0004.6078+6 196.0004.9413+6+C   11A
+C   11A 201.0005.4817+6 206.0006.0250+6 211.0006.6111+6 216.0007.1866+6+C   12A
+C   12A 221.0007.7654+6 226.0008.3763+6 231.0008.9595+6 236.0009.8555+6+C   13A
+C   13A 241.0001.0820+7 246.0001.1868+7 251.0001.2825+7 256.0001.4070+7+C   14A
+C   14A 261.0001.5549+7 266.0001.7011+7 271.0001.8872+7 276.0002.0809+7+C   15A
+C   15A 281.0002.3221+7 286.0002.5996+7 291.0002.9089+7 296.0003.2719+7+C   16A
+C   16A 301.0003.6770+7 306.0004.0860+7 311.0004.4888+7 316.0004.7332+7+C   17A
+C   17A 321.0004.7545+7 326.0004.5762+7 331.0004.3081+7 336.0004.0252+7+C   18A
+C   18A 341.0003.7679+7 346.0003.5367+7 351.0003.3320+7 356.0003.1325+7+C   19A
+C   19A 361.0002.9882+7 366.0002.8407+7 371.0002.7341+7 376.0002.6072+7+C   20A
+C   20A 381.0002.5247+7 386.0002.4534+7 391.0002.3567+7 396.0002.2740+7+C   21A
+C   21A 401.0002.2201+7 406.0002.1760+7 411.0002.1352+7 416.0002.0761+7+C   22A
+C   22A 421.0002.0335+7 426.0001.9805+7 431.0001.9110+7 436.0001.8914+7+C   23A
+C   23A 441.0001.8789+7 446.0001.8584+7 451.0001.8299+7 456.0001.8073+7+C   24A
+C   24A 461.0001.7903+7 466.0001.7659+7 471.0001.7342+7 476.0001.7095+7+C   25A
+C   25A 481.0001.6900+7 486.0001.6637+7 491.0001.6306+7 496.0001.6067+7+C   26A
+C   26A 501.0001.5866+7ENDT
ENDDATA
```

6.5 Application of Response Spectra Data

The application of the generated response spectra consists of a normal modes solution of the structure, data recovery, and the response calculation that combines the modal properties of the analysis model with the spectrum data of the applied loads.

The general approximation for the response quantity, u_k, is

$$u_k(t) = \sum_i \phi_{ik} \xi_i(t)$$

where ϕ and ξ are the modal outputs and generalized displacements with the actual modal equations given by

$$\ddot{\xi}_i + 2\zeta_i \omega_i \dot{\xi}_i(t) + \omega_i^2 \xi_i(t) = \frac{1}{m_i}[\phi]^T P(t)$$

where,

$\zeta_i = b_i / (2 m_i \omega_i) \equiv$ modal damping ratio

$\omega_i^2 = k_i / m_i \equiv$ modal frequency

$P =$ load vector

For loading due to base accelerations we have

$$P(t) = -[M_{aa}][D_{ar}]\{\ddot{u}_r(t)\}$$

where the columns of $[D_{ar}]$ are vectors of rigid body motion and the accelerations correspond to the base motions.

Then, the actual transient response at a physical point is

$$u_k(t) = \sum_i \sum_r \phi_{ik} \psi_{ir} \, x_r(\omega_i, \zeta_i, t)$$

In the above equation, the peak magnitudes of u_k are usually dominated by the peak values of x(t) occuring at the natural frequencies. In response spectrum analysis the peak values of u_k are approximated by combining functions of the peak values using

$$\bar{u}_k \cong \sum_i \sum_r |\phi_{ik}| |\psi_{ir} \bar{x}_{ri}(\omega_i, \zeta_i)|$$

The last two equations define the ABS (Absolute Value) option. This method is conservative in that it assumes all of the modal peak values for every point on the structure occur at the same time and in the same phase.

Another approach is to assume that the modal magnitudes and phases will combine in a probabilistic fashion. If the input loads occur randomly, the probable (RMS) peak values are

$$\bar{u}_k \cong \sqrt{\sum_i (\phi_{ik} \bar{\xi}_i)^2}$$

where the average peak modal magnitude, ξ_i is

$$\xi_i \cong \sqrt{\sum_r (\psi_{ir} \bar{x}_r (\omega_i, \zeta_i))^2}$$

This approach is known as the SRSS (square root of sum-squared) method. The method assume that the modal responses are uncorrelated and the peak value for each mode will occur at a different time. These results can be unconservative and represent a lower bound on the dynamic peak values.

The NRL (Naval Research Laboratories) method was developed as a compromise between the two other methods. The peak response is determined using the equation

$$\bar{u}_k \cong \left|\phi_{jk}\bar{\xi}_j\right| + \sqrt{\sum_{i \neq j} (\phi_{ik}\bar{\xi}_i)^2}$$

where the j-th mode is the mode that has the largest magnitude of the product $\phi_{jk}\bar{\xi}_j$.

Modes that are close in frequency may have their peak response occur at approximately the same time and phase. Because of this, the SRSS and NRL methods contain a provision to sum modal response using the ABS method for modes that have closely spaced natural frequencies. Close natural frequencies are defined using the criteria

$$f_{i+1} < \text{CLOSE} * f_i$$

The value for CLOSE is specified using `PARAM`, `CLOSE` with a default value if 1.0. See the *Nastran Solver Reference Guide*, Section 5, *Parameters,* for more information on `CLOSE`.

6.6 How to Setup a Model Input File for Response Spectrum Analysis –Modal Combination

In Autodesk Inventor Nastran, the spectrum application part of response spectrum analysis is performed as a post processing step in a normal modes solution (`SOL MODAL`, `SOL LINEAR PRESTRESS MODAL`, and `SOL NONLINEAR PRESTRESS MODAL`). The application of response spectra to a normal modes analysis is activated by setting `PARAM, SCRSPEC` to `ON`.

6.6.1 Response Spectrum Application

Response spectra are applied to a modal solution sequence. The additional input required is described below:

`SDAMPING`	Selects the `TABDMP1` Bulk Data entry
`DLOAD`	Selects the `DLOAD` Bulk Data entry
`DTI, SPECSEL`	Specifies the type of input spectrum and its corresponding damping ratio A = absolute acceleration spectrum V = relative velocity spectrum D = relative displacement spectrum
`TABLED1`	Specifies input response spectra
`SUPORT`	Specifies input spectrum grid points and component directions
`PARAM, SCRSPEC, ON`	Model parameter to invoke the Response Spectrum Application module
`PARAM, OPTION,` *method*	Model parameter to specify the modal combination method ABS = Absolute Value SRSS = Square Root of the Sum of the Squares NRL = Naval Research Laboratory CQC = Complete Quadratic Combination (default is ABS)
`PARAM, CLOSE,` *tol*	Model parameter to specify the closeness tolerance used in determining the modal combination method (default is 1.0)
`PARAM, MODALDATABASE,` *option*	Model parameter to control storage and retrieval of modal data DELETE = Modal database is deleted STORE = Modal database is stored FETCH = Modal database is retrieved (default is DELETE)

See *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for more information on each entry.

6.6.2 Response Spectrum Input

If the response spectra was generated from a transient response solution, it will be written to the FEMAP Results Neutral File as data functions and to the Bulk Data Output File (*filename*.BDF) as `TABLED1` Bulk Data entries. This data may then be referenced as the spectra input to the normal modes analysis.

6.7 Interpreting Results

As an example we will use the flat plate shown in Figure 6-5. The plate is completely constrained at one end except for the z-direction. The plate supports a nonstructural mass per unit area of 0.01 pound/inch^2. A large mass is connected at grid point 17 to the left edge of the plate with a rigid element. The response spectrum generated in the previous transient response solution is input at that same point. Listing 6-3 contains the Model Input File.

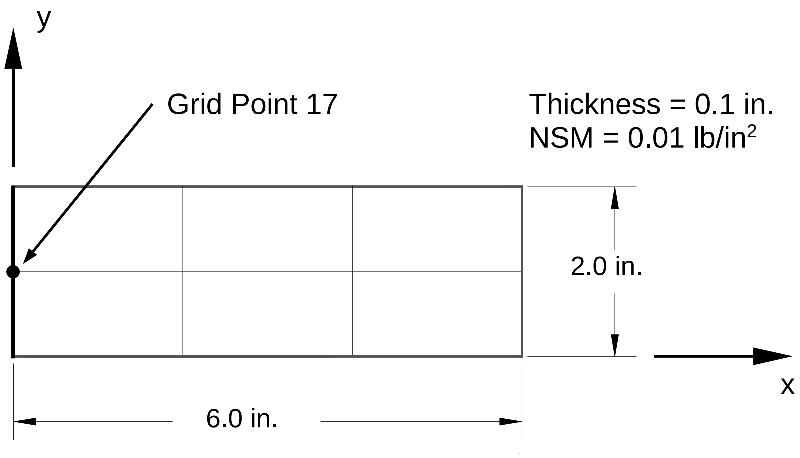


Figure 6-5. 3-D Cantilever Beam Example Problem.

Listing 6-3. Model Input File for the 3-D Cantilever Beam Problem with Response Spectrum Input.

```
$
$ MODAL SOLUTION - RESPONSE SPECTRUM APPLICATION.
$
SOL MODAL
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = MODAL COMBINATION
$
DISPLACEMENT = ALL
VELOCITY = ALL
ACCELERATION = ALL
STRESS = ALL
$
SPC = 1
SDAMPING = 20
METHOD = 1
$
SUBCASE 1
 LABEL = RESPONSE SPECTRUM APPLICATION
 DLOAD = 1
$
BEGIN BULK
$
$ REQUEST RESPONSE SPECTRUM APPLICATION.
$
PARAM, SCRSPEC, ON
$
$ DEFINE RESPONSE SPECTRUM INPUT.
$
DLOAD,  1,  1.,  1.,  10
$
$ RELATE SPECTRA LINES TO DAMPING VALUES.
$
DTI,  SPECSEL,  10,  ,  A,  21,  0.02
$
$ RESPONSE SPECTRUM GENERATED FROM TRANSIENT RESPONSE SOLUTION
$ (ABSOLUTE ACCELERATION WITH 2% CRITICAL DAMPING).
$
$ ACCE     11 T3 0.020
$
TABLED1       21                                                         +C    1A
+C    1A 1.00000 87.4087 6.00000 3143.27 11.0000 10548.8 16.0000 22274.8+C    2A
+C    2A 21.0000 38280.8 26.0000 58515.2 31.0000 82915.6 36.0000 111408.+C    3A
+C    3A 41.0000 143908. 46.0000 180321. 51.0000 220539. 56.0000 264445.+C    4A
+C    4A 61.0000 311910. 66.0000 362795. 71.0000 416950. 76.0000 477095.+C    5A
+C    5A 81.0000 546781. 86.0000 610951. 91.0000 668883. 96.0000 732492.+C    6A
+C    6A 101.000 804690. 106.000 879314. 111.000 986341. 116.0001.1377+6+C    7A
+C    7A 121.0001.3016+6 126.0001.4776+6 131.0001.6653+6 136.0001.8639+6+C    8A
+C    8A 141.0002.0723+6 146.0002.2896+6 151.0002.5142+6 156.0002.7565+6+C    9A
+C    9A 161.0003.0061+6 166.0003.2613+6 171.0003.5201+6 176.0003.7807+6+C   10A
+C   10A 181.0004.0422+6 186.0004.3252+6 191.0004.6078+6 196.0004.9413+6+C   11A
+C   11A 201.0005.4817+6 206.0006.0250+6 211.0006.6111+6 216.0007.1866+6+C   12A
+C   12A 221.0007.7654+6 226.0008.3763+6 231.0008.9595+6 236.0009.8555+6+C   13A
+C   13A 241.0001.0820+7 246.0001.1868+7 251.0001.2825+7 256.0001.4070+7+C   14A
+C   14A 261.0001.5549+7 266.0001.7011+7 271.0001.8872+7 276.0002.0809+7+C   15A
+C   15A 281.0002.3221+7 286.0002.5996+7 291.0002.9089+7 296.0003.2719+7+C   16A
+C   16A 301.0003.6770+7 306.0004.0860+7 311.0004.4888+7 316.0004.7332+7+C   17A
+C   17A 321.0004.7545+7 326.0004.5762+7 331.0004.3081+7 336.0004.0252+7+C   18A
+C   18A 341.0003.7679+7 346.0003.5367+7 351.0003.3320+7 356.0003.1325+7+C   19A
+C   19A 361.0002.9882+7 366.0002.8407+7 371.0002.7341+7 376.0002.6072+7+C   20A
+C   20A 381.0002.5247+7 386.0002.4534+7 391.0002.3567+7 396.0002.2740+7+C   21A
+C   21A 401.0002.2201+7 406.0002.1760+7 411.0002.1352+7 416.0002.0761+7+C   22A
+C   22A 421.0002.0335+7 426.0001.9805+7 431.0001.9110+7 436.0001.8914+7+C   23A
+C   23A 441.0001.8789+7 446.0001.8584+7 451.0001.8299+7 456.0001.8073+7+C   24A
+C   24A 461.0001.7903+7 466.0001.7659+7 471.0001.7342+7 476.0001.7095+7+C   25A
+C   25A 481.0001.6900+7 486.0001.6637+7 491.0001.6306+7 496.0001.6067+7+C   26A
+C   26A 501.0001.5866+7ENDT
```

Listing 6-3. Model Input File for the 3-D Cantilever Beam Problem with Response Spectrum Input. (Continued)

```
$
$ LOCATION OF RESPONSE SPECTRUM INPUT.
$
SUPORT,  17,  3
$
$ LARGE MASS OF (1xE6)*PLATE MASS = (1xE6)*(0.002588)*(0.0036) = 9.3 (MIN)
$
CONM2,  20,  17,  ,  1.+3
$
$ RIGID BODY ELEMENT CONNECTION OF MASS/INPUT POINT TO PLATE EDGE.
$
RBE2,  7,  17,  123456,  16,  18
$
$ 2% CRITICAL DAMPING.
$
TABDMP1,  20,  CRIT,
,  0.,  0.02,  1000.,  0.02,  ENDT
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ REMOVE RIGID BODY MODE FROM ANALYSIS.
$
PARAM, LFREQ, 1.
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  4
$
$ GEOMETRY DEFINITION (6" X 2" RECTANGULAR FLAT PLATE WITH A 3 X 2 MESH).
$
GRID,   4,  ,  2.,  0.,  0.
GRID,   5,  ,  2.,  1.,  0.
GRID,   6,  ,  2.,  2.,  0.
GRID,   7,  ,  6.,  0.,  0.
GRID,   8,  ,  6.,  1.,  0.
GRID,   9,  ,  6.,  2.,  0.
GRID,  10,  ,  4.,  0.,  0.
GRID,  11,  ,  4.,  1.,  0.
GRID,  12,  ,  4.,  2.,  0.
GRID,  16,  ,  0.,  0.,  0.
GRID,  17,  ,  0.,  1.,  0.
GRID,  18,  ,  0.,  2.,  0.
$
$ FLAT PLATE MODELED WITH SHELL ELEMENTS.
$
CQUADR,  1,  10,  16,   4,   5,  17
CQUADR,  2,  10,   4,  10,  11,   5
CQUADR,  3,  10,  10,   7,   8,  11
CQUADR,  4,  10,  17,   5,   6,  18
CQUADR,  5,  10,   5,  11,  12,   6
CQUADR,  6,  10,  11,   8,   9,  12
$
$ ELEMENT MATERIAL AND THICKNESS (0.03") WITH 0.01 LB/IN. NONSTRUCTURAL
$ MASS PER UNIT AREA.
$
PSHELL,  10,  100,  0.03,  100,  ,  100,  ,  0.01
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ FIXED AT ONE END EXCEPT IN Z-DIRECTION.
$
SPC1,  1,  12456,  17
ENDDATA
```

The response spectrum application is requested by setting `PARAM`, `SCRSPEC` to `ON`. The `DLOAD` Case Control command references the `DLOAD` Bulk Data entry. The `DLOAD` entry references the `DTI, SPECSEL` entry which specifies the input spectrum type, spectrum tables, and corresponding critical damping values. The `TABLED1` entry defines a specific line of the input spectrum for a specific damping ratio. The `TABLED1` entry shown was generated in the previous transient response solution for 2% critical damping. The `TABDMP1` entry defines the modal damping (in this case all modes have 2% critical damping).

Listing 6-4a gives the modal results summary information. The eigenvalue summary shows the computed natural frequencies. Note the absence of the rigid body mode due to the specification of `PARAM`, `LFREQ`, `1`. This parameter excludes all modes below 1 Hz. The modal participation factors, modal effective mass, and peak modal responses are also listed. Listing 6-4b and 6-4c gives the combined modal results using the ABS (default) method.

Listing 6-4a. Modal Results Output from the Application of Response Spectra to the 3-D Cantilever Beam.

```
 RESPONSE SPECTRUM APPLICATION                                       SUBCASE 1


                                            R E A L   E I G E N V A L U E S

    MODE         EIGENVALUE       RADIANS          CYCLES        GENERALIZED      GENERALIZED      ORTHOGONALITY       ERROR
   NUMBER                                                           MASS           STIFFNESS            LOSS          MEASURE
      1        6.950168E+03    8.336767E+01    1.326838E+01    1.000000E+00    6.950168E+03    3.278535E-10    1.874723E-09
      2        3.637892E+05    6.031494E+02    9.599421E+01    1.000000E+00    3.637892E+05    4.134745E-13    7.414896E-11
      3        3.746071E+05    6.120516E+02    9.741103E+01    1.000000E+00    3.746071E+05    9.182545E-13    5.826327E-11


 RESPONSE SPECTRUM APPLICATION                                       SUBCASE 1

                                      M O D A L   P A R T I C I P A T I O N   F A C T O R S

    MODE          T1            T2            T3            R1            R2            R3
   NUMBER
      1      3.244768E-15  3.389699E-15 -1.595750E-02 -3.588142E-13  6.868300E-02  1.086650E-14
      2      1.622374E-14  1.619478E-14 -2.710059E-13  9.735860E-03  3.286705E-13  5.097880E-14
      3     -4.704441E-15 -6.709818E-15  9.083581E-03  3.435392E-13 -1.078962E-02 -2.374460E-14


 RESPONSE SPECTRUM APPLICATION                                       SUBCASE 1

                                             M O D A L   E F F E C T I V E   M A S S

    MODE          T1            T2            T3            R1            R2            R3
   NUMBER
      1      0.000000E+00  0.000000E+00  2.546418E-04  0.000000E+00  4.717354E-03  0.000000E+00
      2      0.000000E+00  0.000000E+00  0.000000E+00  9.478698E-05  0.000000E+00  0.000000E+00
      3      0.000000E+00  0.000000E+00  8.251145E-05  0.000000E+00  1.164159E-04  0.000000E+00


   TOTAL     0.000000E+00  0.000000E+00  3.371532E-04  9.478698E-05  4.833770E-03  0.000000E+00

 RESPONSE SPECTRUM APPLICATION                                       SUBCASE 1

                                               P E R C E N T   M O D A L   M A S S

  DIRECTION     TOTAL         MODAL       PERCENT

      1      2.588404E+00  0.000000E+00     0.00
      2      2.588404E+00  0.000000E+00     0.00
      3      2.588404E+00  3.371532E-04     0.01
      4      0.000000E+00  9.478698E-05     0.00
      5      0.000000E+00  4.833770E-03     0.00
      6      0.000000E+00  0.000000E+00     0.00


 RESPONSE SPECTRUM APPLICATION                                       SUBCASE 1

                                              P E A K   M O D A L   R E S P O N S E

    MODE     DISPLACEMENT        VELOCITY        ACCELERATION
   NUMBER
      1     -3.643410E-02     -3.037426E+00     -2.532231E+02
      2     -5.456173E-13     -3.290888E-10     -1.984897E-07
      3     -1.825573E-02     -1.117345E+01     -6.838726E+03
```

Listing 6-4b. Vector Results Output from the Application of Response Spectra to the 3-D Cantilever Beam.

```
 RESPONSE SPECTRUM APPLICATION                                       SUBCASE 1

                                           D I S P L A C E M E N T   V E C T O R

    GRID      COORDINATE        T1            T2            T3            R1            R2            R3
     ID          ID
      4           0      7.696644E-13  5.006702E-13  1.693628E+00  1.823400E-01  1.353139E+00  8.224218E-13
      5           0      6.112543E-13  4.994562E-13  1.638807E+00  4.134821E-11  1.298488E+00  8.108710E-13
      6           0      4.320838E-13  4.998164E-13  1.693628E+00  1.823400E-01  1.353139E+00  8.249431E-13
      7           0      7.886313E-13  1.211846E-12  5.508413E+00  9.346200E-02  2.828237E+00  8.268599E-13
      8           0      6.146136E-13  1.223904E-12  5.547796E+00  1.648646E-10  2.736246E+00  8.198638E-13
      9           0      4.364489E-13  1.223711E-12  5.508413E+00  9.346200E-02  2.828237E+00  8.239522E-13
     10           0      7.766624E-13  8.577293E-13  3.057897E+00  3.149866E-01  1.709608E+00  8.235605E-13
     11           0      6.011629E-13  8.555496E-13  2.943454E+00  1.124769E-10  1.686283E+00  8.294328E-13
     12           0      4.342409E-13  8.501656E-13  3.057897E+00  3.149866E-01  1.709608E+00  8.339933E-13
     16           0      0.000000E+00  0.000000E+00  2.887229E-04  0.000000E+00  0.000000E+00  0.000000E+00
     17           0      0.000000E+00  0.000000E+00  2.887229E-04  0.000000E+00  0.000000E+00  0.000000E+00
     18           0      0.000000E+00  0.000000E+00  2.887229E-04  0.000000E+00  0.000000E+00  0.000000E+00


MAXIMUM DISPLACEMENT MAGNITUDE =  5.547796E+00  AT GRID 8
MAXIMUM ROTATION MAGNITUDE     =  2.829781E+00  AT GRID 7


 RESPONSE SPECTRUM APPLICATION                                       SUBCASE 1

                                               V E L O C I T Y   V E C T O R

    GRID      COORDINATE        T1            T2            T3            R1            R2            R3
     ID          ID
      4           0      2.438172E-10  1.645561E-10  7.452888E+02  9.275189E+01  5.450662E+02  2.587390E-10
      5           0      1.865214E-10  1.629984E-10  7.024558E+02  2.509592E-08  5.026665E+02  2.552615E-10
      6           0      1.229596E-10  1.640161E-10  7.452888E+02  9.275189E+01  5.450662E+02  2.599517E-10
      7           0      2.553289E-10  4.327515E-10  1.441168E+03  5.449291E+01  1.261542E+03  2.652451E-10
      8           0      1.883316E-10  4.364746E-10  1.466762E+03  9.986235E-08  1.210252E+03  2.607160E-10
      9           0      1.200821E-10  4.362571E-10  1.441168E+03  5.449291E+01  1.261542E+03  2.641682E-10
     10           0      2.507568E-10  2.971623E-10  8.536373E+02  1.708707E+02  6.122323E+02  2.640471E-10
     11           0      1.840528E-10  2.960730E-10  7.727822E+02  6.795725E-08  6.052361E+02  2.635962E-10
     12           0      1.199273E-10  2.946968E-10  8.536373E+02  1.708707E+02  6.122323E+02  2.674478E-10
     16           0      0.000000E+00  0.000000E+00  5.794540E-02  0.000000E+00  0.000000E+00  0.000000E+00
     17           0      0.000000E+00  0.000000E+00  5.794540E-02  0.000000E+00  0.000000E+00  0.000000E+00
     18           0      0.000000E+00  0.000000E+00  5.794540E-02  0.000000E+00  0.000000E+00  0.000000E+00


MAXIMUM VELOCITY MAGNITUDE         =  1.466762E+03  AT GRID 8
MAXIMUM ANGULAR VELOCITY MAGNITUDE =  1.262718E+03  AT GRID 9


 RESPONSE SPECTRUM APPLICATION                                       SUBCASE 1

                                           A C C E L E R A T I O N   V E C T O R

    GRID      COORDINATE        T1            T2            T3            R1            R2            R3
     ID          ID
      4           0      1.302828E-07  8.888861E-08  4.318702E+05  5.519750E+04  3.100052E+05  1.379678E-07
      5           0      9.852113E-08  8.786729E-08  4.048806E+05  1.524535E-05  2.833082E+05  1.361388E-07
      6           0      6.346126E-08  8.855668E-08  4.318702E+05  5.519750E+04  3.100052E+05  1.386824E-07
      7           0      1.373205E-07  2.391088E-07  7.211474E+05  3.312649E+04  7.329888E+05  1.422657E-07
      8           0      9.960854E-08  2.410828E-07  7.369368E+05  6.065196E-05  7.020150E+05  1.394731E-07
      9           0      6.123748E-08  2.409413E-07  7.211474E+05  3.312649E+04  7.329888E+05  1.416653E-07
     10           0      1.347516E-07  1.628865E-07  4.376057E+05  1.027545E+05  3.385248E+05  1.416010E-07
     11           0      9.731934E-08  1.622402E-07  3.872169E+05  4.125944E-05  3.348497E+05  1.409878E-07
     12           0      6.124249E-08  1.615579E-07  4.376057E+05  1.027545E+05  3.385248E+05  1.434336E-07
     16           0      0.000000E+00  0.000000E+00  2.556417E+01  0.000000E+00  0.000000E+00  0.000000E+00
     17           0      0.000000E+00  0.000000E+00  2.556417E+01  0.000000E+00  0.000000E+00  0.000000E+00
     18           0      0.000000E+00  0.000000E+00  2.556417E+01  0.000000E+00  0.000000E+00  0.000000E+00


MAXIMUM ACCELERATION MAGNITUDE         =  7.369368E+05  AT GRID 8
MAXIMUM ANGULAR ACCELERATION MAGNITUDE =  7.337370E+05  AT GRID 9
```

Listing 6-4c. Element Results Output from the Application of Response Spectra to the 3-D Cantilever Beam.

```
 RESPONSE SPECTRUM APPLICATION                                       SUBCASE 1


                       S T R E S S E S   I N   Q U A D   E L E M E N T S   O N   S U R F A C E   0

 SURFACE COORDINATE ID = ELEMENT  X-AXIS = X  NORMAL = Z

   ELEMENT      GRID        FIBER       STRESSES IN SURFACE COORDINATE SYSTEM      PRINCIPAL STRESSES (ZERO SHEAR)          HENCKY
     ID          ID        DISTANCE      NORMAL-X      NORMAL-Y     SHEAR-XY        ANGLE         MAJOR         MINOR         VON MISES

       1      CENTER   -1.50000E-02    1.14672E+05  4.61691E+04  5.06617E+03    -88.2843    8.24159E+04  7.84250E+04    1.01105E+05
                        1.50000E-02    1.14672E+05  4.61691E+04  5.06617E+03      1.7157    7.84250E+04  8.24159E+04    1.01105E+05

       2      CENTER   -1.50000E-02    1.49113E+05  1.19077E+04  3.70796E+03    -89.5002    1.17999E+04  1.49220E+05    1.43693E+05
                        1.50000E-02    1.49113E+05  1.19077E+04  3.70796E+03      0.4998    1.49220E+05  1.17999E+04    1.43693E+05

       3      CENTER   -1.50000E-02    8.02007E+04  3.39832E+03  6.51894E+03     78.0803    4.02287E+03  8.08253E+04    8.29987E+04
                        1.50000E-02    8.02007E+04  3.39832E+03  6.51894E+03    -11.9197    8.08253E+04  4.02287E+03    8.29987E+04

       4      CENTER   -1.50000E-02    1.14672E+05  4.61691E+04  5.06617E+03     88.2843    8.24159E+04  7.84250E+04    1.01105E+05
                        1.50000E-02    1.14672E+05  4.61691E+04  5.06617E+03     -1.7157    7.84250E+04  8.24159E+04    1.01105E+05

       5      CENTER   -1.50000E-02    1.49113E+05  1.19077E+04  3.70796E+03     89.5002    1.17999E+04  1.49220E+05    1.43693E+05
                        1.50000E-02    1.49113E+05  1.19077E+04  3.70796E+03     -0.4998    1.49220E+05  1.17999E+04    1.43693E+05

       6      CENTER   -1.50000E-02    8.02007E+04  3.39832E+03  6.51894E+03    -78.0803    4.02287E+03  8.08253E+04    8.29987E+04
                        1.50000E-02    8.02007E+04  3.39832E+03  6.51894E+03     11.9197    8.08253E+04  4.02287E+03    8.29987E+04



MAXIMUM QUAD ELEMENT PRINCIPAL STRESS =  1.492204E+05  AT ELEMENT 5
MINIMUM QUAD ELEMENT PRINCIPAL STRESS =  8.082528E+04  AT ELEMENT 6
MAXIMUM QUAD ELEMENT SHEAR STRESS     =  6.871025E+04  AT ELEMENT 5
MAXIMUM QUAD ELEMENT VON MISES STRESS =  1.436931E+05  AT ELEMENT 5
```

6.8 Dynamic Design Analysis Method (DDAM).

DDAM is a form of shock spectrum analysis where user supplied shock coefficients are used to perform shock excitation calculations and generate shock spectrum data. Additionally, a modal cutoff percentage may be supplied to terminate calculations when a specific modal mass is reached. Shock spectrum data is then applied using the modal summation conventions outlined in Sections 6.5 – 6.7.

DDAM analysis is performed using a standard normal modes solution (see Section 4, *Normal Modes Analysis*, for more information) and the following additional Case Control commands and Bulk Data entries.

`DDAM`	Selects the `DDAMDATA` Bulk Data entry
`DDAMDATA`	Defines data needed to perform DDAM analysis
`PARAM, MODALDATABASE,` *option*	Model parameter to control storage and retrieval of modal data DELETE = Modal database is deleted STORE = Modal database is stored FETCH = Modal database is retrieved (default is DELETE)
`PARAM, SORTMODALMASS,` *option*	Model parameter to control order modes are summed in DDAM analysis ON = In order of increasing modal mass OFF = In order of increasing eigenvalue (default is ON)

See *Nastran Solver Reference Guide*, Section 4, *Bulk Data* and Section 5, *Parameters,* for more information on each entry.

6.9 Interpreting Results

As an example we will use the cantilever beam shown in Figure 6-6. It is desired to find the response of the beam when subjected to the specified DDAM shock environment. Listing 6-5 contains the Model Input File and Listing 6-6 the results.

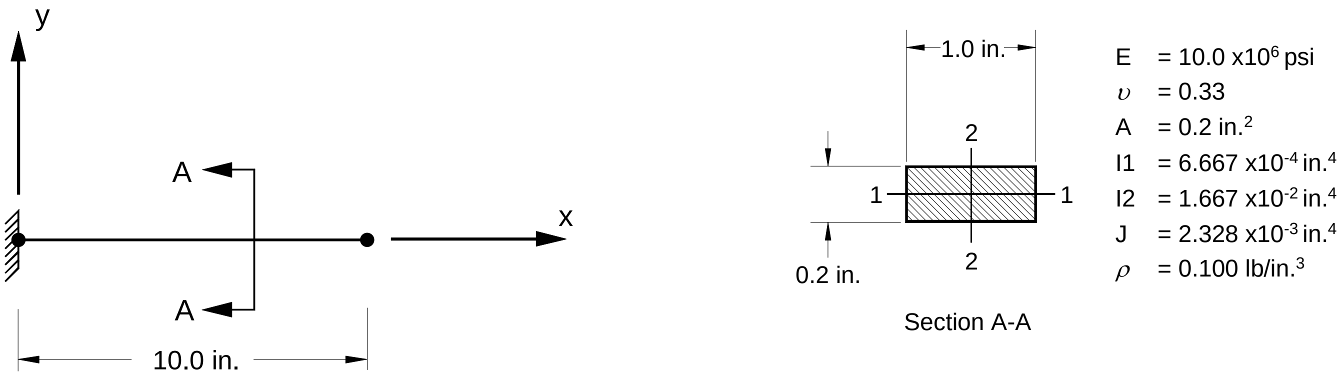


Figure 6-6. 2-D Cantilever Beam Example Problem.

Listing 6-5a. Model Input File for the DDAM Analysis of a 3-D Cantilever Beam.

```
$
$ MODAL SOLUTION.
$
SOL MODAL
CEND
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = DDAM ANALYSIS OF A 3-D CANTILEVER BEAM
$
DISPLACEMENT = ALL
VELOCITY = ALL
ACCELERATION = ALL
STRESS = ALL
$
SUBCASE 1
 SPC = 1
 METHOD = 1
 DDAM = 1
$
BEGIN BULK
$
$ INSERT DDAM DATA.
$
INCLUDE 'DDAM.NAS'
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  20
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  1.,  0.
CBAR,  2,  10,   2,   3,  0.,  1.,  0.
CBAR,  3,  10,   3,   4,  0.,  1.,  0.
CBAR,  4,  10,   4,   5,  0.,  1.,  0.
CBAR,  5,  10,   5,   6,  0.,  1.,  0.
CBAR,  6,  10,   6,   7,  0.,  1.,  0.
CBAR,  7,  10,   7,   8,  0.,  1.,  0.
CBAR,  8,  10,   8,   9,  0.,  1.,  0.
CBAR,  9,  10,   9,  10,  0.,  1.,  0.
CBAR, 10,  10,  10,  11,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3,
,  -0.1,  0.5,  0.1,  0.5,  -0.1,  -0.5,  0.1,  -0.5
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
```

Listing 6-5a. Model Input File for the DDAM Analysis of a 3-D Cantilever Beam. (Continued)

```
$
$ FIXED AT ONE END, FREE AT OTHER.
$
SPC1,  1,  123456,  1
SPC1,  1,       4,  1,  THRU,  11
ENDDATA
```

Listing 6-5b. DDAM Data File Containing Shock Coefficients and Analysis Parameters.

```
$
$ DDAM COEFFICIENTS AND ANALYSIS PARAMETERS.
$
DDAMDAT,  1,  0.25,  0.5,  1.,  0.25,  0.5,  1.,  10.,
,  20.,  50.,  10.,  37.5,  6.,  ,  SURFACE,  DECK,
,  123,  1,  3,  386.4,  6.,  80.
```

Listing 6-5a is set up like a typical normal modes analysis with the exception of the `DDAM` Case Control command and `DDAMDAT` Bulk Data entry. The `DDAM` command initiates a shock spectrum solution sequence and references the `DDAMDAT` entry. The `DDAMDAT` entry shown in Listing 6-5b contains the shock environment in coefficient form and the analysis control settings. The analysis control settings specify shock directions and their labels and the modal mass cutoff percentage at which shock excitation calculations cease.

In our example analysis all three shock directions are requested though only the results of the third (vertical direction) will be presented. A subcase is created automatically for each direction requested on the `DDAMDAT` entry.

Listing 6-6a shows a summary of the user specified analysis control settings. Listing 6-6b shows the verification output, which gives the individual and cumulative modal weights through the calculated frequencies. The cumulative modal weight should be checked in each shock direction to verify the required percentage is achieved. Failure to meet the required modal weight requires either increasing the number of desired modes or the frequency range on the `EIGRL` Bulk Data entry and rerunning the model. Listing 6-6c and 6-6d gives the combined modal results using the NRL method (default for DDAM analysis).

Listing 6-6a. Model Results Output DDAM Analysis Data Definition.

```
                                        D D A M   A N A L Y S I S   D A T A   D E F I N I T I O N



 DDAM DATA SET      = 1

 SHIP TYPE          = SURFACE
 MOUNTING LOCATION  = DECK
 MATERIAL TYPE      = ELASTIC
 SUMMATION METHOD   = NRL

 TOTAL MASS         = 5.176000E-04
 TOTAL WEIGHT       = 2.000006E-01
 CONVERSION FACTOR  = 3.864000E+02
```

Listing 6-6b. Model Results Output for the DDAM Analysis of a 3-D Cantilever Beam.

```
  VERTICAL (Z) DIRECTED SHOCK                                           SUBCASE 3

                                          M O D A L   E F F E C T I V E   W E I G H T

      MODE        CYCLES      PARTICIPATION           MODAL              CUMULATIVE
     NUMBER                FACTOR          WEIGHT     PERCENT       WEIGHT     PERCENT
        1      6.320250E+01    0.000000E+00    0.000000E+00   0.0000    0.000000E+00   0.0000
        2      3.160362E+02    1.777455E-02    1.220771E-01  61.0383    1.220771E-01  61.0383
        3      3.910655E+02    0.000000E+00    0.000000E+00   0.0000    1.220771E-01  61.0383
        4      1.081427E+03    0.000000E+00    0.000000E+00   0.0000    1.220771E-01  61.0383
        5      1.955474E+03   -9.873827E-03    3.767108E-02  18.8355    1.597482E-01  79.8738
        6      2.090234E+03    0.000000E+00    0.000000E+00   0.0000    1.597482E-01  79.8738
        7      3.404135E+03    0.000000E+00    0.000000E+00   0.0000    1.597482E-01  79.8738
        8      4.909206E+03    0.000000E+00    0.000000E+00   0.0000    1.597482E-01  79.8738
        9      4.998670E+03    0.000000E+00    0.000000E+00   0.0000    1.597482E-01  79.8738
       10      5.407542E+03   -5.776554E-03    1.289362E-02   6.4468    1.726418E-01  86.3206


 MASS AVAILABLE  = 92.8737 PERCENT
 MASS USED       = 86.3206 PERCENT


VERTICAL (Z) DIRECTED SHOCK                                           SUBCASE 3

                                                   M O D A L   R E A C T I O N

      MODE        CYCLES      PARTICIPATION            RESPONSE                        INPUT
     NUMBER                FACTOR          ACCELERATION      REACTION      ACCELERATION     SOURCE
        1      6.320250E+01    0.000000E+00    0.000000E+00    0.000000E+00    6.000000E+00    MINIMUM-G
        2      3.160362E+02    1.777455E-02    1.411812E+02    2.509431E+00    2.055612E+01    VELOCITY
        3      3.910655E+02    0.000000E+00    0.000000E+00    0.000000E+00    2.543620E+01    VELOCITY
        4      1.081427E+03    0.000000E+00    0.000000E+00    0.000000E+00    6.250000E+01    ACCELERATION
        5      1.955474E+03   -9.873827E-03    2.384517E+02   -2.354430E+00    6.249967E+01    ACCELERATION
        6      2.090234E+03    0.000000E+00    0.000000E+00    0.000000E+00    6.250000E+01    ACCELERATION
        7      3.404135E+03    0.000000E+00    0.000000E+00    0.000000E+00    6.250000E+01    ACCELERATION
        8      4.909206E+03    0.000000E+00    0.000000E+00    0.000000E+00    6.250000E+01    ACCELERATION
        9      4.998670E+03    0.000000E+00    0.000000E+00    0.000000E+00    6.250000E+01    ACCELERATION
       10      5.407542E+03   -5.776554E-03    1.395035E+02   -8.058497E-01    6.249989E+01    ACCELERATION

VERTICAL (Z) DIRECTED SHOCK                                           SUBCASE 3

                                              T O T A L   B A S E   R E A C T I O N

    COMPONENT     MAXIMUM      REACTION          SRSS          NRL SUM
                   MODE
        1            8       0.000000E+00    0.000000E+00    0.000000E+00
        2            1       0.000000E+00    0.000000E+00    0.000000E+00
        3            2       2.509431E+00    2.488521E+00    4.997952E+00
        4            0       0.000000E+00    0.000000E+00    0.000000E+00
        5            2       1.831502E+01    5.077361E+00    2.339238E+01
        6            1       0.000000E+00    0.000000E+00    0.000000E+00

  VERTICAL (Z) DIRECTED SHOCK                                           SUBCASE 3

                                              P E A K   M O D A L   R E S P O N S E

      MODE     DISPLACEMENT        VELOCITY        ACCELERATION
     NUMBER
        1      0.000000E+00      0.000000E+00      0.000000E+00
        2      3.580498E-05      7.109844E-02      1.411812E+02
        3      0.000000E+00      0.000000E+00      0.000000E+00
        4      0.000000E+00      0.000000E+00      0.000000E+00
        5      1.579561E-06      1.940745E-02      2.384517E+02
        6      0.000000E+00      0.000000E+00      0.000000E+00
        7      0.000000E+00      0.000000E+00      0.000000E+00
        8      0.000000E+00      0.000000E+00      0.000000E+00
        9      0.000000E+00      0.000000E+00      0.000000E+00
       10      1.208442E-07      4.105872E-03      1.395035E+02
```

Listing 6-6c. Vector Results Output for the DDAM Analysis of a 3-D Cantilever Beam.

```
 VERTICAL (Z) DIRECTED SHOCK                                         SUBCASE 3

                                              D I S P L A C E M E N T   V E C T O R

    GRID      COORDINATE        T1            T2            T3            R1            R2            R3
     ID           ID
      2             0      0.000000E+00  0.000000E+00  6.518836E-05  0.000000E+00  1.254091E-04  0.000000E+00
      3             0      0.000000E+00  0.000000E+00  2.410175E-04  0.000000E+00  2.214736E-04  0.000000E+00
      4             0      0.000000E+00  0.000000E+00  4.988605E-04  0.000000E+00  2.900596E-04  0.000000E+00
      5             0      0.000000E+00  0.000000E+00  8.126876E-04  0.000000E+00  3.349217E-04  0.000000E+00
      6             0      0.000000E+00  0.000000E+00  1.160542E-03  0.000000E+00  3.721221E-04  0.000000E+00
      7             0      0.000000E+00  0.000000E+00  1.525512E-03  0.000000E+00  4.227767E-04  0.000000E+00
      8             0      0.000000E+00  0.000000E+00  1.896187E-03  0.000000E+00  4.614182E-04  0.000000E+00
      9             0      0.000000E+00  0.000000E+00  2.280511E-03  0.000000E+00  4.857448E-04  0.000000E+00
     10             0      0.000000E+00  0.000000E+00  2.771175E-03  0.000000E+00  4.969525E-04  0.000000E+00
     11             0      0.000000E+00  0.000000E+00  3.269731E-03  0.000000E+00  4.996089E-04  0.000000E+00


MAXIMUM DISPLACEMENT MAGNITUDE =  3.269731E-03  AT GRID 11
MAXIMUM ROTATION MAGNITUDE     =  4.996089E-04  AT GRID 11


 VERTICAL (Z) DIRECTED SHOCK                                         SUBCASE 3

                                                  V E L O C I T Y   V E C T O R

    GRID      COORDINATE        T1            T2            T3            R1            R2            R3
     ID           ID
      2             0      0.000000E+00  3.196421E-15  2.774292E-01  0.000000E+00  5.118434E-01  5.223227E-15
      3             0      0.000000E+00  8.550874E-15  9.414644E-01  0.000000E+00  7.810605E-01  5.929024E-15
      4             0      0.000000E+00  1.254578E-14  1.767875E+00  0.000000E+00  8.639533E-01  7.107284E-15
      5             0      0.000000E+00  1.801354E-14  2.592429E+00  0.000000E+00  8.579103E-01  4.465469E-15
      6             0      0.000000E+00  1.985898E-14  3.316326E+00  0.000000E+00  9.284067E-01  0.000000E+00
      7             0      0.000000E+00  1.675528E-14  3.886363E+00  0.000000E+00  1.143220E+00  5.340543E-15
      8             0      0.000000E+00  1.175145E-14  4.280451E+00  0.000000E+00  1.383732E+00  8.401322E-15
      9             0      0.000000E+00  4.195788E-15  4.687265E+00  0.000000E+00  1.581798E+00  1.004263E-14
     10             0      0.000000E+00  8.213237E-15  6.214531E+00  0.000000E+00  1.693281E+00  1.409348E-14
     11             0      0.000000E+00  2.308426E-14  7.913390E+00  0.000000E+00  1.723525E+00  1.560215E-14


MAXIMUM VELOCITY MAGNITUDE         =  7.913390E+00  AT GRID 11
MAXIMUM ANGULAR VELOCITY MAGNITUDE =  1.723525E+00  AT GRID 11


 VERTICAL (Z) DIRECTED SHOCK                                         SUBCASE 3

                                              A C C E L E R A T I O N   V E C T O R

    GRID      COORDINATE        T1            T2            T3            R1            R2            R3
     ID           ID
      2             0      0.000000E+00  1.988682E-11  3.497311E+03  0.000000E+00  6.024982E+03  2.646874E-11
      3             0      0.000000E+00  3.381270E-11  1.026438E+04  0.000000E+00  6.842889E+03  1.956346E-11
      4             0      0.000000E+00  3.301914E-11  1.584517E+04  0.000000E+00  5.258847E+03  4.442006E-11
      5             0      0.000000E+00  5.428613E-11  1.841657E+04  0.000000E+00  6.369994E+03  3.994589E-11
      6             0      0.000000E+00  6.318238E-11  1.903135E+04  0.000000E+00  8.077564E+03  1.683356E-11
      7             0      0.000000E+00  4.259609E-11  1.916863E+04  0.000000E+00  7.821655E+03  3.021701E-11
      8             0      0.000000E+00  3.412415E-11  1.776743E+04  0.000000E+00  8.491689E+03  3.380699E-11
      9             0      0.000000E+00  2.659331E-11  1.428505E+04  0.000000E+00  1.193285E+04  2.585953E-11
     10             0      0.000000E+00  1.886599E-11  2.124213E+04  0.000000E+00  1.478642E+04  5.805221E-11
     11             0      0.000000E+00  8.121098E-11  3.565282E+04  0.000000E+00  1.567313E+04  7.688031E-11


MAXIMUM ACCELERATION MAGNITUDE         =  3.565282E+04  AT GRID 11
MAXIMUM ANGULAR ACCELERATION MAGNITUDE =  1.567313E+04  AT GRID 11
```

Listing 6-6d. Element Results Output for the DDAM Analysis of a 3-D Cantilever Beam.

```
DDAM ANALYSIS OF A 3-D CANTILEVER BEAM

  VERTICAL (Z) DIRECTED SHOCK                                          SUBCASE 3

                                       S T R E S S E S   I N   B A R   E L E M E N T S

    ELEMENT         DISTANCE      SX-C           SX-D           SX-E           SX-F           AXIAL           SX-MAX         SX-MIN
      ID

       1            0.0000   7.016311E+02  7.016311E+02  7.016311E+02  7.016311E+02   0.000000E+00   7.016311E+02  7.016311E+02
                    0.0000   5.528090E+02  5.528090E+02  5.528090E+02  5.528090E+02   0.000000E+00   5.528090E+02  5.528090E+02

       2            0.0000   5.527326E+02  5.527326E+02  5.527326E+02  5.527326E+02   0.000000E+00   5.527326E+02  5.527326E+02
                    0.0000   4.158424E+02  4.158424E+02  4.158424E+02  4.158424E+02   0.000000E+00   4.158424E+02  4.158424E+02

       3            0.0000   4.158269E+02  4.158269E+02  4.158269E+02  4.158269E+02   0.000000E+00   4.158269E+02  4.158269E+02
                    0.0000   3.776503E+02  3.776503E+02  3.776503E+02  3.776503E+02   0.000000E+00   3.776503E+02  3.776503E+02

       4            0.0000   3.776992E+02  3.776992E+02  3.776992E+02  3.776992E+02   0.000000E+00   3.776992E+02  3.776992E+02
                    0.0000   3.446546E+02  3.446546E+02  3.446546E+02  3.446546E+02   0.000000E+00   3.446546E+02  3.446546E+02

       5            0.0000   3.446550E+02  3.446550E+02  3.446550E+02  3.446550E+02   0.000000E+00   3.446550E+02  3.446550E+02
                    0.0000   2.958716E+02  2.958716E+02  2.958716E+02  2.958716E+02   0.000000E+00   2.958716E+02  2.958716E+02

       6            0.0000   2.958301E+02  2.958301E+02  2.958301E+02  2.958301E+02   0.000000E+00   2.958301E+02  2.958301E+02
                    0.0000   2.332558E+02  2.332558E+02  2.332558E+02  2.332558E+02   0.000000E+00   2.332558E+02  2.332558E+02

       7            0.0000   2.331687E+02  2.331687E+02  2.331687E+02  2.331687E+02   0.000000E+00   2.331687E+02  2.331687E+02
                    0.0000   1.616366E+02  1.616366E+02  1.616366E+02  1.616366E+02   0.000000E+00   1.616366E+02  1.616366E+02

       8            0.0000   1.614848E+02  1.614848E+02  1.614848E+02  1.614848E+02   0.000000E+00   1.614848E+02  1.614848E+02
                    0.0000   8.967301E+01  8.967301E+01  8.967301E+01  8.967301E+01   0.000000E+00   8.967301E+01  8.967301E+01

       9            0.0000   8.943972E+01  8.943972E+01  8.943972E+01  8.943972E+01   0.000000E+00   8.943972E+01  8.943972E+01
                    0.0000   2.928275E+01  2.928275E+01  2.928275E+01  2.928275E+01   0.000000E+00   2.928275E+01  2.928275E+01

      10            0.0000   2.897641E+01  2.897641E+01  2.897641E+01  2.897641E+01   0.000000E+00   2.897641E+01  2.897641E+01
                    0.0000   1.797935E-01  1.797935E-01  1.797935E-01  1.797935E-01   0.000000E+00   1.797935E-01  1.797935E-01



 MAXIMUM BAR ELEMENT TOTAL STRESS =  7.016311E+02  AT ELEMENT 1
 MINIMUM BAR ELEMENT TOTAL STRESS =  2.897641E+01  AT ELEMENT 10
```

7. FREQUENCY RESPONSE ANALYSIS

7.1 Introduction

Frequency response analysis solves for the steady state response (amplitudes and phase angles of displacements, velocities, accelerations, forces, stresses, and strains) of structures subjected to sinusoidal (harmonic) loading. Examples of oscillatory excitation include rotating machinery, unbalanced tires, and propeller blades. Unlike transient response where the excitation is explicitly defined in the time domain, in frequency response it is defined in the frequency domain. Applied loads are specified as a function of frequency.

Oscillatory loading is inherently sinusoidal. At a rudimentary level, this loading is defined as having amplitude at a specific frequency. The steady-state oscillatory response occurs at the same frequency as the loading. Damping results in a shift in time of the response. This shift in response is called a phase shift because the peak loading and response no longer occur at the same time. Figure 7-1 depicts this graphically.

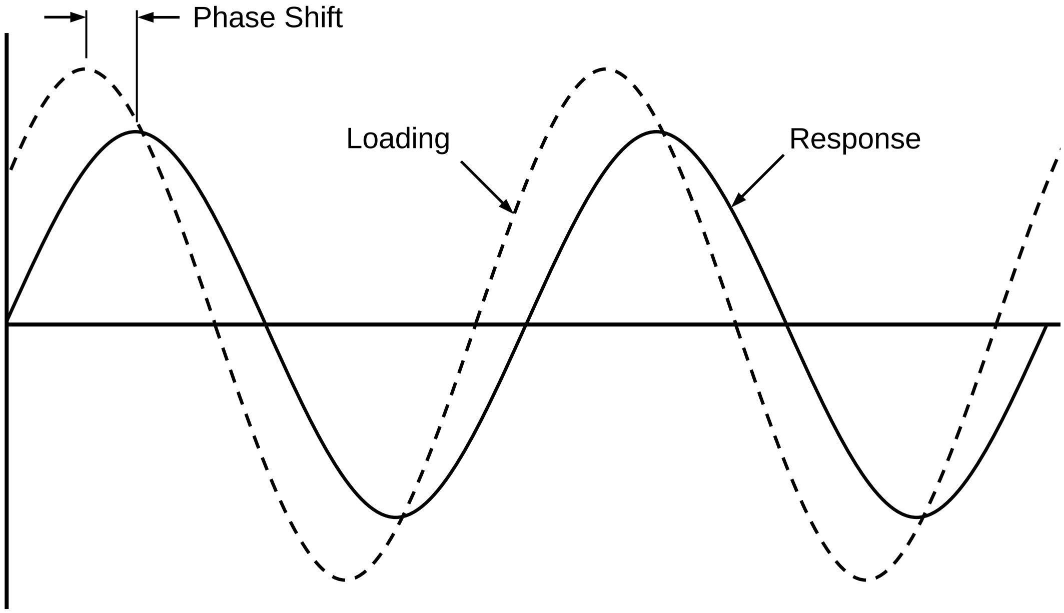


Figure 7-1. Loading Versus Response in a Damped System.

There are two methods available for performing frequency response analysis: direct and modal. In the direct method, the structural response is computed at discrete excitation frequencies by solving a set of coupled matrix equations using complex algebra. The modal method uses the mode shapes of the structure to reduce and uncouple the equations of motion. The solution is then obtained through the summation of the individual modal responses. Generally the modal method is more efficient especially for larger models where a large number of solution frequencies are specified. The direct method may be more efficient for models where high-frequency excitation require the extraction of a large number of modes. The direct method may also be more accurate because there are no mode truncation effects.

Autodesk Inventor Nastran will also handle frequency response of structures under initial stress, for example the forced vibration of a cable in tension. For more information see Section 13, *Linear Prestress Frequency Response Analysis*.

7.1.1 Direct Frequency Response Analysis

In direct frequency response structural response is computed at discrete excitation frequencies by solving a set of coupled matrix equations using complex algebra. We begin with the damped forced vibration equation of motion with harmonic excitation

$$[M]\{\ddot{x}(t)\}+[B]\{\dot{x}(t)\}+[K]\{x(t)\}=\{P(\omega)\}e^{i\omega t}$$

The load is introduced as a complex vector and can be real or imaginary or both. The same interpretation can be used for response quantities.

For harmonic motion (which is the basis of a frequency response analysis), assume a solution of the form:

$$\{x\}=\{u(\omega)\}e^{i\omega t}$$

where $\{u(\omega)\}$ is the complex displacement vector. Taking the first and second derivatives gives:

$$\{\dot{x}\}=i\omega\{u(\omega)\}e^{i\omega t}$$

$$\{\ddot{x}\}=-\omega^2\{u(\omega)\}e^{i\omega t}$$

Substituting into the equation of motion and dividing by $e^{i\omega t}$ we have:

$$-\left(\omega^2[M]+i\omega[B]+[K]\right)\{u(\omega)\}=\{P(\omega)\}$$

The equation is then solved for a given forcing frequency ω. This expression represents a system of equations with complex coefficients if damping is included or the applied loads have phase angles. The equations of motion at each solution frequency are then solved in a manner similar to a statics problem using complex arithmetic.

The damping matrix $[B]$ is used to represent energy dissipation characteristics of the structure. Damping in direct frequency response is represented by the damping matrix $[B]$ and additions to the stiffness matrix $[K]$. The damping matrix is given by:

$$[B]=[B_1]+[B_2]+\alpha[K]+\beta[M]$$

where,

$[B_1]$	damping from damping elements (`CVISC`, `CDAMPi`) and `B2GG DMIG`
$[B_2]$	damping from `B2PP DMIG`
$[K]$	global stiffness matrix
$[M]$	global mass matrix
α	Rayleigh damping stiffness matrix scale factor
β	Rayleigh damping mass matrix scale factor

In frequency response, `PARAM, G` and GE on the `MATi` entry do not form a damping matrix. Instead they form the complex stiffness matrix:

$$[K] = (1+iG)[K] + i\sum G_{ELEM} K_{ELEM}$$

where,

$[K]$	global stiffness matrix
$[K_{ELEM}]$	element stiffness matrix
G	overall structural damping coefficient (`PARAM, G`)
G_{ELEM}	element structural damping coefficient (`GE` on the `MATi` entry)

When the above parameters and/or coefficients are specified, they are automatically incorporated into the stiffness matrix and therefore into the equation of motion for the solution. In frequency response analysis it is not necessary to assume an equivalent viscous form for structural damping since the solution is complex.

7.1.2 Modal Frequency Response Analysis

Modal frequency response is an alternative method for computing the frequency response of a structure. The modal method uses the mode shapes of the structure to reduce and uncouple the equations of motion. The solution is then obtained through the summation of the individual modal responses.

The general equation of equilibrium for a finite element system in motion is:

$$[M]\{\ddot{x}(t)\}+[B]\{\dot{x}(t)\}+[K]\{x(t)\}=\{P(\omega)\}e^{i\omega t}$$

where,

$[K]$	is the global stiffness matrix
$[M]$	is the global mass matrix
$[B]$	is the global damping matrix
$\{P\}$	is the global load vector
$\{\ddot{x}\}$	is the global acceleration vector
$\{\dot{x}\}$	is the global velocity vector
$\{x\}$	is the global displacement vector

For harmonic motion assume a solution of the form:

$$\{x\} = \{u(\omega)\} e^{i\omega t}$$

Next, the variables are transformed from physical coordinates $\{u\}$ to modal coordinates $\{\xi\}$ by assuming

$$\{x\} = [\phi]\{\xi(\omega)\} e^{i\omega t}$$

The mode shapes $\{\phi\}$ are used to transform the problem in terms of the behavior of the modes as opposed to the behavior of the grid points.

$$-\omega^2[\phi]^T[M][\phi]\{\xi(\omega)\} + i\omega[\phi]^T[B][\phi]\{\xi(\omega)\} + [\phi]^T[K][\phi]\{\xi(\omega)\} = [\phi]^T\{P(\omega)\}$$

where,

$[\phi]^T[M][\phi]$	is the modal or generalized mass matrix
$[\phi]^T[B][\phi]$	is the modal or generalized stiffness matrix
$[\phi]^T[K][\phi]$	is the modal damping matrix
$[\phi]^T[P]$	is the modal force vector

Using the orthogonality property of the mode shapes we can formulate the equations of motion in terms of the diagonal generalized mass, stiffness, and damping (modal damping). Since these matrices do not have off-diagonal terms that couple the equations of motion, the modal equations of motion are uncoupled. The equations of motion can then be written as:

$$-\omega^2 m_i \xi_i(\omega) + i\omega b_i \xi_i(\omega) + k_i \xi_i(\omega) = p_i(\omega)$$

where,

m_i = i-th modal mass
b_i = i-th modal damping
k_i = i-th modal stiffness
p_i = i-th modal force
ξ_i = i-th modal degree of freedom

Then, each of the modal responses can be calculated using:

$$\xi_i(\omega) = \frac{p_i(\omega)}{-m_i\omega^2 + ib_i\omega + k_i}$$

The physical responses are then recovered from the summation of the individual modal responses using

$$\{x\} = [\phi]\{\xi(\omega)\} e^{i\omega t}$$

These responses are in complex form (magnitude/phase or real/imaginary) and are used to recover additional results quantities as requested in the Case Control Section.

7.2 How to Setup a Model Input File for Frequency Response Analysis

7.2.1 Direct Frequency Response

In Autodesk Inventor Nastran you can perform direct frequency response analysis by setting `SOLUTION = DIRECT FREQUENCY RESPONSE` in the Model Initialization File or by specifying `SOL 108` or `SOL DIRECT FREQUENCY RESPONSE` above the Case Control Section in the Model Input File. Multiple subcases can be specified, each requesting a different constraint, load, or output set.

7.2.2 Modal Frequency Response

In Autodesk Inventor Nastran you can perform modal frequency response analysis by setting `SOLUTION = MODAL FREQUENCY RESPONSE` in the Model Initialization File or by specifying `SOL 111` or `SOL MODAL FREQUENCY RESPONSE` above the Case Control Section in the Model Input File. Multiple subcases can be specified, each requesting a different load or output set. Only one reference to an `EIGRL` Bulk Data entry (`METHOD` Case Control command) is permitted. This request should be placed above the first subcase.

7.2.3 Frequency-Dependent Load Definition

Setting up a frequency response analysis can be challenging due to flexibility permitted in defining the excitation loading. The following Bulk Data entries are used to define the dynamic loading:

`RLOAD1`	Tabular input – real and imaginary
`RLOAD2`	Tabular input – magnitude and phase
`DAREA`	Spatial distribution of dynamic load
`TABLEDi`	Tabular values versus time
`LSEQ`	Generates the spatial distribution of dynamic loads from static load entries
`DLOAD`	Combines dynamic load sets
`DELAY`	Time delay
`DPHASE`	Phase lead

See *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for more information on each entry.

Dynamic load definition can be divided into two parts, one being the location and the other being the frequency variation of the loading. Spatial distribution is the characteristic which defines the location of the loading. Temporal distribution is the characteristic which defines the frequency variation. The complete dynamic load definition consists of the product of spatial and temporal distributions. This method of defining dynamic loading allows you to combine simple loadings to create complicated loading distributions that vary in position as well as frequency. Figure 7-2 describes pictorially the relationships between static and dynamic load Case Control commands and Bulk Data entries.

Figure 7-2a. Relationship of Dynamic and Static Loads –with `LOADSET` and `LSEQ` Reference.

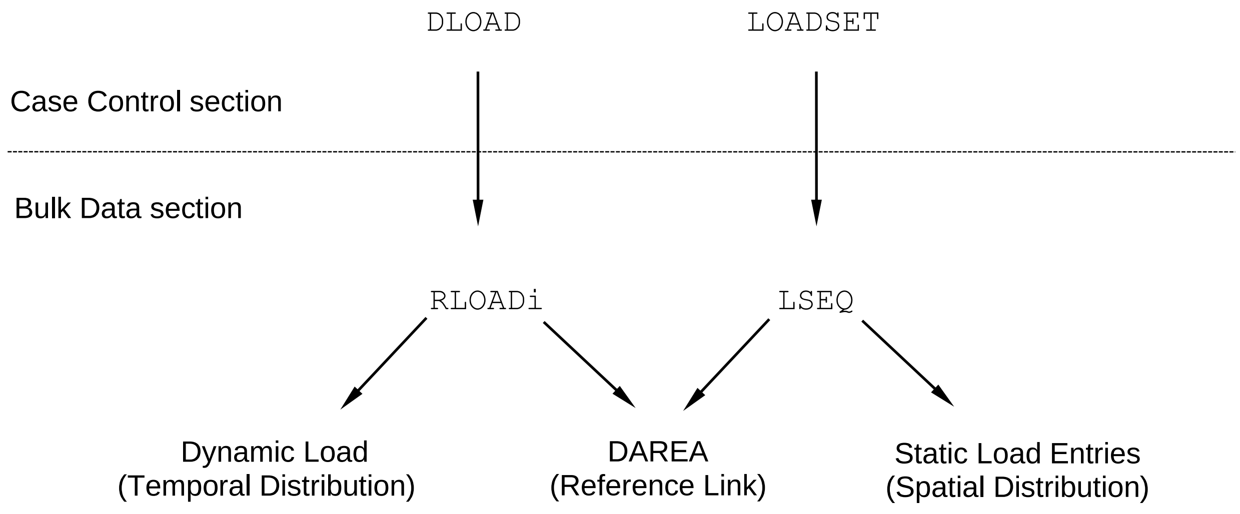
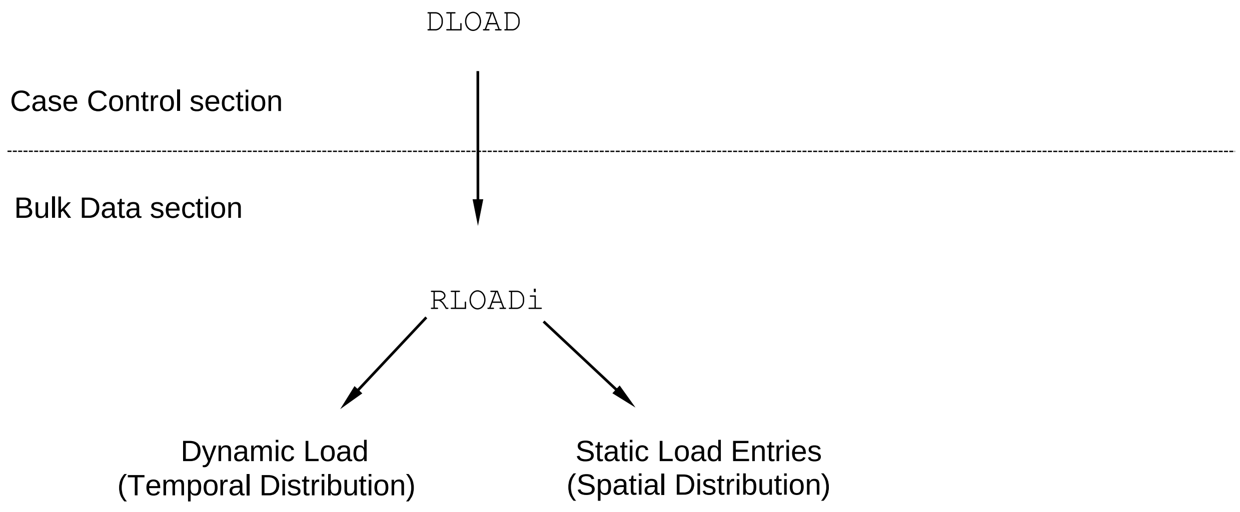


Figure 7-2b. Relationship of Dynamic and Static Loads –without `LOADSET` and `LSEQ` Reference.



7.2.4 Solution Frequencies

Just as the selection of the integration time step is important in transient response analysis, the selection of solution frequencies in frequency response analysis is a major consideration. Unlike the integration time step however, an independent solution is performed at each specified excitation frequency. Still, it is important that the solution frequency step size (Δf) is fine enough to adequately predict peak response. Use at least five points across the half-power bandwidth as shown in Figure 7-3. Note that the half-power bandwidth is approximately $2\zeta f_i$ for a single degree of freedom system.

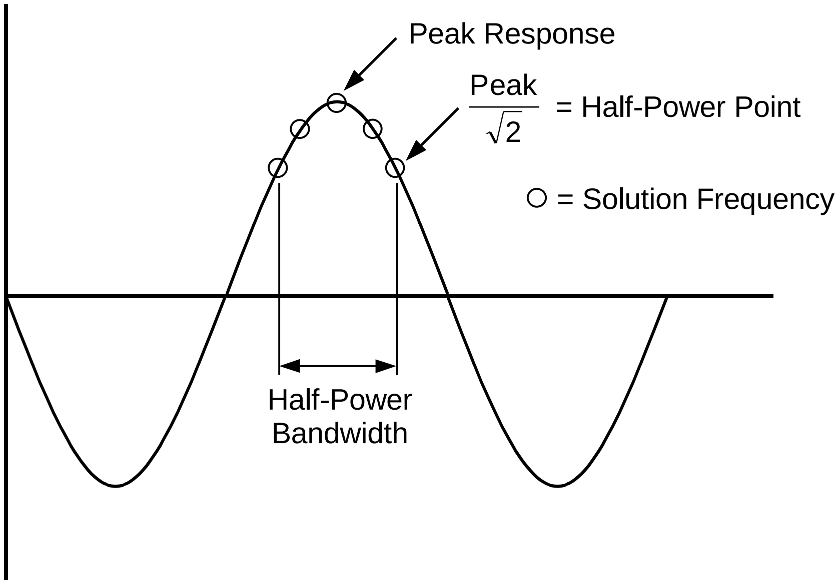


Figure 7-3. Half-Power Point and Bandwidth.

The `FREQ`, `FREQ1`, `FREQ2`, `FREQ3`, and `FREQ4` Bulk Data entries are used to specify the solution frequencies. The `FREQi` entries are selected by the `FREQUENCY` Case Control command. The `FREQ`, `FREQ1`, and `FREQ2` Bulk Data entries define solution frequencies by either specifying them in a discrete list or generating them using parameters. The `FREQ3` and `FREQ4` Bulk Data entries define frequencies around the modal frequencies where the largest response usually occurs. Any combination of `FREQ`, `FREQ1`, `FREQ2`, `FREQ3`, and `FREQ4` Bulk Data entries with the same set identification number are used to generate the solution frequencies. The `DFREQ` parameter specifies the threshold for the elimination of duplicate frequencies.

For maximum efficiency an uneven frequency step is recommended with a more narrow frequency step near resonant frequencies and a wider step away from resonant frequencies.

7.2.5 Dynamic Data Recovery

A frequency response analysis can produce very large amounts of output data since there are usually a large number of solution frequencies involved for a given solution. There are several options available for recovering and storing this data. For data recovery, results can be calculated using one of two methods: mode displacement method and matrix method.

The mode displacement method calculates element results from the global displacement vector in physical coordinates for every solution frequency. The number of operations is proportional to the number of frequencies requested.

The matrix method calculates element results from the global displacement vector produced for each mode shape during eigenvalue extraction. Then the results for each solution frequency are computed as the sum of the modal responses. The number of operations is proportional to the number of modes used.

Since the number of modes is typically much less than the number of time steps, the matrix method is usually more efficient. The `DYNRSLTMETHOD` Model Initialization directive controls these operations. The default for this directive is `AUTO`, which allows the program to choose which method is most efficient based on the number of modes versus time steps. See the *Nastran Solver Reference Guide*, Section 2, *Initialization,* for more information.

For storing and importing results into a post-processor, for example FEMAP®, it is recommended that the Model Initialization directive `RSLTFILETYPE` be set to `FEMAPBINARY`. This will produce a single, binary results neutral file which will contain all results data for each solution frequency.

7.3 Interpreting Results

In this section we will present two examples demonstrating the features and capabilities of frequency response analysis. For both examples we will use the cantilever beam shown in Figure 7-4 with a `MODAL FREQUENCY RESPONSE` solution. For the first problem, it is desired to find the response of the beam to a frequency dependent point load applied at the beam mid-span and free end in the y-direction. The mid-span load has a 45-degree phase lead and end load is scaled to be twice that of the mid-span magnitude. The loads vary with frequency as shown in Figure 7-5. The response is computed over a frequency range from 0 to 100 Hz. Modal damping is used with 5% critical damping across all modes. The eigenvalue extraction is run over a range from 0 to 1000 Hz. The beam supports a nonstructural mass per unit length of 2 pounds per inch. Listing 7-1 contains the Model Input File.

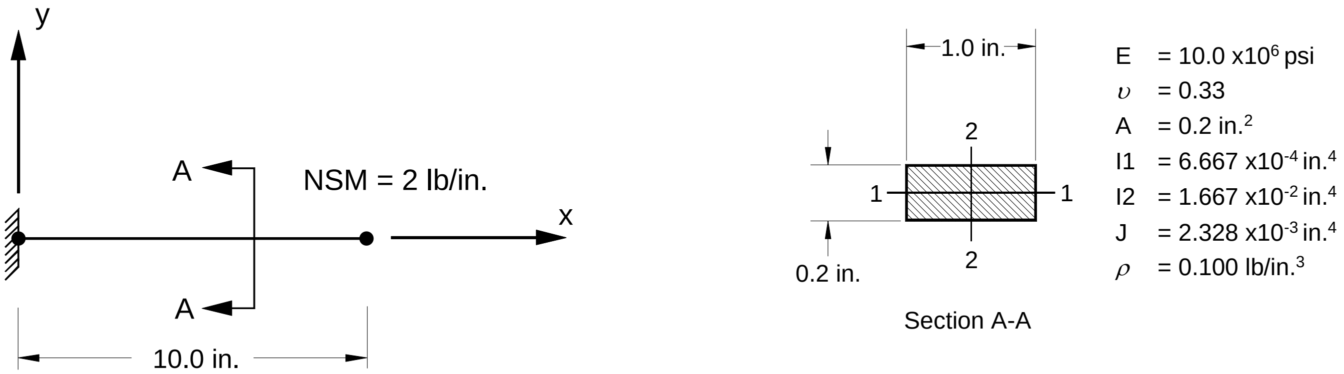


Figure 7-4. 2-D Cantilever Beam Example Problem.

Listing 7-1. Model Input File for the 2-D Cantilever Beam Problem with a Frequency-Dependent Point Load at the Free End and Mid-Span.

```
$
$ FREQUENCY RESPONSE SOLUTION - FORCED RESPONSE.
$
SOL MODAL FREQUENCY RESPONSE
$
SET 1 = 6, 11
DISPLACEMENTS(PHASE) = 1
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = FORCED DYNAMIC RESPONSE
$
SPC = 1
SDAMPING = 20
FREQUENCY = 25
LOADSET = 10
METHOD = 1
$
SUBCASE 1
 LABEL = MID-SPAN AND END LOADS IN Y-DIRECTION
 DLOAD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS (0 TO 1000 Hz).
$
EIGRL,  1,  0.,  1000.
$
$ DEFINE SOLUTION FREQUENCIES (O TO 100 Hz).
$
FREQ1,  25,  0.,  1.,  100
FREQ3,  25,  1.,  100.,  ,  20,  1.0
$
$ DEFINE LOADING.
$
DLOAD,  1,  1.,  1.,  6,  2.,  11
$
$ DEFINE FREQUENCY-DEPENDENT HARMONIC LOADING.
$
TABLED1,  20,
,  0.,  0.,  100.,  1.,  ENDT
RLOAD2,   6,  100,  ,  45,  20
RLOAD2,  11,  200,  ,    ,  20
$
$ 45 DEGREE PHASE LEAD.
$
DPHASE,  45,  6,  2,  45.
```

Listing 7-1. Model Input File for the 2-D Cantilever Beam Problem with a Frequency-Dependent Point Load at the Free End and Mid-Span. (Continued)

```
$
$ 10 LB POINT LOAD IN Y-DIRECTION AT MID-SPAN.
$
FORCE,  1,  6,  0,  10.,  0.,  1.,  0.
LSEQ,  10,  100,  1
$
$ 10 LB POINT LOAD IN Y-DIRECTION AT FREE END.
$
FORCE,  2,  11,  0,  10.,  0.,  1.,  0.
LSEQ,  10,  200,  2
$
$ 5% CRITICAL DAMPING.
$
TABDMP1,  20,  CRIT,
,  0.,  0.05,  1000.,  0.05,  ENDT
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  1.,  0.
CBAR,  2,  10,   2,   3,  0.,  1.,  0.
CBAR,  3,  10,   3,   4,  0.,  1.,  0.
CBAR,  4,  10,   4,   5,  0.,  1.,  0.
CBAR,  5,  10,   5,   6,  0.,  1.,  0.
CBAR,  6,  10,   6,   7,  0.,  1.,  0.
CBAR,  7,  10,   7,   8,  0.,  1.,  0.
CBAR,  8,  10,   8,   9,  0.,  1.,  0.
CBAR,  9,  10,   9,  10,  0.,  1.,  0.
CBAR, 10,  10,  10,  11,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION)
$ WITH 2 LB/IN. NONSTRUCTURAL MASS PER UNIT LENGTH.
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3,  2.,
,  -0.1,  0.5,  0.1,  0.5,  -0.1,  -0.5,  0.1,  -0.5
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ FIXED AT ONE END, MOVEMENT CONSTRAINED TO X-Y PLANE ONLY.
$
SPC1,  1,  123456,  1
SPC1,  1,     345,  1,  THRU,  11
ENDDATA
```

The spatial definition of the dynamic load vector is defined using a static point load (`FORCE`) applied in the y-direction at the free end and mid-span of the beam. This load is then referenced by a load sequence entry (`LSEQ`), which references an area factor (`DAREA`) that serves as the reference link between static and dynamic load definitions. The frequency-dependent dynamic load (`RLOAD2`) then references the area factor defined by the `LSEQ` entry for spatial definition (area) and a `TABLED1` for temporal definition (frequency). The `DLOAD` Bulk Data entry is used to combine and scale dynamic loads defined using the `RLOADi` Bulk Data entries. The `DLOAD` and `LSEQ` Bulk Data entries are called out in the Case Control Section using the `DLOAD` and `LOADSET` Case Control commands, respectively. The resulting frequency dependent loads are shown graphically in Figure 7-5. Note that a `DLOAD` Case Control command can directly call out an `RLOADi` Bulk Data entry, which is not shown in this example.

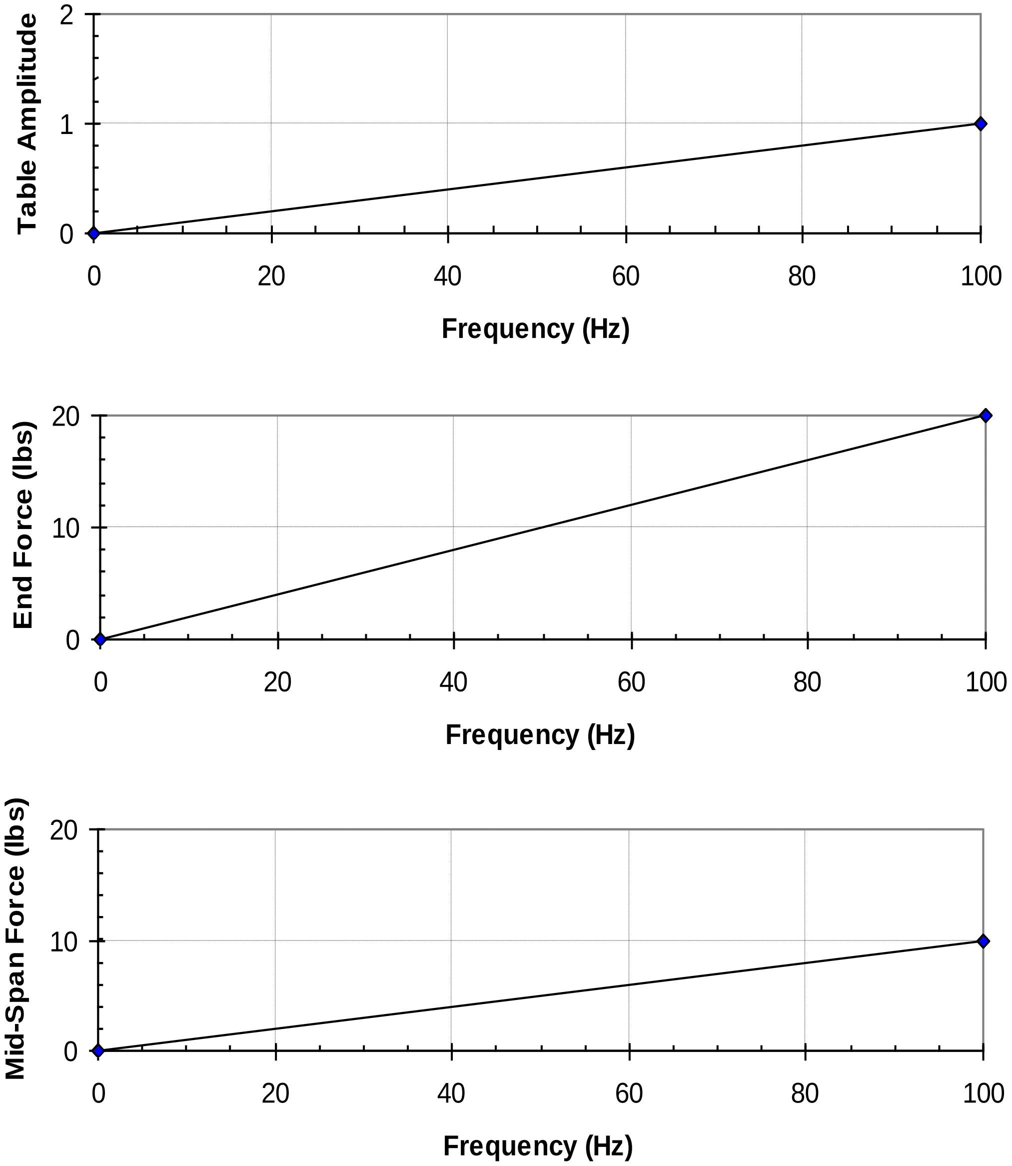


Figure 7-5. Frequency Dependent Loads from the `TABLED1` Entry (top) and Resulting Applied Loads at Beam End (middle) and Mid-Span (bottom).

The `RLOAD2` entry describes a sinusoidal load in the form:

$$P(f) = AB(f)e^{i[\phi(f) + \theta - 2\pi f\tau]}$$

For the example shown in Listing 7-1,

A = 10.0 and comes from the DAREA reference link

B = the function defined on the `TABLED1` entry

ϕ = 0.0 (field 7 on the `RLOAD2` entry is blank)

θ = phase lead of 45 degrees for the point load at grid 6 (entered on the `DPHASE` entry).

τ = 0.0 (field 4 on the `RLOAD2` entry is blank)

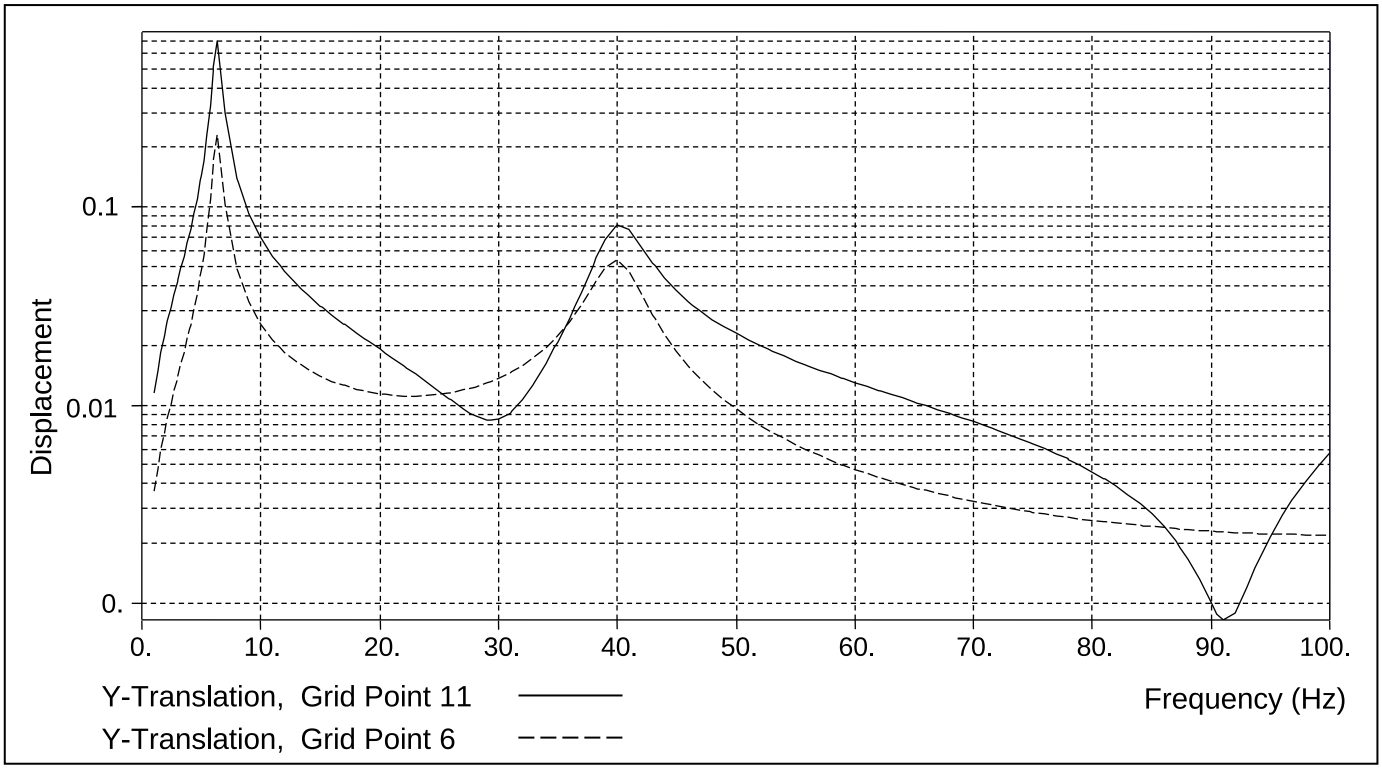
The `FREQ1` and `FREQ3` Bulk Data entries are used to generate the solution frequencies. They are called out in the Case Control Section using the `FREQUENCY` command. In this example the `FREQ1` entry generates frequencies from 0 to 100 Hz with 1 Hz increments. The `FREQ3` entry generates 20 frequencies around each mode between 1 Hz and 100 Hz.

Listing 7-2 gives the extracted resonant frequencies for the beam (0 – 1000 Hz). Figure 7-6 shows the response at the beam free end (grid point 11) and mid-span (grid point 6) plotted logarithmically. As expected, there is a sharp increase in response near the resonant frequencies (6.3 Hz and 40.0 Hz).

Listing 7-2. Extracted Eigenvalues for the 2-D Cantilever Beam.

```
MID-SPAN AND END LOADS IN Y-DIRECTION                               SUBCASE 1

                                                R E A L   E I G E N V A L U E S

    MODE         EIGENVALUE       RADIANS          CYCLES       GENERALIZED     GENERALIZED     ORTHOGONALITY      ERROR
   NUMBER                                                          MASS         STIFFNESS           LOSS         MEASURE
      1         1.574308E+03    3.967755E+01    6.314878E+00    1.000000E+00    1.574308E+03    0.000000E+00    4.257921E-16
      2         6.325622E+04    2.515079E+02    4.002872E+01    1.000000E+00    6.325622E+04    5.506859E-15    3.757854E-16
      3         5.220050E+05    7.224991E+02    1.149893E+02    1.000000E+00    5.220050E+05    3.107161E-15    2.388164E-16
      4         2.172016E+06    1.473776E+03    2.345588E+02    1.000000E+00    2.172016E+06    8.681207E-16    2.093397E-16
      5         6.580798E+06    2.565307E+03    4.082812E+02    1.000000E+00    6.580798E+06    7.743372E-16    2.790956E-15
      6         9.459037E+06    3.075555E+03    4.894897E+02    1.000000E+00    9.459037E+06    1.935053E-21    9.627963E-09
      7         1.651153E+07    4.063438E+03    6.467162E+02    1.000000E+00    1.651153E+07    1.461980E-19    7.108161E-14
      8         3.612442E+07    6.010359E+03    9.565784E+02    1.000000E+00    3.612442E+07    4.060473E-16    1.285729E-12
```

Figure 7-6. Tip and Mid-Span Displacement of the 2-D Cantilever Beam with a Frequency-Dependent Point Load at the Free End and Mid-Span.



7.4 Enforced Motion

Enforced motion specifies the displacement, velocity, and/or acceleration at a set of grid points in frequency response analysis. Enforced motion is used when base excitation is desired and can be combined with externally applied dynamic loading. A good example would be a building subjected to base motion due to an earthquake. In this case, instead of applied loads, the base is subjected to an enforced displacement or acceleration time history.

Autodesk Inventor Nastran uses the large mass method to convert applied forces to enforced motion. The idea is that if a very large mass m_0.is connected to a degree of freedom and a dynamic load p is applied to that same degree of freedom, then the acceleration of the degree of freedom is closely approximated by:

$$\ddot{u} = \frac{1}{M_0} p$$

Which can be re-written in terms of the load that produces the desired acceleration as:

$$p = m_0 \ddot{u}$$

The accuracy of this approximation improves as m_0 becomes larger in comparison to the mass of the structure. A good rule-of-thumb value for m_0 is approximately 10^6 times the mass of the entire structure for an enforced translational degree of freedom and 10^6 times the mass moment of inertia for a rotational degree of freedom.

The following are the basic steps involved in the large mass method:

1. Remove any constraints from the enforced degrees of freedom.
2. Apply large masses m_0 with the `CMASSi` or `CONMi` Bulk Data entries to the degrees of freedom where the motion is enforced. The magnitude for m_0 should be approximately 10^6 times the mass of the entire structure for an enforced translational degree of freedom and 10^6 times the mass moment of inertia for a rotational degree of freedom.
3. Use the `TABLED4` Bulk Data entry to apply scale factors for enforced velocity and displacement.

The `TABLED4` entry uses the algorithm

$$y = \sum_{i=0}^{N} \mathrm{Ai} \left(\frac{x - \mathrm{X1}}{\mathrm{X2}} \right)^{i}$$

where x is input to the table, y is returned. Whenever x < X3, then X3 is used for x and whenever x > X4, X4 is used for x. There are $N + 1$ entries in the table.

For constant acceleration, the force is proportional to the mass for all frequencies. The power series becomes

$$\mathrm{A0} + \mathrm{A1} \left(\frac{x - \mathrm{X1}}{\mathrm{X2}} \right)$$

where,

A0 = 1.0

X1 = 0.0

X2 = 1.0

The above terms define a constant (1.0) in this case.

Constant velocity involves a scale factor that is directly proportional to the circular frequency ($2\pi f$). The power series becomes

$$A0 + A1\left(\frac{x - X1}{X2}\right)$$

where,

A0 = 0.0

A1 = 2π = 6.283185

X1 = 0.0

X2 = 1.0

Note that a phase change of 90 degrees is also required. This change is input using the TD field (field 7) on the `RLOAD1` entry.

Constant displacement involves a scale factor that is directly proportional to the circular frequency squared $(2\pi f)^2$ with a sign change. The power series becomes

$$A0 + A1\left(\frac{x - X1}{X2}\right) + A2\left(\frac{x - X1}{X2}\right)^2$$

where,

A0 = 0.0

A1 = 0.0

A2 = $-(2\pi)^2$ = -39.4784

X1 = 0.0

X2 = 1.0

The following example demonstrates the large mass method for enforced motion. Again, we will use the cantilever beam shown in Figure 7-4 except for the removal of the y-direction constraint at the fixed end. Listing 7-3 contains the Model Input File and Figure 7-7 the response of the beam free end.

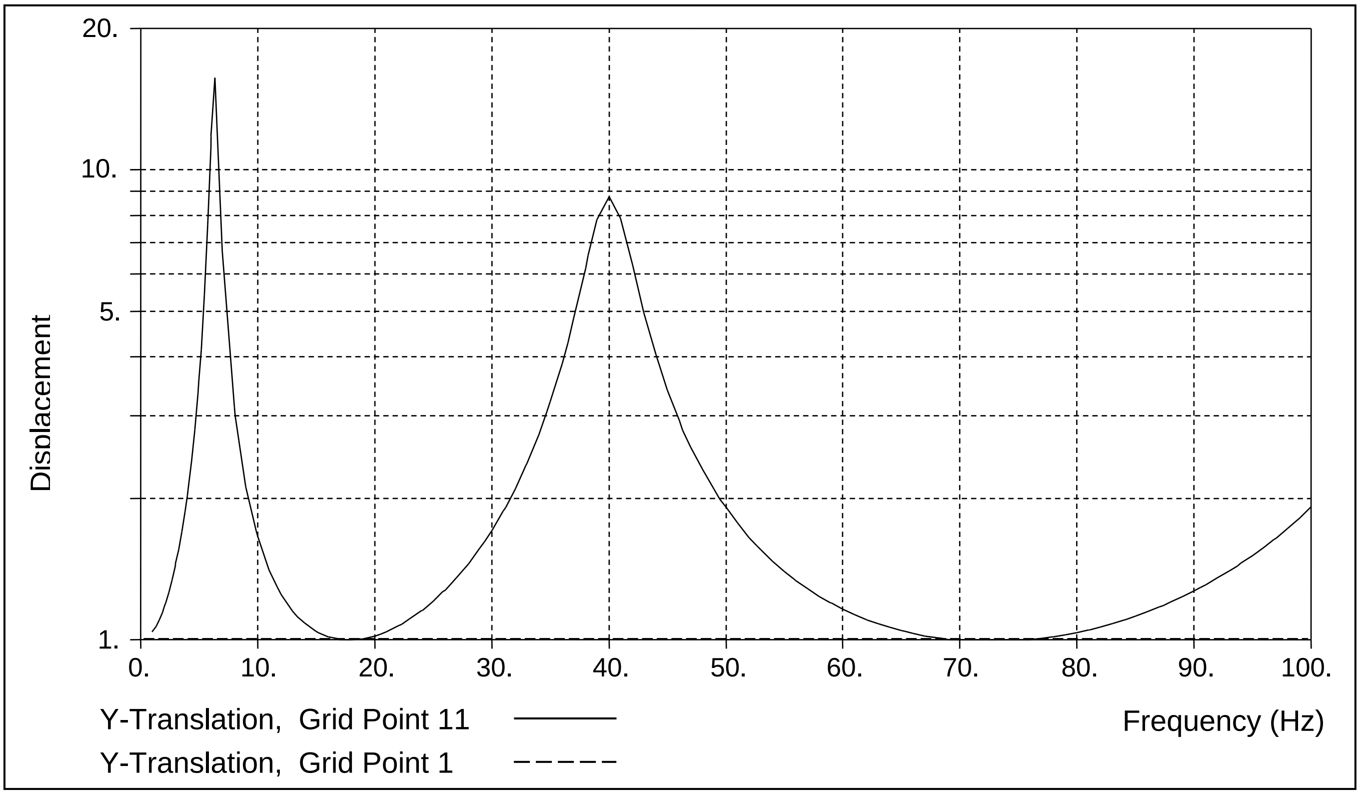
Listing 7-3. Model Input File for the 2-D Cantilever Beam Problem with Enforced Displacement.

```
$
$ FREQUENCY RESPONSE SOLUTION – ENFORCED DISPLACEMENT
$
SOL MODAL FREQUENCY RESPONSE
$
DISPLACEMENTS(PHASE) = ALL
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = FORCED DYNAMIC RESPONSE – ENFORCED DISPLACEMENT
$
SPC = 1
SDAMPING = 20
FREQUENCY = 25
METHOD = 1
$
SUBCASE 1
 LABEL = ENFORCED DISPLACEMENT AT CONSTRAINED END
 DLOAD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS (0 TO 1000 Hz).
$
EIGRL,  1,  0.,  1000.
$
$ DEFINE SOLUTION FREQUENCIES (O TO 100 Hz).
$
FREQ1,  25,  0.,  1.,  100
FREQ3,  25,  1.,  100.,  ,  20,  1.0
$
$ DEFINE LOADING.
$
DLOAD,  1,  2.+5,  0.002588,  11
$
$ DEFINE FREQUENCY-DEPENDENT LOADING.
$
RLOAD1,  11,  100,  ,  ,  10
TABLED4,  10,  0.,  1.,  0.,  100.,
,  0.,  0.,  -39.4784,  ENDT
DAREA,  100,  1,  2,  1.0
$
$ LARGE MASS OF (1xE6)*BAR MASS = (1xE6)*2.0
$
CONM2,  20,  1,  ,  2.+5
```

Listing 7-3. Model Input File for the 2-D Cantilever Beam Problem with Enforced Displacement. (Continued)

```
$
$ 5% CRITICAL DAMPING.
$
TABDMP1,  20,  CRIT,
,  0.,  0.05,  1000.,  0.05,  ENDT
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  1.,  0.
CBAR,  2,  10,   2,   3,  0.,  1.,  0.
CBAR,  3,  10,   3,   4,  0.,  1.,  0.
CBAR,  4,  10,   4,   5,  0.,  1.,  0.
CBAR,  5,  10,   5,   6,  0.,  1.,  0.
CBAR,  6,  10,   6,   7,  0.,  1.,  0.
CBAR,  7,  10,   7,   8,  0.,  1.,  0.
CBAR,  8,  10,   8,   9,  0.,  1.,  0.
CBAR,  9,  10,   9,  10,  0.,  1.,  0.
CBAR, 10,  10,  10,  11,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION)
$ WITH 2 LB/IN. NONSTRUCTURAL MASS PER UNIT LENGTH.
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3,  2.,
,  -0.1,  0.5,  0.1,  0.5,  -0.1,  -0.5,  0.1,  -0.5
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ FIXED AT ONE END EXCEPT IN Y-DIRECTION, MOVEMENT CONSTRAINED TO
$ X-Y PLANE ONLY.
$
SPC1,  1,  12456,  1
SPC1,  1,    345,  1,  THRU,  11
ENDDATA
```

Figure 7-7. **Tip Displacement of a 2-D Cantilever Beam Subjected to a 1.0 Inch Enforced Displacement at the Constrained End.**



8. RANDOM RESPONSE ANALYSIS

8.1 Introduction

In Autodesk Inventor Nastran, random response analysis is performed as a data reduction procedure to the results of a frequency response analysis. The first step is a frequency response analysis with sinusoidal loading input, $\{P_a\}$, as a function of frequency ω_i. Multiple subcases may be defined for various input sources, but all must reference the same sequence of solution frequencies. Data reduction procedures are then applied to the frequency response results producing output quantities $u_{ja}(\omega_i)$, which correspond to an output j and subcase a. Random response results quantities include power spectral densities, root mean square, and autocorrelation functions.

Each loading condition subcase represents a unique random input source and may be applied to multiple grid points or elements. Usually, these loads are chosen to be unit loads with the probabilistic magnitude of each load source defined by spectral density functions on `RANDPS` Bulk Data entries. Correlated load subcases require additional `RANDPS` entries to define the coupling spectral density.

Figure 8-1 is a simplified flow diagram for the Random Response module. The inputs to the module are the frequency responses, $H_{ja}(\omega_i)$, of quantities u_i to loading conditions $\{P_a\}$ at frequencies, ω_i, and the auto- and cross-spectral densities of the loading conditions S_a and S_{ab}. The response quantities, u_j, may be displacements, velocities, accelerations, internal forces, stresses, or strains. The power spectral densities (PSD) of the response quantities are calculated using different methods depending on whether the loading conditions are correlated or uncorrelated. The spectral densities due to all sources considered independent will be combined into one set of outputs.

The application of frequency response techniques to the analysis of random processes requires that the system be linear and that the excitation be stationary with respect to time. The theory includes a few important theorems that will be reviewed.

An important quantity in random analysis theory is the autocorrelation function $R_j(\tau)$, of a physical variable, u_j, which is defined by:

$$R_j(\tau) = \lim_{T \to \infty} \frac{1}{T} \int_0^T u_j(t) u_j(t-\tau) dt$$

Note that $R_j(0)$ is the time average value of u_j^2, which is an important quantity in the analysis of structural failure. The power spectral density, $S_j(\omega)$ of u_j is defined by:

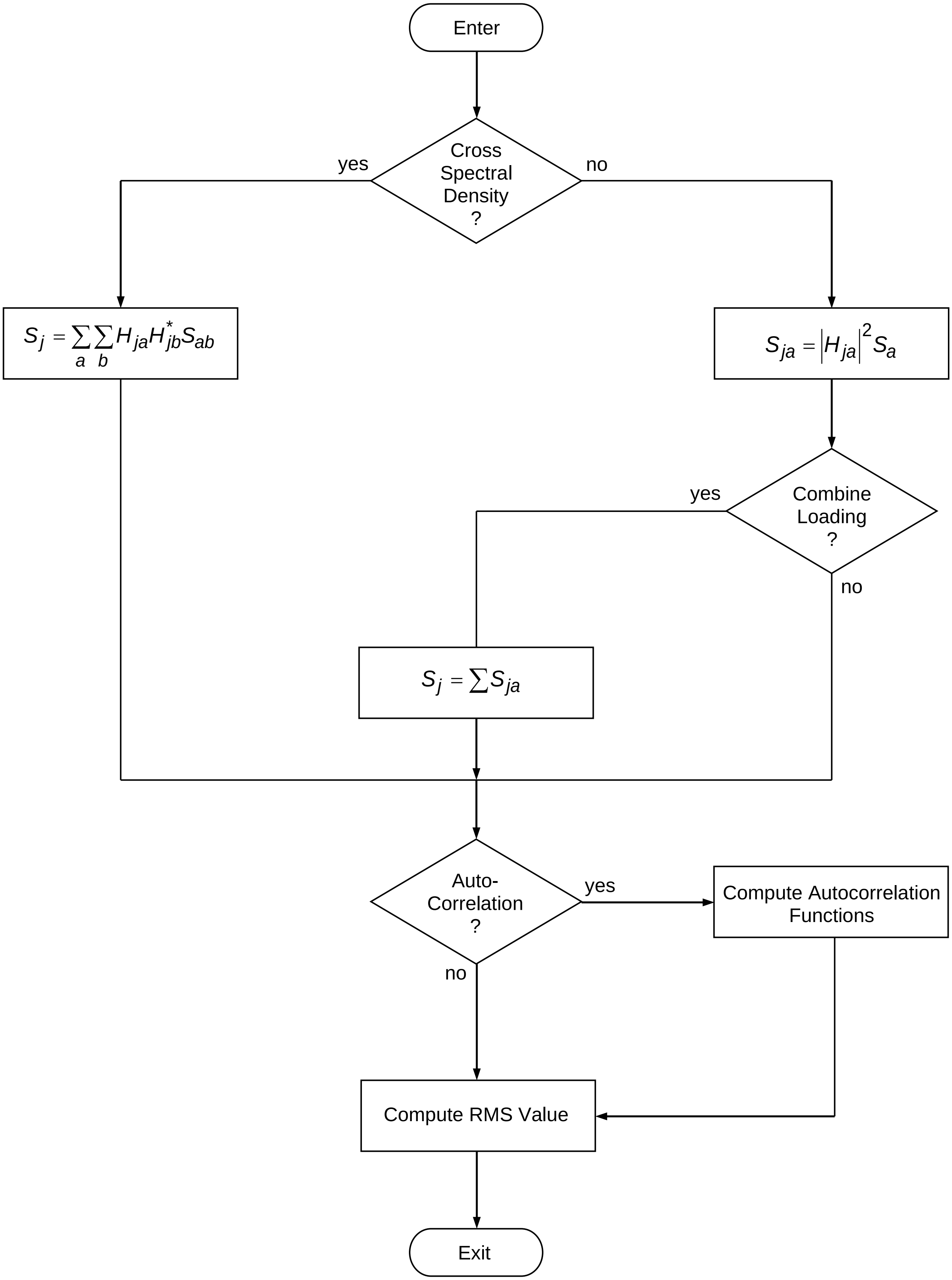
$$S_j(\tau) = \lim_{T \to \infty} \frac{2}{T} \left| \int_0^T e^{-i\omega t} u_j(t) dt \right|^2$$

It may be shown (using the theory of Fourier integrals) that the autocorrelation function and the power spectral density are Fourier transforms of each other. Thus

$$R_j(\tau) = \frac{1}{2\pi} \int_0^\infty S_j(\omega) \cos(\omega t) d\omega$$

from which follows the mean-square theorem,

Figure 8-1. Flow Diagram for Random Response Module.



$$\bar{u}_j^2 = R_j(0) = \frac{1}{2\pi}\int_0^{\infty} S_j(\omega)d\omega$$

The transfer theorem states that if $H_{ja}(\omega)$ is the frequency response of any physical variable, u_j, due to an excitation source, $Q_a(\omega)$, which may be a point force, a loading condition, or some other form of excitation, i.e., if

$$u_j(\omega) = H_{ja}(\omega)Q_a(\omega)$$

where $u_j(\omega)$ and $Q_a(\omega)$ are the Fourier transforms of u_j and Q_a, then the power spectral density of the response $S_j(\omega)$, is related to the power spectral density of the source, $S_a(\omega)$, by:

$$S_j(\omega) = \left|H_{ja}(\omega)\right|^2 S_a(\omega)$$

The above equation is an important result because it allows the statistical properties (e.g., the autocorrelation function) of the response of the system to random excitation to be evaluated via the techniques of frequency response. Another useful result is that, if sources Q_1, Q_2, Q_3, etc., are statistically independent, i.e., if the cross-correlation function between any pair of sources

$$R_{ab}(\tau) = \lim_{T\to\infty}\frac{1}{T}\int_0^{T} Q_a(t)Q_b(t-\tau)d\tau$$

is null, then the power spectral density of the total response is equal to the sum of the power spectral densities of the responses due to individual sources. Thus

$$S_j(\omega) = \sum_a S_{ja}(\omega) = \sum_a \left|H_{ja}(\omega)\right|^2 S_a(\omega)$$

If the sources are statistically correlated, the degree of correlation can be expressed by cross-spectral density, S_{ab}, and the spectral density of the response may be evaluated from

$$S_j = \sum_a\sum_b H_{ja}H_{jb}^{*}S_{ab}$$

where H_{jb}^{*} is the complex conjugate of H_{jb}.

In applying the theory it is not necessary to consider the sources to be forces at individual points. An ensemble of applied forces that are completely correlated (i.e., a loading condition) may also be treated as a source. For example, a plane pressure wave in a specified direction may be treated as a source. Furthermore, the response may be any physical variable including internal forces and stresses as well as displacements, velocities, and accelerations.

8.2 How to Setup a Model Input File for Random Response Analysis

In Autodesk Inventor Nastran, random response analysis is performed as a data reduction procedure to the results of a frequency response solution (`SOL MODAL FREQUENCY RESPONSE`, `SOL DIRECT FREQUENCY RESPONSE`, `SOL PRESTRESS FREQUENCY RESPONSE`, and `SOL NONLINEAR PRESTRESS FREQUENCY RESPONSE`). Therefore, the first step in setting up a random response analysis is the same as for a frequency response analysis. The Random Response module is activated by the specification of the `RANDOM` Case Control command, which references one or more `RANDPS` Bulk Data entries that control the data reduction process depicted in Figure 8-1. Multiple loading condition subcases may be specified, but only one solution or excitation frequency sequence is allowed.

8.3 Interpreting Results

In this section we will present two examples demonstrating the features and capabilities of random response analysis. For both examples we will use the cantilever beam shown in Figure 8-2. For the first problem, it is desired to find the response of the beam to noise excitation displacement at the constrained end in the y-direction. The input PSD is in inch^2/Hz and is plotted in Figure 8-3. The response is computed over a frequency range from 0 to 100 Hz. Modal damping is used with 10% critical damping across all modes. The eigenvalue extraction is run over a range from 0 to 1000 Hz. Listing 8-1 contains the Model Input File.

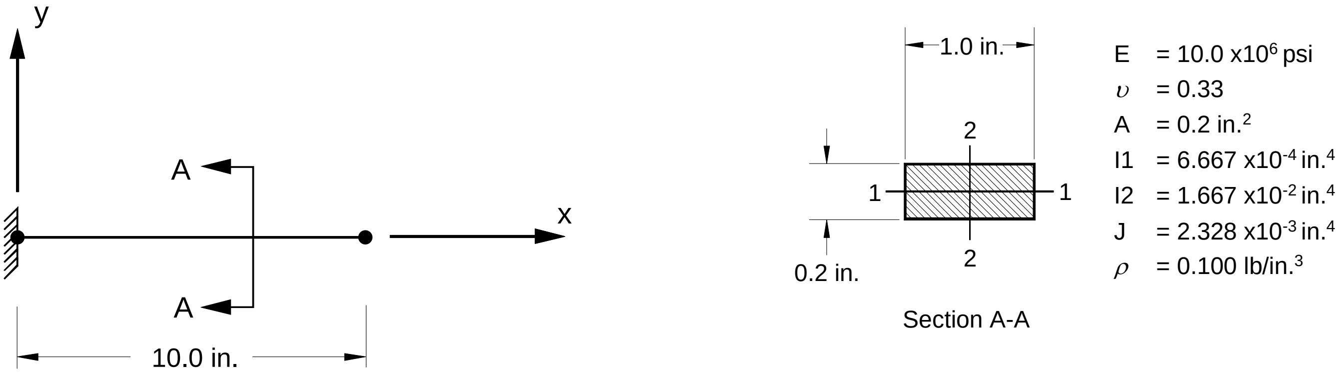


Figure 8-2. 2-D Cantilever Beam Example Problem.

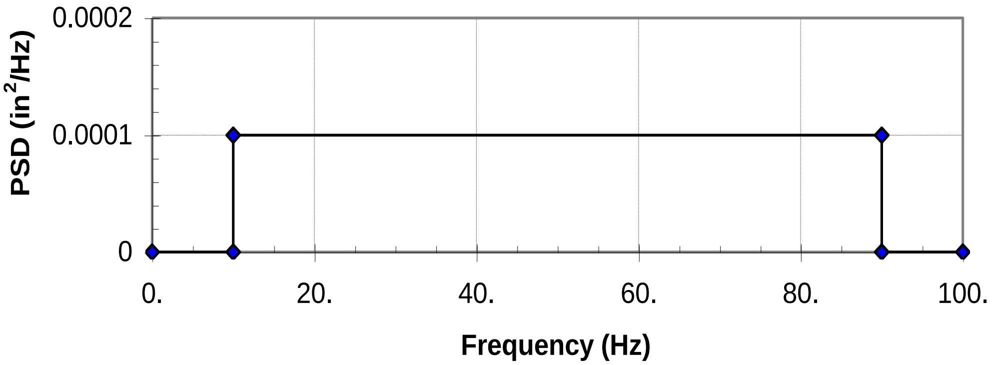


Figure 8-3. Input Power Spectral Density.

Listing 8-1. Model Input File for the 2-D Cantilever Beam Problem with Enforced Motion and PSD Input.

```
$
$ RANDOM RESPONSE SOLUTION - UNCORRELATED SINGLE SOURCE
$
SOL MODAL FREQUENCY RESPONSE
$
DISPLACEMENT(PHASE, PSDF) = ALL
VELOCITY(PHASE, PSDF) = ALL
ACCELERATION(PHASE, PSDF) = ALL
FORCE(PHASE, PSDF) = ALL
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = FORCED DYNAMIC RESPONSE
$
SPC = 1
SDAMPING = 20
FREQUENCY = 25
METHOD = 1
RANDOM = 200
$
SUBCASE 1
 LABEL = ENFORCED DISPLACEMENT AT CONSTRAINED END
 DLOAD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS (0 TO 1000 Hz).
$
EIGRL,  1,  0.,  1000.
$
$ DEFINE SOLUTION FREQUENCIES (O TO 100 Hz).
$
FREQ1,  25,  0.,  1.,  100
FREQ3,  25,  1.,  100.,  ,  20,  1.0
$
$ DEFINE LOADING.
$
DLOAD,  1,  2.+5,  0.002588,  11
$
$ DEFINE FREQUENCY-DEPENDENT LOADING.
$
RLOAD1,  11,  100,  ,  ,  10
TABLED4,  10,  0.,  1.,  0.,  100.,
,  0.,  0.,  -39.4784,  ENDT
DAREA,  100,  1,  2,  1.0
$
$ LARGE MASS OF (1xE6)*BAR MASS = (1xE6)*2.0
$
CONM2,  20,  1,  ,  2.+5
$
$ DEFINE RANDOM ANALYSIS DATA.
$
RANDPS,  200,  1,  1, 1.-4,  0.,  40
TABRND1,  40,
,  0.,  0.,  9.99,  0.,  10.,  1.,  90.,  1.,
,  90.01, 0.,  100.,  0.,  ENDT
```

Listing 8-1. Model Input File for the 2-D Cantilever Beam Problem with Enforced Motion and PSD Input. (Continued)

```
$
$ 10% CRITICAL DAMPING.
$
TABDMP1,  20,  CRIT,
,  0.,  0.10,  1000.,  0.10,  ENDT
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  1.,  0.
CBAR,  2,  10,   2,   3,  0.,  1.,  0.
CBAR,  3,  10,   3,   4,  0.,  1.,  0.
CBAR,  4,  10,   4,   5,  0.,  1.,  0.
CBAR,  5,  10,   5,   6,  0.,  1.,  0.
CBAR,  6,  10,   6,   7,  0.,  1.,  0.
CBAR,  7,  10,   7,   8,  0.,  1.,  0.
CBAR,  8,  10,   8,   9,  0.,  1.,  0.
CBAR,  9,  10,   9,  10,  0.,  1.,  0.
CBAR, 10,  10,  10,  11,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION)
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3,
,  -0.1,  0.5,  0.1,  0.5,  -0.1,  -0.5,  0.1,  -0.5
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ FIXED AT ONE END EXCEPT IN Y-DIRECTION, MOVEMENT CONSTRAINED TO
$ X-Y PLANE ONLY.
$
SPC1,  1,  12456,  1
SPC1,  1,    345,  1,  THRU,  11
ENDDATA
```

In Autodesk Inventor Nastran random response analysis is performed as a data reduction procedure that expands the results of a frequency response analysis. The input to the Random Response module are the output quantities from the frequency response analysis and the specified spectral data for the loading condition (Figure 8-3). Random response results include the results power spectral density functions, the autocorrelation functions, the number of positive crossings (NPX) of the line $y(t) = 0$, and the root mean square (RMS) value of the response. This output is requested using PSDF, ATOC, and RALL options on the result output request Case Control commands.

Figure 8-4 shows the PSD displacement of the beam free end (grid point 11). Table 8-1 gives the RMS values and Table 8-2 compares the number of positive crossings to the first mode frequency of the beam. The RMS value of the response is a measure of the dynamic magnitudes. The RMS value is calculated by taking the square root of the integral of the PSD from the lower frequency to the upper frequency. The number of positive crossings is a measure of the apparent frequency of the response. It is calculated by taking the square root of the second moment of the PSD divided by the RMS value.

Figure 8-4. PSD Tip Displacement of a 2-D Cantilever Beam.

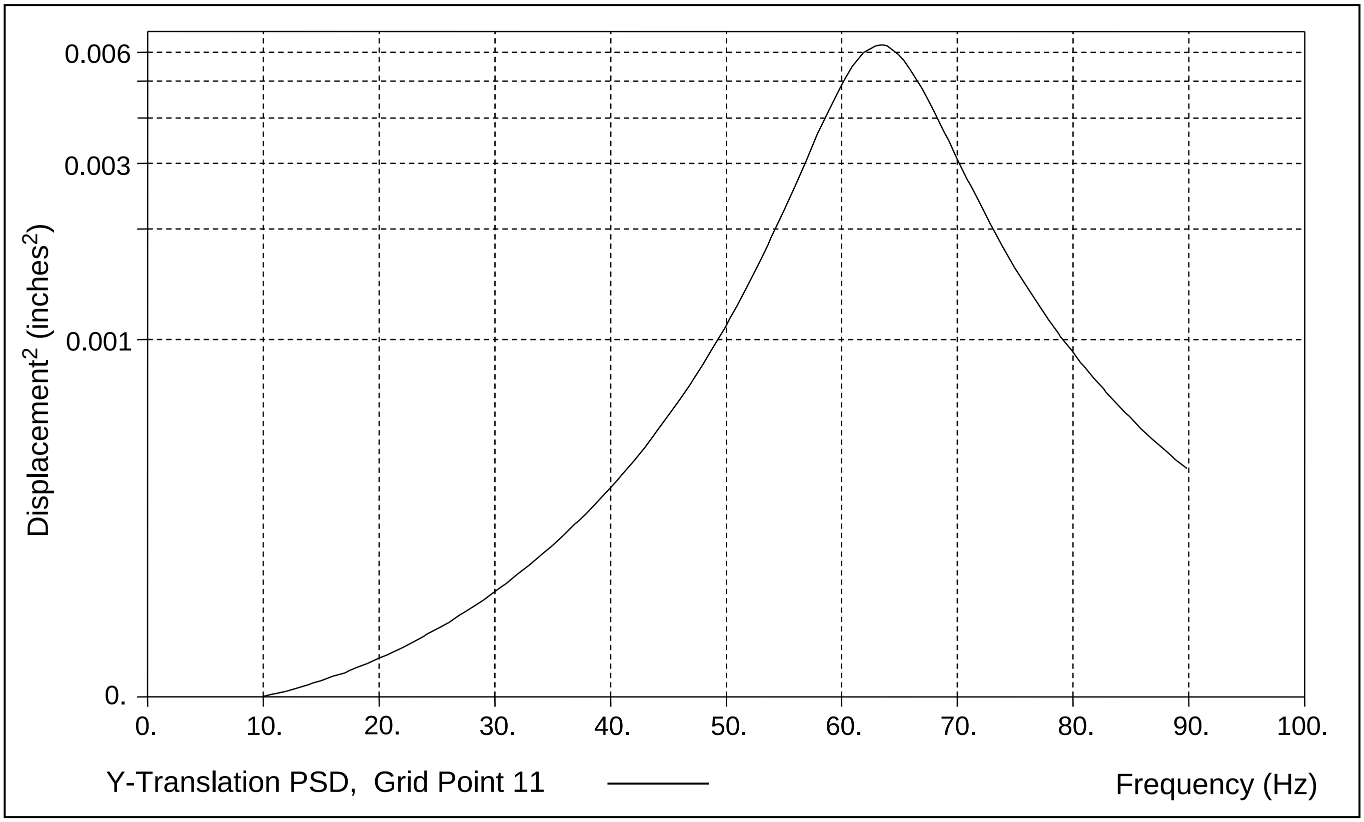


Table 8-1. Response RMS Values of Beam Tip.

Displacement (inches)	Velocity (inches/sec)	Acceleration (inches/sec^2)
0.33457	133.48	56,292

Table 8-2. Number of Positive Zero Crossings.

1st Mode Frequency (Hz)	Tip Displacement (Hz)
63.50	63.49

For the second problem, it is desired to find the response of the beam to both a noise excitation displacement at the constrained end and a noise excitation force at the mid-span. The input PSD for the mid-span load is in pound^2/Hz and is plotted in Figure 8-5. The two loadings are correlated through a cross-spectral density, which is plotted in Figure 8-6. Listing 8-2 contains the Model Input File. Figure 8-7 shows the response at the beam free end. Table 8-3 gives the RMS values and Table 8-4 compares the number of positive crossings to the first mode frequency of the beam.

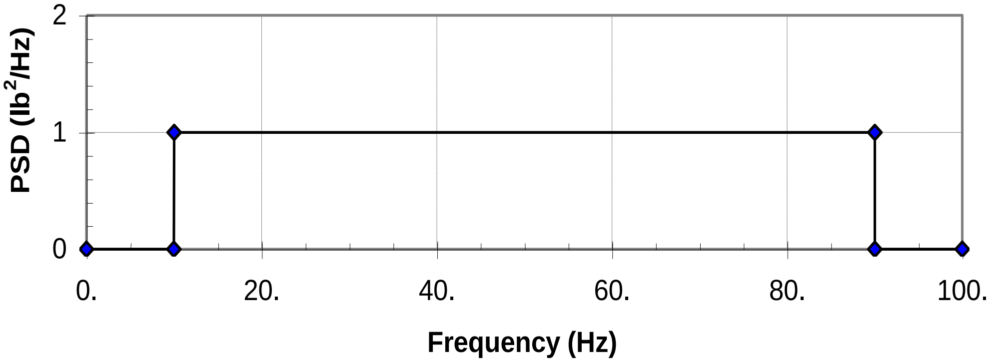


Figure 8-5. Secondary Loading Input Power Spectral Density.

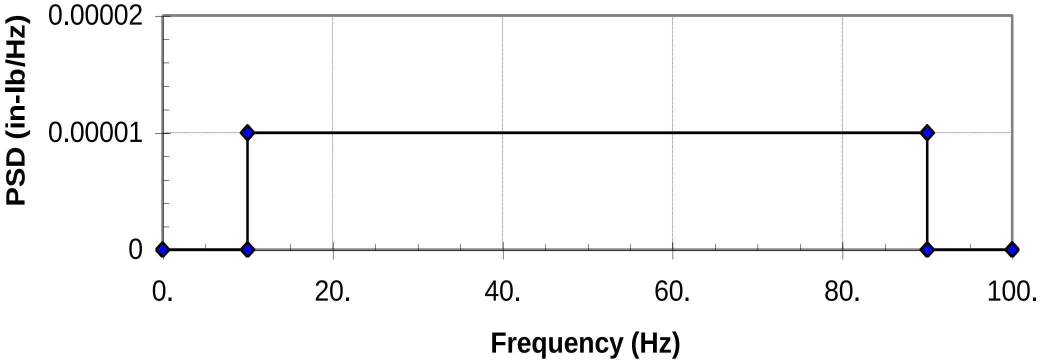


Figure 8-6. Cross-Spectral Density.

Listing 8-2. Model Input File for the Random Response 2-D Cantilever Beam Problem with Combined Loading and PSD Input.

```
$
$ RANDOM RESPONSE SOLUTION - CORRELATED MULTIPLE SOURCE
$
SOL MODAL FREQUENCY RESPONSE
$
DISPLACEMENT(PHASE, PSDF) = ALL
VELOCITY(PHASE, PSDF) = ALL
ACCELERATION(PHASE, PSDF) = ALL
FORCE(PHASE, PSDF) = ALL
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = FORCED DYNAMIC RESPONSE
$
SPC = 1
SDAMPING = 20
FREQUENCY = 25
METHOD = 1
RANDOM = 200
$
SUBCASE 1
 LABEL = ENFORCED DISPLACEMENT AT CONSTRAINED END
 DLOAD = 1
SUBCASE 2
 LABEL = FORCING FUNCTION AT MID-SPAN
 DLOAD = 2
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS (0 TO 1000 Hz).
$
EIGRL,  1,  0.,  1000.
$
$ DEFINE SOLUTION FREQUENCIES (O TO 100 Hz).
$
FREQ1,  25,  0.,  1.,  100
FREQ3,  25,  1.,  100.,  ,  20,  1.0
$
$ DEFINE LOADING -ENFORCED DISPLACEMENT.
$
DLOAD,  1,  2.+5,  0.002588,  11
$
$ DEFINE FREQUENCY-DEPENDENT LOADING.
$
RLOAD1,  11,  100,  ,  ,  10
TABLED4,  10,  0.,  1.,  0.,  100.,
,  0.,  0.,  -39.4784,  ENDT
DAREA,  100,  1,  2,  1.
$
$ LARGE MASS OF (1xE6)*BAR MASS = (1xE6)*2.0
$
CONM2,  20,  1,  ,  2.+5
```

Listing 8-2. Model Input File for the Random Response 2-D Cantilever Beam Problem with Combined Loading and PSD Input. (Continued)

```
$
$ DEFINE LOADING -FORCING FUNCTION.
$
DLOAD,  2,  1.,  1.,  22
$
$ DEFINE FREQUENCY-DEPENDENT LOADING.
$
RLOAD1,  22,  200,  ,  ,  20
TABLED1, 20,
, 0., 0., 100., 1., ENDT
DAREA,  200,  6,  2,  1.
$
$ DEFINE RANDOM ANALYSIS DATA.
$
RANDPS,  200,  1,  1,  1.-4,  0.,  40
RANDPS,  200,  2,  2,  1.,    0.,  40
RANDPS,  200,  1,  2,  1.-5,  0.,  40
TABRND1,  40,
,  0.,  0.,  9.99,  0.,  10.,  1.,  90.,  1.,
,  90.01, 0.,  100.,  0.,  ENDT
$
$ 10% CRITICAL DAMPING.
$
TABDMP1,  20,  CRIT,
,  0.,  0.1,  1000.,  0.1,  ENDT
$
$ INSERT BASIC MODEL (SEE LISTING 8-1).
$
ENDDATA
```

Figure 8-7. PSD Tip Displacement of a 2-D Cantilever Beam with Combined Loading.

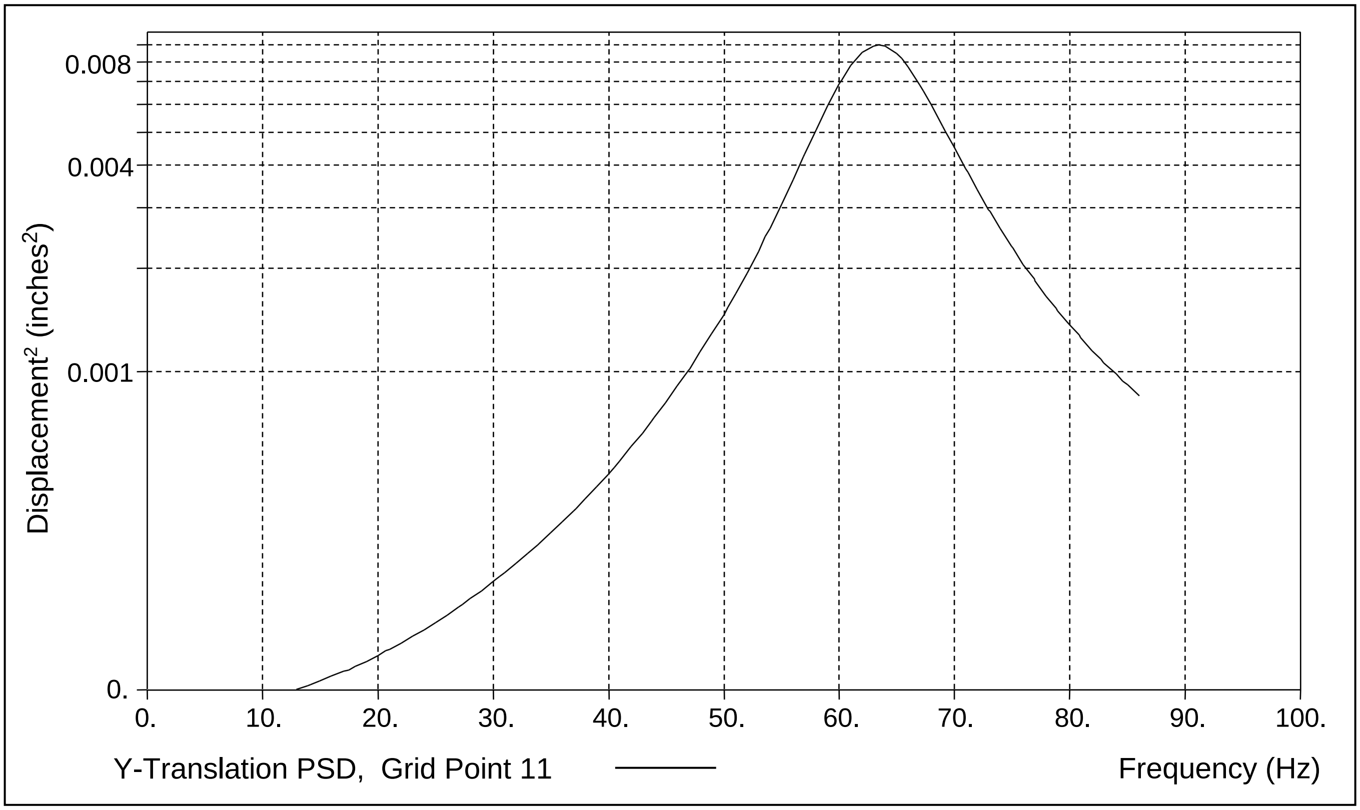


Table 8-3. Response RMS Values of Beam Tip.

Displacement (inches)	Velocity (inches/sec)	Acceleration (inches/sec^2)
0.39898	160.45	67,958

Table 8-4. Number of Positive Zero Crossings.

1st Mode Frequency (Hz)	Tip Displacement (Hz)
63.50	64.00

9. COMPLEX EIGENVALUE ANALYSIS

9.1 Introduction

Complex eigenvalue analysis is required when the global matrixes contain unsymmetric terms or damping effects where real modes analysis cannot be used. It is used for the analysis of aeroelastic flutter, acoustics, rotating bodies, and many other physical effects.

The unforced motion of a system of equations can be expressed as the sum of the motion of its eigenvectors, oscillating and decaying or expanding with terms of the form:

$$u(t) = Real\left(\sum \{\phi_i\} e^{(\alpha_i + i\omega_i)t} \right)$$

The value of α_i gives a measure of the rate of decay or divergence of the i-th natural dynamic mode. If the value of ω_i is nonzero, it gives a measure of the rate of oscillation of the solution. For the most general case, ϕ_i, which represents the shape of the mode, contains complex numbers. The relative size of these numbers indicates which parts of the structure are most active in this mode of motion. The imaginary parts of $\{u\}$ signify phase differences or lag times between degrees of freedom.

The results of complex eigenvalue analysis can be used for such tasks as measuring the effect of damping materials on system performance and determining the stability of a system when it contains sources of energy such as rotating components. The solution is an end result in Autodesk Inventor Nastran. There are no provisions for using the complex shapes for modal response analysis.

The matrix equation used for the complex eigenvalue problem is:

$$\left| [K] + p[B] + p^2[M] \right| [\phi] = 0$$

$$p = \alpha + i\omega$$

where,

$[K]$ is the global linear stiffness matrix

$[B]$ is the global damping matrix

$[M]$ is the global mass matrix

ϕ_i is the complex eigenvector for each mode

and K, B, or M may be symmetric or unsymmetric. As with response solutions, complex eigenvalue analysis can be performed using direct or modal methods. The direct method is not supported currently. The modal method uses reduced matrixes which are generated from the undamped real modes of the problem. If damping is not present ($[B] = 0$) we have

$$\left| [A] + \lambda [I] \right| [\phi] = 0$$

$$[A] = -[M]^{-1}[K]$$

$$\lambda = p^2$$

When damping is present ($[B] \neq 0$) we have

$$\left| [A] - p[I] \right| [\phi] = 0$$

$$[\phi] = \begin{bmatrix} u \\ \hline v \end{bmatrix}$$

$$[A] = \begin{bmatrix} 0 & [I] \\ -[M]^{-1}[K] & -[M]^{-1}[B] \end{bmatrix}$$

For stable systems $\alpha < 0$. The damping coefficient is given by

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

which is approximately twice the value of the conventional modal damping ratio.

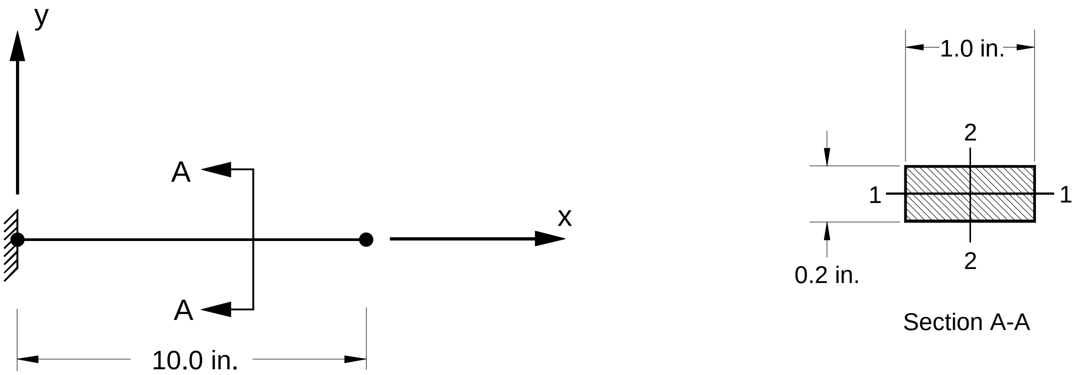
Autodesk Inventor Nastran will also handle complex eigenvalue analysis of structures under initial stress. For more information see Section 14, *Linear Prestress Complex Eigenvalue Analysis*.

9.2 How to Setup a Model Input File for Complex Eigenvalue Analysis

In Autodesk Inventor Nastran you can perform frequency response analysis by setting `SOLUTION = MODAL COMPLEX EIGENVALUE` in the Model Initialization File or by specifying `SOL 110` or `SOL MODAL COMPLEX EIGENVALUE` above the Case Control Section in the Model Input File. Both a `METHOD` and a `CMETHOD` command must be specified in the Case Control referencing an `EIGRL` and `EIGC` entry in the Bulk Data respectively. Only one reference to an `EIGRL` Bulk Data entry (`METHOD` Case Control command) is permitted. This request should be placed above the first subcase. Multiple subcases can be specified, each requesting a different output set.

9.3 Interpreting Results

As an example we will use the cantilever beam shown in Figure 9-1. It is desired to find the first 6 complex modes of the beam with 5% critical damping. Listing 9-1 contains the Model Input File and Listings 9-2 and 9-3 show the extracted frequencies and eigenvectors from the Model Results Output File.



Section A-A

$E = 10.0 \times 10^6$ psi
$\upsilon = 0.33$
$A = 0.2$ in.^{2}
$I1 = 6.667 \times 10^{-4}$ in.^{4}
$I2 = 1.667 \times 10^{-2}$ in.^{4}
$J = 2.328 \times 10^{-3}$ in.^{4}
$\rho = 0.100$ lb/in.^{3}

Figure 9-1. 2-D Cantilever Beam Example Problem.

Listing 9-1. Model Input File for the Damped 2-D Cantilever Beam Problem.

```
$
$ MODAL COMPLEX EIGENVALUE SOLUTION.
$
SOL 110
CEND
$
DISPLACEMENT = ALL
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = DAMPED VIBRATION OF A 2-D CANTILEVER BEAM
$
SUBCASE 1
 SPC = 1
 SDAMPING = 20
 METHOD = 1
 CMETHOD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE REAL EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  6
$
$ DEFINE COMPLEX EIGENVALUE EXTRACTION PARAMETERS.
$
EIGC,  1,  ,  ,  ,  ,  ,  6
$
$ 5% CRITICAL DAMPING.
$
TABDMP1,  20,  CRIT,
,  1.,  0.05,  10000.,  0.05,  ENDT
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  1.,  0.
CBAR,  2,  10,   2,   3,  0.,  1.,  0.
CBAR,  3,  10,   3,   4,  0.,  1.,  0.
CBAR,  4,  10,   4,   5,  0.,  1.,  0.
CBAR,  5,  10,   5,   6,  0.,  1.,  0.
CBAR,  6,  10,   6,   7,  0.,  1.,  0.
CBAR,  7,  10,   7,   8,  0.,  1.,  0.
CBAR,  8,  10,   8,   9,  0.,  1.,  0.
CBAR,  9,  10,   9,  10,  0.,  1.,  0.
CBAR, 10,  10,  10,  11,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3,
,  -0.1,  0.5,  0.1,  0.5,  -0.1,  -0.5,  0.1,  -0.5
```

Listing 9-1. Model Input File for the Damped 2-D Cantilever Beam Problem. (Continued)

```
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ FIXED AT ONE END, MOVEMENT CONSTRAINED TO X-Z PLANE ONLY.
$
SPC1,  1,  123456,  1
SPC1,  1,     345,  1,  THRU,  11
ENDDATA
```

The complex eigenvalue, p, is formed from the real quantities α and ω where $p = \alpha + i\omega$. If $\alpha = 0.$, the radian frequency, ω of complex eigenvalue analysis is the same as that of real eigensolution. The real part, α, is a measure of the decay rate of a damped structure, or if negative, the rate of divergence of an unstable system. The imaginary part, ω, is the modified frequency in radians/unit time. However, roots with negative values of should be treated as special terms. The circular frequency in cycles per unit time is equal to $\omega/2\pi$.

The `EIGC` entry controls the number of modes extracted. Here, we have requested 6 modes as shown in Listings 9-2 and 9-3. The first and second columns contain α and ω respectively. The next column contains the circular frequency followed by the damping coefficient which is approximately twice the value of the conventional modal damping ratio. Note that if the magnitude of this term is computed to be less than 1.0E-12, it is reset to zero. The last column is the error measure and is determined using:

$$\varepsilon_i = \frac{\left| [A]\{\phi\}_i - \lambda_i \{\phi\}_i \right|}{\sqrt{{\alpha_i}^2 + {\omega_i}^2}}$$

The eigenvalues are sorted first on ω and then on increasing magnitude.

Listing 9-2. Extracted Eigenvalues for the Damped 2-D Cantilever Beam.

```
COMPLEX EIGENVALUE                                                      SUBCASE 1

                                                C O M P L E X   E I G E N V A L U E S

     MODE              EIGENVALUE                  CYCLES         DAMPING         ERROR
    NUMBER        (REAL)       (IMAGINARY)                     COEFFICIENT       MEASURE
       1       -1.995058E+01   -3.985126E+02    6.342525E+01    1.001252E-01    1.027507E-09
       2       -1.995058E+01    3.985126E+02    6.342525E+01    1.001252E-01    1.027697E-09
       3       -1.249744E+02   -2.496361E+03    3.973082E+02    1.001252E-01    2.222772E-10
       4       -1.249744E+02    2.496361E+03    3.973082E+02    1.001252E-01    2.222561E-10
       5       -3.497479E+02   -6.986210E+03    1.111890E+03    1.001252E-01    1.019996E-10
       6       -3.497479E+02    6.986210E+03    1.111890E+03    1.001252E-01    1.019839E-10
       7       -6.850966E+02   -1.368479E+04    2.178002E+03    1.001252E-01    8.444388E-11
       8       -6.850965E+02    1.368479E+04    2.178002E+03    1.001252E-01    8.444607E-11
       9       -1.132679E+03   -2.262526E+04    3.600921E+03    1.001252E-01    1.461267E-11
      10       -1.132680E+03    2.262526E+04    3.600921E+03    1.001252E-01    1.461228E-11
      11       -1.545447E+03   -3.087028E+04    4.913158E+03    1.001252E-01    2.866571E-11
      12       -1.545447E+03    3.087028E+04    4.913158E+03    1.001252E-01    2.866561E-11
```

Listing 9-3. Extracted Eigenvectors for the Damped 2-D Cantilever Beam.

```
MODE = 1  COMPLEX EIGENVALUE = -1.995058E+01, -3.985126E+02        SUBCASE 1

                                   C O M P L E X   E I G E N V E C T O R   N U M B E R   1
                                                        (REAL/IMAGINARY)

    GRID      COORDINATE        T1            T2            T3            R1            R2            R3
     ID           ID
       2            0     -2.706882E-14  1.677282E-02  0.000000E+00  0.000000E+00  0.000000E+00  3.273936E-02
                           4.756704E-13 -9.659390E-12  0.000000E+00  0.000000E+00  0.000000E+00 -2.470024E-11
       3            0     -5.347111E-14  6.386868E-02  0.000000E+00  0.000000E+00  0.000000E+00  6.064993E-02
                           9.396283E-13 -5.582915E-11  0.000000E+00  0.000000E+00  0.000000E+00 -6.698559E-11
       4            0     -7.855676E-14  1.364788E-01  0.000000E+00  0.000000E+00  0.000000E+00  8.378142E-02
                           1.380449E-12 -1.320497E-10  0.000000E+00  0.000000E+00  0.000000E+00 -7.569811E-11
       5            0     -1.017081E-13  2.298786E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.022576E-01
                           1.787279E-12 -1.893848E-10  0.000000E+00  0.000000E+00  0.000000E+00 -3.454774E-11
       6            0     -1.223550E-13  3.395162E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.163047E-01
                           2.150100E-12 -2.051540E-10  0.000000E+00  0.000000E+00  0.000000E+00 -3.082215E-12
       7            0     -1.399891E-13  4.611274E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.262747E-01
                           2.459979E-12 -2.028958E-10  0.000000E+00  0.000000E+00  0.000000E+00  9.448827E-12
       8            0     -1.541762E-13  5.908697E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.326623E-01
                           2.709284E-12 -1.747007E-10  0.000000E+00  0.000000E+00  0.000000E+00  5.190285E-11
       9            0     -1.645670E-13  7.254726E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.361176E-01
                           2.891878E-12 -1.038498E-10  0.000000E+00  0.000000E+00  0.000000E+00  7.980615E-11
      10            0     -1.709057E-13  8.623968E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.374542E-01
                           3.003265E-12 -3.671165E-11  0.000000E+00  0.000000E+00  0.000000E+00  4.894805E-11
      11            0     -1.730360E-13  1.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.376534E-01
                           3.040701E-12  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.164341E-11

MODE = 2  COMPLEX EIGENVALUE = -1.995058E+01, 3.985126E+02         SUBCASE 1

                                   C O M P L E X   E I G E N V E C T O R   N U M B E R   2
                                                        (REAL/IMAGINARY)

    GRID      COORDINATE        T1            T2            T3            R1            R2            R3
     ID           ID
       2            0      1.938738E-13  1.677282E-02  0.000000E+00  0.000000E+00  0.000000E+00  3.273936E-02
                          -4.327038E-13 -2.677015E-11  0.000000E+00  0.000000E+00  0.000000E+00 -4.628953E-11
       3            0      3.829738E-13  6.386868E-02  0.000000E+00  0.000000E+00  0.000000E+00  6.064993E-02
                          -8.547531E-13 -7.346420E-11  0.000000E+00  0.000000E+00  0.000000E+00 -3.659553E-11
       4            0      5.626437E-13  1.364788E-01  0.000000E+00  0.000000E+00  0.000000E+00  8.378142E-02
                          -1.255755E-12 -8.921962E-11  0.000000E+00  0.000000E+00  0.000000E+00  2.056610E-12
       5            0      7.284595E-13  2.298786E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.022576E-01
                          -1.625837E-12 -8.712483E-11  0.000000E+00  0.000000E+00  0.000000E+00 -6.143463E-12
       6            0      8.763381E-13  3.395162E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.163047E-01
                          -1.955885E-12 -9.951562E-11  0.000000E+00  0.000000E+00  0.000000E+00 -8.461580E-12
       7            0      1.002638E-12  4.611274E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.262747E-01
                          -2.237772E-12 -8.491194E-11  0.000000E+00  0.000000E+00  0.000000E+00  3.875474E-11
       8            0      1.104250E-12  5.908697E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.326623E-01
                          -2.464559E-12 -4.203039E-11  0.000000E+00  0.000000E+00  0.000000E+00  3.221472E-11
       9            0      1.178672E-12  7.254726E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.361176E-01
                          -2.630659E-12 -3.294796E-11  0.000000E+00  0.000000E+00  0.000000E+00 -7.585660E-12
      10            0      1.224071E-12  8.623968E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.374542E-01
                          -2.731984E-12 -3.229427E-11  0.000000E+00  0.000000E+00  0.000000E+00  1.807628E-11
      11            0      1.239329E-12  1.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.376534E-01
                          -2.766039E-12  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.837291E-11

MODE = 3  COMPLEX EIGENVALUE = -1.249744E+02, -2.496361E+03        SUBCASE 1

                                   C O M P L E X   E I G E N V E C T O R   N U M B E R   3
                                                        (REAL/IMAGINARY)

    GRID      COORDINATE        T1            T2            T3            R1            R2            R3
     ID           ID
       2            0     -9.863026E-12 -9.262050E-02  0.000000E+00  0.000000E+00  0.000000E+00 -1.677458E-01
                          -1.192465E-11 -5.830062E-10  0.000000E+00  0.000000E+00  0.000000E+00 -1.039212E-09
       3            0     -1.948319E-11 -3.010453E-01  0.000000E+00  0.000000E+00  0.000000E+00 -2.324165E-01
                          -2.355568E-11 -1.914746E-09  0.000000E+00  0.000000E+00  0.000000E+00 -1.590792E-09
       4            0     -2.862362E-11 -5.261513E-01  0.000000E+00  0.000000E+00  0.000000E+00 -2.035496E-01
                          -3.460669E-11 -3.692362E-09  0.000000E+00  0.000000E+00  0.000000E+00 -1.876463E-09
       5            0     -3.705923E-11 -6.835468E-01  0.000000E+00  0.000000E+00  0.000000E+00 -1.012129E-01
                          -4.480557E-11 -5.335255E-09  0.000000E+00  0.000000E+00  0.000000E+00 -1.201914E-09
       6            0     -4.458232E-11 -7.138195E-01  0.000000E+00  0.000000E+00  0.000000E+00  4.523659E-02
                          -5.390119E-11 -5.815188E-09  0.000000E+00  0.000000E+00  0.000000E+00  2.536147E-10
       7            0     -5.100765E-11 -5.896992E-01  0.000000E+00  0.000000E+00  0.000000E+00  2.018830E-01
                          -6.166958E-11 -5.075868E-09  0.000000E+00  0.000000E+00  0.000000E+00  1.023977E-09
       8            0     -5.617701E-11 -3.173098E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.370850E-01
                          -6.791946E-11 -4.078385E-09  0.000000E+00  0.000000E+00  0.000000E+00  9.192811E-10
       9            0     -5.996309E-11  6.980181E-02  0.000000E+00  0.000000E+00  0.000000E+00  4.288217E-01
                          -7.249694E-11 -3.130558E-09  0.000000E+00  0.000000E+00  0.000000E+00  1.084760E-09
      10            0     -6.227269E-11  5.236094E-01  0.000000E+00  0.000000E+00  0.000000E+00  4.710752E-01
                          -7.528930E-11 -1.767885E-09  0.000000E+00  0.000000E+00  0.000000E+00  1.634734E-09
      11            0     -6.304893E-11  1.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  4.782311E-01
                          -7.622779E-11  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.819634E-09
```

Listing 9-3. Extracted Eigenvectors for the Damped 2-D Cantilever Beam. (Continued)

```
MODE = 4  COMPLEX EIGENVALUE = -1.249744E+02, 2.496361E+03          SUBCASE 1

                                   C O M P L E X   E I G E N V E C T O R   N U M B E R   4
                                                        (REAL/IMAGINARY)

    GRID      COORDINATE        T1            T2            T3            R1            R2            R3
     ID           ID
       2            0     -1.109426E-11 -9.262050E-02  0.000000E+00  0.000000E+00  0.000000E+00 -1.677458E-01
                           1.074754E-11 -6.435569E-10  0.000000E+00  0.000000E+00  0.000000E+00 -1.333895E-09
       3            0     -2.191535E-11 -3.010453E-01  0.000000E+00  0.000000E+00  0.000000E+00 -2.324165E-01
                           2.123044E-11 -2.595778E-09  0.000000E+00  0.000000E+00  0.000000E+00 -2.418737E-09
       4            0     -3.219681E-11 -5.261513E-01  0.000000E+00  0.000000E+00  0.000000E+00 -2.035496E-01
                           3.119057E-11 -4.997701E-09  0.000000E+00  0.000000E+00  0.000000E+00 -2.118104E-09
       5            0     -4.168547E-11 -6.835468E-01  0.000000E+00  0.000000E+00  0.000000E+00 -1.012129E-01
                           4.038269E-11 -6.463261E-09  0.000000E+00  0.000000E+00  0.000000E+00 -7.517669E-10
       6            0     -5.014770E-11 -7.138195E-01  0.000000E+00  0.000000E+00  0.000000E+00  4.523659E-02
                           4.858045E-11 -6.586383E-09  0.000000E+00  0.000000E+00  0.000000E+00  4.112654E-10
       7            0     -5.737513E-11 -5.896992E-01  0.000000E+00  0.000000E+00  0.000000E+00  2.018830E-01
                           5.558200E-11 -5.782551E-09  0.000000E+00  0.000000E+00  0.000000E+00  1.166343E-09
       8            0     -6.318979E-11 -3.173098E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.370850E-01
                           6.121494E-11 -4.296511E-09  0.000000E+00  0.000000E+00  0.000000E+00  1.754818E-09
       9            0     -6.744851E-11  6.980181E-02  0.000000E+00  0.000000E+00  0.000000E+00  4.288217E-01
                           6.534057E-11 -2.483817E-09  0.000000E+00  0.000000E+00  0.000000E+00  1.733065E-09
      10            0     -7.004643E-11  5.236094E-01  0.000000E+00  0.000000E+00  0.000000E+00  4.710752E-01
                           6.785729E-11 -1.023456E-09  0.000000E+00  0.000000E+00  0.000000E+00  1.174570E-09
      11            0     -7.091956E-11  1.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  4.782311E-01
                           6.870314E-11  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  9.636255E-10

MODE = 5  COMPLEX EIGENVALUE = -3.497479E+02, -6.986210E+03         SUBCASE 1

                                   C O M P L E X   E I G E N V E C T O R   N U M B E R   5
                                                        (REAL/IMAGINARY)

    GRID      COORDINATE        T1            T2            T3            R1            R2            R3
     ID           ID
       2            0     -9.346263E-12  2.280767E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.766304E-01
                          -2.415375E-12  8.075124E-09  0.000000E+00  0.000000E+00  0.000000E+00  1.317785E-08
       3            0     -1.846239E-11  6.047084E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.121163E-01
                          -4.771275E-12  2.069064E-08  0.000000E+00  0.000000E+00  0.000000E+00  9.667908E-09
       4            0     -2.712391E-11  7.568022E-01  0.000000E+00  0.000000E+00  0.000000E+00 -3.513278E-02
                          -7.009691E-12  2.483371E-08  0.000000E+00  0.000000E+00  0.000000E+00 -1.307111E-09
       5            0     -3.511755E-11  5.267839E-01  0.000000E+00  0.000000E+00  0.000000E+00 -4.058157E-01
                          -9.075504E-12  2.033773E-08  0.000000E+00  0.000000E+00  0.000000E+00 -6.141994E-09
       6            0     -4.224648E-11  2.053366E-02  0.000000E+00  0.000000E+00  0.000000E+00 -5.554151E-01
                          -1.091785E-11  1.450911E-08  0.000000E+00  0.000000E+00  0.000000E+00 -5.340223E-09
       7            0     -4.833516E-11 -4.733227E-01  0.000000E+00  0.000000E+00  0.000000E+00 -3.796808E-01
                          -1.249136E-11  9.258382E-09  0.000000E+00  0.000000E+00  0.000000E+00 -5.225090E-09
       8            0     -5.323367E-11 -6.575998E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.507367E-02
                          -1.375729E-11  4.857943E-09  0.000000E+00  0.000000E+00  0.000000E+00 -2.970407E-09
       9            0     -5.682139E-11 -3.955047E-01  0.000000E+00  0.000000E+00  0.000000E+00  4.732967E-01
                          -1.468447E-11  3.571737E-09  0.000000E+00  0.000000E+00  0.000000E+00 -1.689824E-10
      10            0     -5.900998E-11  2.279262E-01  0.000000E+00  0.000000E+00  0.000000E+00  7.341997E-01
                          -1.525008E-11  2.824088E-09  0.000000E+00  0.000000E+00  0.000000E+00 -1.904478E-09
      11            0     -5.974554E-11  1.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  7.855798E-01
                          -1.544017E-11  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00 -3.215544E-09

MODE = 6  COMPLEX EIGENVALUE = -3.497479E+02, 6.986210E+03          SUBCASE 1

                                   C O M P L E X   E I G E N V E C T O R   N U M B E R   6
                                                        (REAL/IMAGINARY)

    GRID      COORDINATE        T1            T2            T3            R1            R2            R3
     ID           ID
       2            0     -8.842369E-12  2.280767E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.766304E-01
                           3.943561E-12  1.110826E-08  0.000000E+00  0.000000E+00  0.000000E+00  1.477227E-08
       3            0     -1.746701E-11  6.047084E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.121163E-01
                           7.790018E-12  2.100222E-08  0.000000E+00  0.000000E+00  0.000000E+00  4.699155E-09
       4            0     -2.566155E-11  7.568022E-01  0.000000E+00  0.000000E+00  0.000000E+00 -3.513278E-02
                           1.144466E-11  2.392219E-08  0.000000E+00  0.000000E+00  0.000000E+00  2.518356E-09
       5            0     -3.322422E-11  5.267839E-01  0.000000E+00  0.000000E+00  0.000000E+00 -4.058157E-01
                           1.481749E-11  2.438510E-08  0.000000E+00  0.000000E+00  0.000000E+00 -3.808008E-09
       6            0     -3.996880E-11  2.053366E-02  0.000000E+00  0.000000E+00  0.000000E+00 -5.554151E-01
                           1.782547E-11  1.483056E-08  0.000000E+00  0.000000E+00  0.000000E+00 -1.337957E-08
       7            0     -4.572922E-11 -4.733227E-01  0.000000E+00  0.000000E+00  0.000000E+00 -3.796808E-01
                           2.039453E-11  4.404116E-09  0.000000E+00  0.000000E+00  0.000000E+00 -4.744435E-09
       8            0     -5.036363E-11 -6.575998E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.507367E-02
                           2.246140E-11  5.037579E-09  0.000000E+00  0.000000E+00  0.000000E+00  3.336627E-09
       9            0     -5.375792E-11 -3.955047E-01  0.000000E+00  0.000000E+00  0.000000E+00  4.732967E-01
                           2.397521E-11  5.945119E-09  0.000000E+00  0.000000E+00  0.000000E+00 -2.366942E-09
      10            0     -5.582852E-11  2.279262E-01  0.000000E+00  0.000000E+00  0.000000E+00  7.341997E-01
                           2.489866E-11  2.279091E-09  0.000000E+00  0.000000E+00  0.000000E+00 -3.407295E-09
      11            0     -5.652443E-11  1.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  7.855798E-01
                           2.520902E-11  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00 -1.738740E-09
```

10. LINEAR PRESTRESS STATIC ANALYSIS

10.1 Introduction

In Autodesk Inventor Nastran you can simulate how a structure’s modes, displacements, stresses, etc. are affected by a pre-stiffened or prestressed structure. One example would be the out of plane deflection of a flat plate in an initial state of biaxial stress. Prestress tensile stresses would stiffen the plate in flexure and thus decrease the out of plane deflection resulting from a transverse load. The opposite is true for prestress compressive stresses.

In prestress analysis, the global stiffness matrix is augmented by the initial stress or differential stiffness matrix as shown below:

$$\left| [K_l] + [K_S] \right| \{D\} = \{R\}$$

where,

$[K_l]$	is the global linear stiffness matrix
$[K_S]$	is the global initial stress stiffness matrix
$\{D\}$	is the global displacement vector
$\{R\}$	is the global load vector

10.2 How to Setup a Model Input File for Linear Prestress Static Analysis

In Autodesk Inventor Nastran you can solve a linear prestress static problem by setting `SOLUTION = LINEAR PRESTRESS STATIC` in the Model Initialization File or by specifying `SOL 181` or `SOL LINEAR PRESTRESS STATIC` above the Case Control Section in the Model Input File, and following the procedure listed below:

1. Apply prestress static loads to the first subcase. These loads will generate internal loads that are used to formulate the initial stress or differential stiffness matrix.
2. Additional loading is referenced in the second through n subcases.
3. The initial stress stiffness matrix is automatically generated for each element that supports differential stiffness. Elements that support differential stiffness are: `CROD`, `CBAR`, `CBEAM`, `CQUAD4`, `CQUADR`, `CTRIA3`, `CTRIAR`, `CHEXA`, `CPENTA`, `CPYRA`, and `CTETRA`.
4. Each subcase may have a different boundary condition; however, the global differential stiffness matrix will be based on the boundary conditions specified in the first subcase.
5. The parameter `ADDPRESTRESS` is used to control if the stress from the first subcase (initial stress state) is added to subsequent subcases. The default setting for `ADDPRESTRESS` is `ON`.

10.3 Interpreting Results

As an example we will use the cantilever beam shown in Figure 10-1. The 50 pound compressive load defines the initial stress state. It is desired to find deflection and rotation at the free end resulting from a 5 pound shear load. Listing 10-1 contains the Model Input File and Listing 10-2 shows the predicted displacements from the Model Results Output File. The deflected shape is plotted in Figure 10-2.

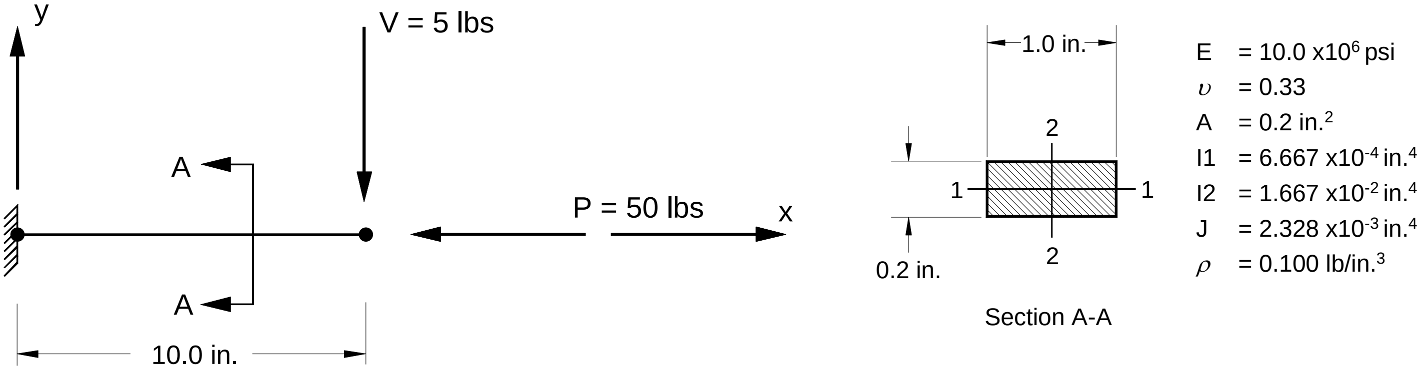


Figure 10-1. Prestress Beam Example Problem.

Listing 10-1. Model Input File for the Prestress Beam Problem.

```
$
$ LINEAR PRESTRESS STATIC SOLUTION.
$
SOL LINEAR PRESTRESS STATIC
$
$ OPTION FOR ADDING PRESTRESS SUBCASE 1 RESULTS TO FOLLOWING SUBCASES.
$
PARAM, ADDPRESTRESS, ON
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = 2-D CANTILEVER BEAM WITH A PRESTRESS COMPRESSIVE LOAD
$
DISPLACEMENT = ALL
STRESS = ALL
$
SUBCASE 1
 LABEL = PRESTRESS COMPRESSIVE LOAD (AXIAL).
 SPC = 1
 LOAD = 1
SUBCASE 2
 LABEL = POINT LOAD AT FREE END (SHEAR).
 SPC = 1
 LOAD = 2
$
BEGIN BULK
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  1.,  0.
CBAR,  2,  10,   2,   3,  0.,  1.,  0.
CBAR,  3,  10,   3,   4,  0.,  1.,  0.
CBAR,  4,  10,   4,   5,  0.,  1.,  0.
CBAR,  5,  10,   5,   6,  0.,  1.,  0.
CBAR,  6,  10,   6,   7,  0.,  1.,  0.
CBAR,  7,  10,   7,   8,  0.,  1.,  0.
CBAR,  8,  10,   8,   9,  0.,  1.,  0.
CBAR,  9,  10,   9,  10,  0.,  1.,  0.
CBAR, 10,  10,  10,  11,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3,
,  -0.1,  0.5,  0.1,  0.5,  -0.1,  -0.5,  0.1,  -0.5
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ FIXED AT BOTH ENDS, MOVEMENT CONSTRAINED TO X-Y PLANE ONLY.
$
SPC1,  1,  123456,  1
SPC1,  1,     345,  1,  THRU,  11
```

Listing 10-1. Model Input File for the Prestress Beam Problem. (Continued)

```
$
$ PRESTRESS COMPRESSIVE LOAD (AXIAL).
$
FORCE,  1,  11,  0, -50.,  1.,  0.,  0.
$
$ POINT LOAD AT FREE END (SHEAR).
$
FORCE,  2,  11,  0,  -5.,  0.,  1.,  0.
ENDDATA
```

Listing 10-2. Displacement Vector for the Prestress Beam Problem.

```
  PRESTRESS COMPRESSIVE LOAD (AXIAL)                                  SUBCASE 1

                                     D I S P L A C E M E N T   V E C T O R

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID          ID
       2           0     -0.250000E-04  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       3           0     -0.500000E-04  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       4           0     -0.750000E-04  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       5           0     -0.100000E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       6           0     -0.125000E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       7           0     -0.150000E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       8           0     -0.175000E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       9           0     -0.200000E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      10           0     -0.225000E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      11           0     -0.250000E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00


 EPSILON       =  0.397904E-15
 STRAIN ENERGY =  0.625000E-02

 MAXIMUM DISPLACEMENT MAGNITUDE =  0.250000E-03  AT GRID 11
 MAXIMUM ROTATION MAGNITUDE     =  0.000000E+00  AT GRID 11



  POINT LOAD AT FREE END (SHEAR)                                      SUBCASE 2

                                     D I S P L A C E M E N T   V E C T O R

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID          ID
       2           0      0.000000E+00 -0.496293E-02  0.000000E+00  0.000000E+00  0.000000E+00 -0.979465E-02
       3           0      0.000000E+00 -0.193149E-01  0.000000E+00  0.000000E+00  0.000000E+00 -0.187664E-01
       4           0      0.000000E+00 -0.421989E-01  0.000000E+00  0.000000E+00  0.000000E+00 -0.268480E-01
       5           0      0.000000E+00 -0.726939E-01  0.000000E+00  0.000000E+00  0.000000E+00 -0.339789E-01
       6           0      0.000000E+00 -0.109822E+00  0.000000E+00  0.000000E+00  0.000000E+00 -0.401056E-01
       7           0      0.000000E+00 -0.152555E+00  0.000000E+00  0.000000E+00  0.000000E+00 -0.451822E-01
       8           0      0.000000E+00 -0.199823E+00  0.000000E+00  0.000000E+00  0.000000E+00 -0.491707E-01
       9           0      0.000000E+00 -0.250524E+00  0.000000E+00  0.000000E+00  0.000000E+00 -0.520412E-01
      10           0      0.000000E+00 -0.303526E+00  0.000000E+00  0.000000E+00  0.000000E+00 -0.537721E-01
      11           0      0.000000E+00 -0.357684E+00  0.000000E+00  0.000000E+00  0.000000E+00 -0.543506E-01


 MAXIMUM DISPLACEMENT MAGNITUDE =  0.357684E+00  AT GRID 11
 MAXIMUM ROTATION MAGNITUDE     =  0.543506E-01  AT GRID 11

 EPSILON       =  0.123335E-11
 STRAIN ENERGY =  0.894209E+00
```

Note that regardless of the setting for the parameter `ADDPRESTRESS`, the subcase 1 and subcase 2 global displacement vectors are not combined. The subcase 2 displacement vector does, however, account for the decrease in flexural stiffness due the axial compressive load.

Table 10-1 shows a comparison between Autodesk Inventor Nastran and the theoretical result for beam end deflection and rotation. The theoretical result is based on the formula for a beam under simultaneous axial compression and transverse loading. The formula is:

$$\delta_y = \frac{-V}{kP}(\tan k\ell - k\ell)$$

$$\theta_z = \frac{-V}{P}\frac{1-\cos k\ell}{\cos k\ell}$$

$$k = \sqrt{\frac{P}{EI}}$$

where,

δ_y	is the deflection in the y-direction at the beam end
θ_z	is the rotation about the z-axis at the beam end
V	is the transverse load at the beam end
P	is the axial load
ℓ	is the length of the beam

Table 10-1. Comparison of Theoretical Versus Predicted Beam End Deflection and Rotation.

Deflection			Rotation		
Theoretical (inches)	Autodesk Inventor Nastran (inches)	Difference (%)	Theoretical (radians)	Autodesk Inventor Nastran (radians)	Difference (%)
-0.3577	-0.3577	0.0	-0.05435	-0.05435	0.0

Figure 10-2. Deformed Shape of a Prestress Cantilever Beam.

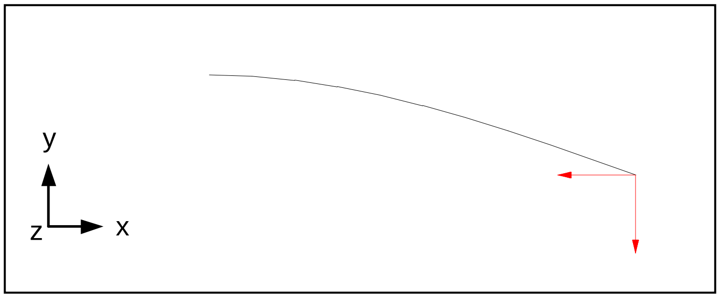


Table 10-2 compares Autodesk Inventor Nastran running a standard `LINEAR STATIC` solution compared to the theoretical result. The differences between the two solutions highlight the importance of prestress analysis.

Table 10-2. Comparison of Theoretical Versus Predicted Beam End Deflection and Rotation Using a Standard Static Solution.

Deflection			Rotation		
Theoretical (inches)	**Autodesk Inventor Nastran (inches)**	**Difference (%)**	**Theoretical (radians)**	**Autodesk Inventor Nastran (radians)**	**Difference (%)**
-0.3577	-0.2500	30.1	-0.05435	-0.03750	31.0

11. LINEAR PRESTRESS MODAL ANALYSIS

11.1 Introduction

Linear prestress modal analysis allows you to determine the natural frequencies and mode shapes of a structure under a defined state of initial stress. A typical example is a vibrating string in tension. The state of initial stress can have a significant effect of the natural frequency of a structure. Tensile membrane forces increase natural frequencies. Compressive membrane forces decrease them and produce a zero frequency at the critical buckling load.

Autodesk Inventor Nastran determines prestress natural frequency by solving the eigenvalue problem:

$$\left| \left([K_l] + [K_S] \right) + \lambda [M] \right| [\phi] = 0$$

$$\lambda_i = \omega_i^2$$

$$f_i = \frac{\omega_i}{2\pi}$$

where,

$[K_l]$ is the global linear stiffness matrix

$[K_S]$ is the global initial stress stiffness matrix

$[M]$ is the global mass matrix

λ_i are the eigenvalues that yield the natural frequencies

ϕ_i are the eigenvectors that represent the natural mode shapes

ω_i are the circular frequencies (radians per second)

f_i are the cyclic frequencies (hertz)

11.2 How to Setup a Model Input File for Linear Prestress Modal Analysis

In Autodesk Inventor Nastran you can perform linear prestress modal analysis by setting `SOLUTION = LINEAR PRESTRESS MODAL` in the Model Initialization File or by specifying `SOL 182` or `SOL LINEAR PRESTRESS MODAL` above the Case Control Section in the Model Input File. Multiple subcases can be specified, each requesting a different constraint or output set.

11.3 Interpreting Results

As an example we will use the simply-supported beam with an applied axial load shown in Figure 11-1. It is desired to find the lowest natural frequency and the corresponding mode shape. Listing 11-1 contains the Model Input File and Listings 11-2 and 11-3 show the extracted frequencies and eigenvectors from the Model Results Output File. The mode shapes are plotted in Figure 11-2.

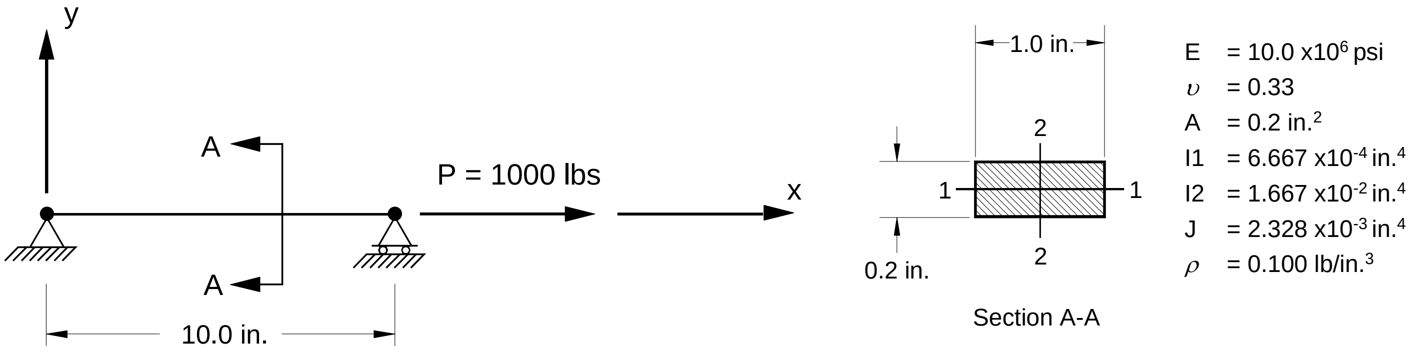


Figure 11-1. 2-D Simply-Supported Beam Example Problem.

Listing 11-1. Model Input File for the 2-D Simply-Supported Beam Problem.

```
$
$ LINEAR PRESTRESS MODAL SOLUTION.
$
SOL LINEAR PRESTRESS MODAL
$
TITLE = INSTALLATION TEST CASE
SUBTITLE= VIBRATION OF A 2-D SIMPLY-SUPPORTED BEAM IN TENSION
$
DISPLACEMENT = ALL
$
SUBCASE 1
 LABEL = PRESTRESS TENSILE LOAD (AXIAL)
 SPC = 1
 LOAD = 1
SUBCASE 2
 LABEL = MODAL
 SPC = 1
 METHOD = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  3,  ,  ,
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  1.,  0.
CBAR,  2,  10,   2,   3,  0.,  1.,  0.
CBAR,  3,  10,   3,   4,  0.,  1.,  0.
CBAR,  4,  10,   4,   5,  0.,  1.,  0.
CBAR,  5,  10,   5,   6,  0.,  1.,  0.
CBAR,  6,  10,   6,   7,  0.,  1.,  0.
CBAR,  7,  10,   7,   8,  0.,  1.,  0.
CBAR,  8,  10,   8,   9,  0.,  1.,  0.
CBAR,  9,  10,   9,  10,  0.,  1.,  0.
CBAR, 10,  10,  10,  11,  0.,  1.,  0.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  6.667E-4,  1.667E-2,  2.328E-3,
,  -0.1,  0.5,  0.1,  0.5,  -0.1,  -0.5,  0.1,  -0.5
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
```

Listing 11-1. Model Input File for the 2-D Simply-Supported Beam Problem. (Continued)

```
$
$ FIXED AT BOTH ENDS, MOVEMENT CONSTRAINED TO X-Y PLANE ONLY.
$
SPC1,  1,  123,   1
SPC1,  1,   23,  11
SPC1,  1,  345,   1,  THRU,  11
$
$ PRESTRESS TENSILE LOAD (AXIAL).
$
FORCE,  1,  11,  0,  1000.,  1.,  0.,  0.
ENDDATA
```

If the beam in Figure 11-1 was unloaded, it would have natural frequency of 178.3 Hz. The 1000 pound tensile load increases the natural frequency to 282.9 Hz. If the load was reversed (i.e., 1000 pound compressive load) the extracted eigenvalues would all be negative, indicating that the beam has buckled (See Figure 11-3).

Listing 11-2. Extracted Eigenvectors for a 2-D Simply-Supported Beam under an Applied Tensile Load.

```
MODAL                                                                   SUBCASE 2

                                                    R E A L   E I G E N V A L U E S

      MODE        EIGENVALUE        RADIANS          CYCLES        GENERALIZED      GENERALIZED      ORTHOGONALITY      ERROR
     NUMBER                                                            MASS          STIFFNESS           LOSS          MEASURE
        1        3.160466E+06    1.777770E+03    2.829409E+02    1.000000E+00    3.160466E+06    0.000000E+00    2.262793E-15
        2        2.767011E+07    5.260238E+03    8.371930E+02    1.000000E+00    2.767011E+07    7.439795E-16    3.461170E-14
        3        1.185489E+08    1.088802E+04    1.732882E+03    1.000000E+00    1.185489E+08    4.250073E-16    5.587210E-12
```

The theoretical result is based on the following formula for the natural frequency of a uniform simply-supported beam under an applied axial load:

$$f_i = \frac{i^2 \pi a}{2} \sqrt{1 + \frac{P_i \ell^2}{i^2 E I \pi^2}}$$

$$a = \sqrt{\frac{EIg}{\rho A}}$$

where,

f_i are the natural frequencies (hertz) corresponding to the i-th mode shape

E is Young's Modulus

I is the moment of inertia about the applicable plane

A is the cross-sectional area

ρ is the material density

g is the gravitational acceleration (units consistent with length dimensions)

ℓ is the length of the beam

Table 11-1 shows a comparison between Autodesk Inventor Nastran and the theoretical natural frequency. Both diagonal and coupled mass matrix formulations are included for comparison.

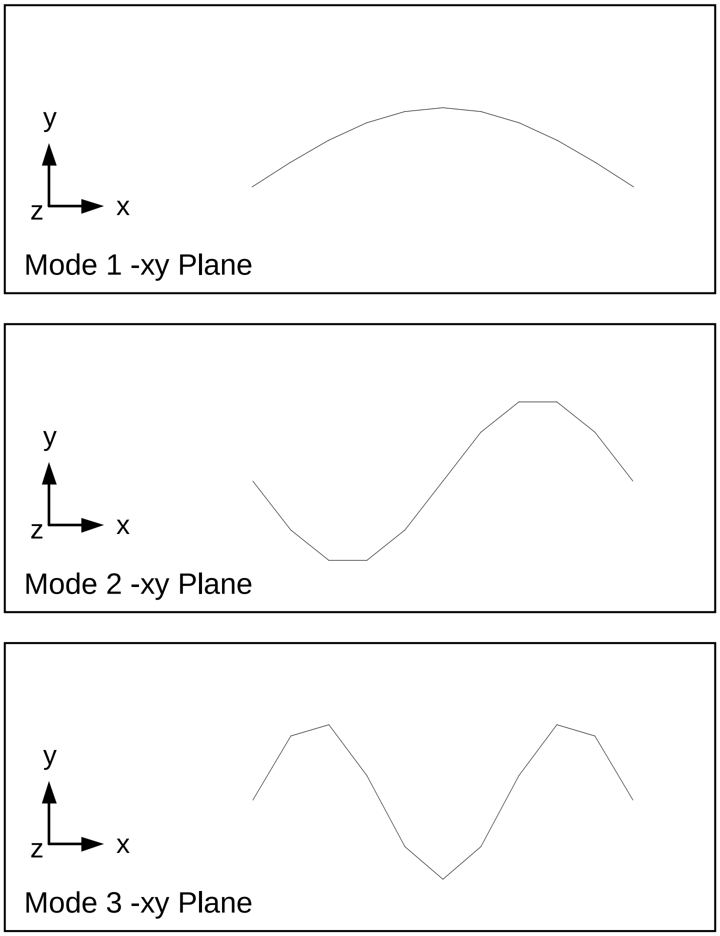
Table 11-1. Comparison of Theoretical Versus Predicted Natural Frequency for a Simply-Supported Beam under an Applied Tensile Load.

	Theoretical	Autodesk Inventor Nastran Diagonal Mass Formulation		Autodesk Inventor Nastran Coupled Mass Formulation	
Mode Number	**Natural Frequency (Hz)**	**Natural Frequency (Hz)**	**Difference (%)**	**Natural Frequency (Hz)**	**Difference (%)**
1	283.0	282.8	0.0	282.9	0.0
2	837.7	835.6	0.3	837.2	0.1
3	1734.7	1724.5	0.6	1732.9	0.1

Listing 11-3. Extracted Eigenvectors for a 2-D Simply-Supported Beam under an Applied Tensile Load.

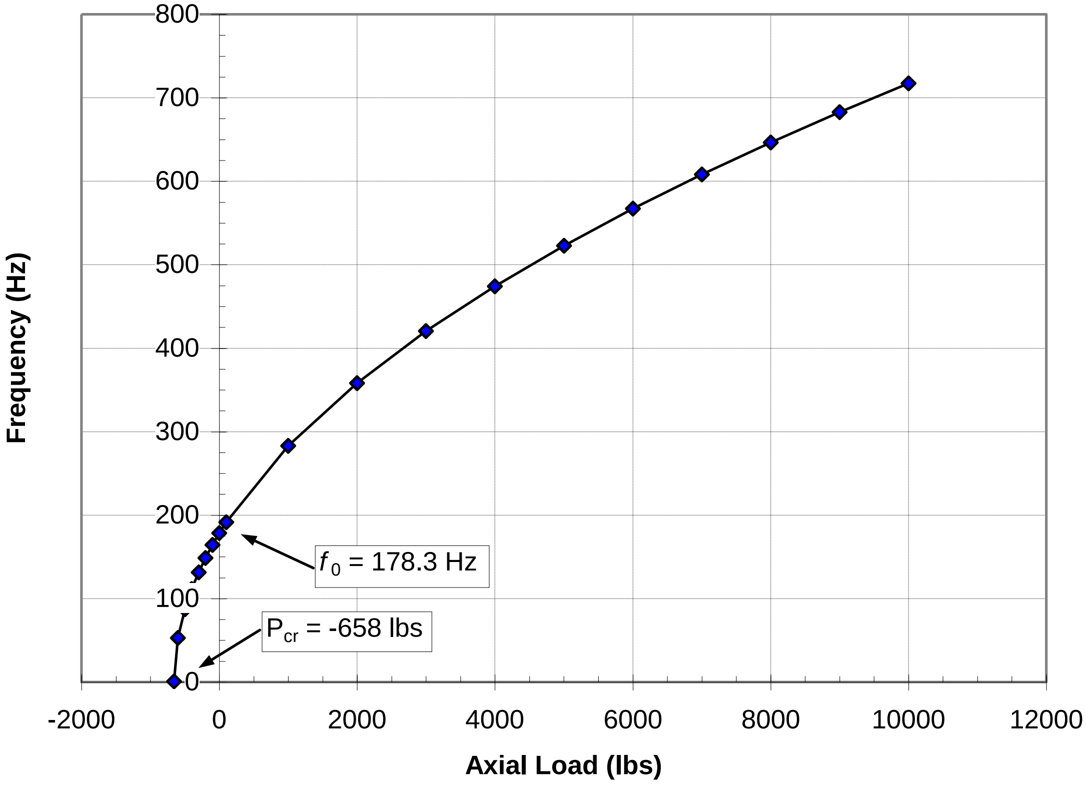
```
MODE = 1  EIGENVALUE = 3.160466E+06  CYCLES = 2.829409E+02          SUBCASE 2

                                       R E A L   E I G E N V E C T O R   N U M B E R   1

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID          ID
        1            0      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.952550E+01
        2            0      0.000000E+00  1.920590E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.856986E+01
        3            0      0.000000E+00  3.653179E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.579646E+01
        4            0      0.000000E+00  5.028170E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.147680E+01
        5            0      0.000000E+00  5.910968E+01  0.000000E+00  0.000000E+00  0.000000E+00  6.033712E+00
        6            0      0.000000E+00  6.215160E+01  0.000000E+00  0.000000E+00  0.000000E+00 -9.558105E-15
        7            0      0.000000E+00  5.910968E+01  0.000000E+00  0.000000E+00  0.000000E+00 -6.033712E+00
        8            0      0.000000E+00  5.028170E+01  0.000000E+00  0.000000E+00  0.000000E+00 -1.147680E+01
        9            0      0.000000E+00  3.653179E+01  0.000000E+00  0.000000E+00  0.000000E+00 -1.579646E+01
       10            0      0.000000E+00  1.920590E+01  0.000000E+00  0.000000E+00  0.000000E+00 -1.856986E+01
       11            0      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00 -1.952550E+01

MODE = 2  EIGENVALUE = 2.767011E+07  CYCLES = 8.371930E+02          SUBCASE 2

                                       R E A L   E I G E N V E C T O R   N U M B E R   2

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID          ID
        1            0      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.903967E+01
        2            0      0.000000E+00  3.652110E+01  0.000000E+00  0.000000E+00  0.000000E+00  3.158375E+01
        3            0      0.000000E+00  5.909239E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.206392E+01
        4            0      0.000000E+00  5.909239E+01  0.000000E+00  0.000000E+00  0.000000E+00 -1.206392E+01
        5            0      0.000000E+00  3.652110E+01  0.000000E+00  0.000000E+00  0.000000E+00 -3.158375E+01
        6            0      0.000000E+00  8.511426E-15  0.000000E+00  0.000000E+00  0.000000E+00 -3.903967E+01
        7            0      0.000000E+00 -3.652110E+01  0.000000E+00  0.000000E+00  0.000000E+00 -3.158375E+01
        8            0      0.000000E+00 -5.909239E+01  0.000000E+00  0.000000E+00  0.000000E+00 -1.206392E+01
        9            0      0.000000E+00 -5.909239E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.206392E+01
       10            0      0.000000E+00 -3.652110E+01  0.000000E+00  0.000000E+00  0.000000E+00  3.158375E+01
       11            0      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.903967E+01

MODE = 3  EIGENVALUE = 1.185489E+08  CYCLES = 1.732882E+03          SUBCASE 2

                                       R E A L   E I G E N V E C T O R   N U M B E R   3

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID          ID
        1            0      0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  5.856185E+01
        2            0      2.951504E-15  5.026867E+01  0.000000E+00  0.000000E+00  0.000000E+00  3.442179E+01
        3            0      5.254517E-15  5.909436E+01  0.000000E+00  0.000000E+00  0.000000E+00 -1.809661E+01
        4            0      6.406638E-15  1.920092E+01  0.000000E+00  0.000000E+00  0.000000E+00 -5.569563E+01
        5            0      6.160262E-15 -3.652232E+01  0.000000E+00  0.000000E+00  0.000000E+00 -4.737753E+01
        6            0      4.571684E-15 -6.213549E+01  0.000000E+00  0.000000E+00  0.000000E+00 -7.534974E-15
        7            0      1.987842E-15 -3.652232E+01  0.000000E+00  0.000000E+00  0.000000E+00  4.737753E+01
        8            0     -1.026673E-15  1.920092E+01  0.000000E+00  0.000000E+00  0.000000E+00  5.569563E+01
        9            0     -3.813385E-15  5.909436E+01  0.000000E+00  0.000000E+00  0.000000E+00  1.809661E+01
       10            0     -5.767348E-15  5.026867E+01  0.000000E+00  0.000000E+00  0.000000E+00 -3.442179E+01
       11            0     -6.468533E-15  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00 -5.856185E+01
```

Figure 11-2. Mode Shapes of a 2-D Simply-Supported Beam Under an Applied Tensile Load.



The effect of axial load on natural frequency for the problem of Figure 11-1 is shown in Figure 11-3. Note that when the applied load becomes compressive, the frequency decreases rapidly until instability occurs at -658 pounds and 0 Hz. This load is the same critical load you would have arrived at if you ran a linear buckling solution.

Figure 11-3. Effect of Axial Load on Natural Frequency for a Simply-Supported Beam.



12. LINEAR PRESTRESS TRANSIENT RESPONSE ANALYSIS

12.1 Introduction

Linear prestress transient response analysis allows you to determine the response of a structure under a defined state of initial stress. A typical example is a vibrating string in tension. The state of initial stress can have a significant effect of the natural frequency of a structure, which will affect the response. Tensile membrane forces increase natural frequencies. Compressive membrane forces decrease them and produce a zero frequency at the critical buckling load.

12.2 How to Setup a Model Input File for Linear Prestress Transient Response Analysis

In Autodesk Inventor Nastran you can perform linear prestress transient response analysis by setting `SOLUTION = LINEAR PRESTRESS TRANSIENT RESPONSE` in the Model Initialization File or by specifying `SOL 184` or `SOL LINEAR PRESTRESS TRANSIENT RESPONSE` above the Case Control Section in the Model Input File. Multiple subcases can be specified, each requesting a different load or output set. Only one reference to an `EIGRL` Bulk Data entry (`METHOD` Case Control command) is permitted. This request should be placed above the first subcase. Linear prestress transient response requires a modal transient response solution. Direct transient response is not supported.

13. LINEAR PRESTRESS FREQUENCY RESPONSE ANALYSIS

13.1 Introduction

Linear prestress frequency response analysis allows you to determine the response of a structure under a defined state of initial stress. A typical example is a vibrating string in tension. The state of initial stress can have a significant effect of the natural frequency of a structure, which will affect the response. Tensile membrane forces increase natural frequencies. Compressive membrane forces decrease them and produce a zero frequency at the critical buckling load.

13.2 How to Setup a Model Input File for Linear Prestress Frequency Response Analysis

In Autodesk Inventor Nastran you can perform linear prestress frequency response analysis by setting `SOLUTION = LINEAR PRESTRESS FREQUENCY RESPONSE` in the Model Initialization File or by specifying `SOL 183` or `SOL LINEAR PRESTRESS FREQUENCY RESPONSE` above the Case Control Section in the Model Input File. Multiple subcases can be specified, each requesting a different load or output set. Only one reference to an `EIGRL` Bulk Data entry (`METHOD` Case Control command) is permitted. This request should be placed above the first subcase. Linear prestress frequency response requires a modal frequency response solution. Direct frequency response is not supported.

14. LINEAR PRESTRESS COMPLEX EIGENVALUE ANALYSIS

14.1 Introduction

Linear complex eigenvalue analysis allows you to determine the response of a structure under a defined state of initial stress. A typical example is turbine rotating at high speed. The state of initial stress can have a significant effect of the natural frequency of a structure, which will affect the response. Tensile membrane forces increase natural frequencies. Compressive membrane forces decrease them and produce a zero frequency at the critical buckling load.

14.2 How to Setup a Model Input File for Linear Prestress Complex Eigenvalue Analysis

In Autodesk Inventor Nastran you can perform linear prestress complex eigenvalue analysis by setting `SOLUTION = LINEAR PRESTRESS COMPLEX EIGENVALUE` in the Model Initialization File or by specifying `SOL 188` or `SOL LINEAR PRESTRESS COMPLEX EIGENVALUE` above the Case Control Section in the Model Input File. Multiple subcases can be specified, each requesting a different output set. Each subcase must also reference an `EIGC` Bulk Data entry via the `CMETHOD` Case Control command.

15. NONLINEAR STATIC ANALYSIS

15.1 Introduction

There are many types of behavior that may be referred to as nonlinear. Some examples of nonlinear behavior include materials that change properties as they are loaded, displacements which cause loads to alter their distribution or magnitude, gaps which may open or close. The degree of nonlinearity may be mild or severe.

In linear static analysis we assume that displacements and rotations are small, supports do not settle, stress is directly proportional to strain, and loads maintain their original directions as the structure deforms. Most problems can usually be considered linear because they are loaded in their linear elastic, small deflection range. For these types of problems, the slight nonlinearity does not affect the results and the difference between a linear and nonlinear solution is negligible.

While many practical problems can be solved using linear analysis, some or all of its inherent assumptions may not be valid. Adjacent parts may make or break contact with the contact area changing as the loads change. Elastic materials may become plastic, or the material may not have a linear stress-strain relation at any stress level. Part of the structure may lose stiffness because of buckling or material failure. Displacements and rotations may become large enough that equilibrium equations must be written for the deformed rather than the original configuration. Large rotations cause pressure loads to change in direction, and also to change in magnitude if there is a change in area to which they are applied.

Nonlinear static analysis is implemented in Autodesk Inventor Nastran as an iterative process using the Newton-Raphson method where the path dependent problem is broken down into several linear steps. The equilibrium equations in incremental form can be written as:

$$\left[K_t\right]\left\{\Delta D\right\}=\left\{\Delta R\right\}$$

where,

$\left[K_t\right]$ is the global tangent stiffness matrix

$\left\{\Delta D\right\}$ is the global incremental displacement vector

$\left\{\Delta R\right\}$ is the global incremental load vector

The global tangent stiffness matrix $\left[K_t\right]$ is a function of the global displacements $\left\{D\right\}$ because the problem is nonlinear. The current global displacement vector is the sum of the preceding $\left\{\Delta D\right\}$'s.

The iterative process allows Autodesk Inventor Nastran to solve contact, geometric (large displacement and rotation) or material (nonlinear elastic, elastic-plastic, and perfectly plastic) nonlinear statics problems. Several examples are given in this section which demonstrate how to setup, run, and interpret results for these types of problems.

15.2 How to Setup a Model Input File for Nonlinear Static Analysis

In Autodesk Inventor Nastran you can solve a nonlinear statics problem by setting `SOLUTION = NONLINEAR STATIC` in the Model Initialization File or by specifying `SOL 106` or `SOL NONLINEAR STATIC` above the Case Control Section in the Model Input File, and following the guidelines listed below:

1. Most nonlinear statics problems can be setup the same as for linear statics (geometry, boundary conditions, loading, etc.). As a minimum, all subcases must reference an `NLPARM` Bulk Data entry via the `NLPARM` Case Control Command. The `NLPARM` entry controls the nonlinear iteration parameters (increment size, number of increments, output control, etc.). Since the solution to a particular load involves a nonlinear search procedure, the solution is not guaranteed. Care must be used when selecting the search procedures on the `NLPARM` Bulk Data entry. You may override nearly all iteration control restrictions.
2. All loads, boundary conditions, elements, element properties (except `PCOMP` with material nonlinearity), and material properties that are supported in linear statics analysis are supported in nonlinear statics.
3. For contact solutions, gap (`CGAP)`, slide line (`BCONP`), or surface contact (`BSCONP`) elements must be specified. Contact elements can be used with all loads, boundary conditions, elements and types of nonlinearity supported. Note that for gap elements, contact planes do not rotate as a function of displacement. The user-specified stiffnesses (`KA`, `KB`, and `KT` on the `PGAP` Bulk Data entry) must be carefully selected when the non-adaptive form is used (`TMAX` ≤ 0.0 on the `PGAP` Bulk Data entry). An optimal selection of values is usually a compromise between accuracy and numerical performance. Slide line and surface contact elements rotate as a function of displacement, if large displacement effects are turned on (`PARAM`, `LGDISP`, `ON`), and allow elements to slide past each other.
4. Follower forces (forces that follow the deformed geometry) are generated automatically when using element pressures (`PLOAD1`, `PLOAD2`, and `PLOAD4`), element temperatures (`TEMP`, `TEMPD`, `TEMPP1`, and `TEMPRB`), acceleration loads (`GRAV` and `RFORCE`), and grid point forces and moments (`FORCE1` and `MOMENT1`). Follower force effects are controlled using the `LGDISP` parameter.
5. Constraints apply only to the nonrotated displacements at a grid point. In particular, multipoint constraints and rigid elements may cause problems if the connected grid points undergo large motions. However, also note that replacement of the constraints with overly stiff elements may result in convergence problems.
6. Large deformations of elements may cause nonequilibrium loading effects. All elements are assumed to have constant length, area, and volume. Large displacement effects are controlled using the `LGDISP` parameter.
7. In large displacement analysis there are two different approaches for the angular motions: gimbal angle and rotation vector. In the gimbal angle approach, angular motions are treated as three ordered rotations about the x, y, and z-axes. The gimbal angle approach is requested by specifying `PARAM`, `LANGLE`, `1` (default) in the Model Input File. In the rotation vector approach, the three angular motions are treated as a vector. The rotation is about the rotation axis and the magnitude of rotation is equal to the amplitude of the rotation vector. The rotation vector approach is requested by specifying `PARAM`, `LANGLE`, `2` in the Model Input File.

8. Material nonlinear solutions require a `MATS1` Bulk Data entry be specified for elements that have nonlinear material properties. Both linear and nonlinear materials may be specified in the same solution. Material nonlinear properties can be used with all loads, boundary conditions, elements and types of nonlinearity supported. Bar and rod elements support material nonlinearity only in the axial direction. Better performance may be achieved when using quad elements and elastic-plastic materials if `PARAM`, `QUADINODE` is set to `OFF` and `PARAM`, `QUADRNODE` is set to `ON`.
9. The use of `CQUADR` and `CTRIAR` elements are preferred over the use of `PARAM`, `K6ROT` when large displacements effects are turned on (`PARAM`, `LGDISP`, `ON`).
10. Unlike other solutions, subcase loads and results are additive. This allows different loads and boundary conditions to be applied in a specific sequence to the structure. Additionally, different nonlinear iteration parameters (`NLPARM`) may be specified for each subcase allowing further control. To initialize each subcase to zero set `PARAM`, `NLSUBCREINIT` to `ON`. This setting allows multiple subcases with each having the same zero starting point.
11. Arc-length methods are recommended for models where snap-through and post-buckling behavior exists. Arc-length parameters are specified on the `NLPCI` Bulk Data entry.
12. Models should be simple and relatively small initially to gain insight into behavior and verify the approach taken. A linear static solution should be run first to verify boundary conditions and loading. Large displacement and follower force effects can be turned off by setting `PARAM`, `LGDISP` to `OFF`.

15.3 Interpreting Results

In this section we will present several examples demonstrating the features and capabilities of nonlinear static analysis. We will look at three types of nonlinearity: geometric (large displacement and rotation), material (nonlinear elastic and elastic-plastic), and contact.

15.3.1 Large Rotations

The first problem is an example of very large displacement and rotation. It is desired to rotate the free end of the beam shown in Figure 15-1 completely around 360°. Listing 15-1 contains the Model Input File.

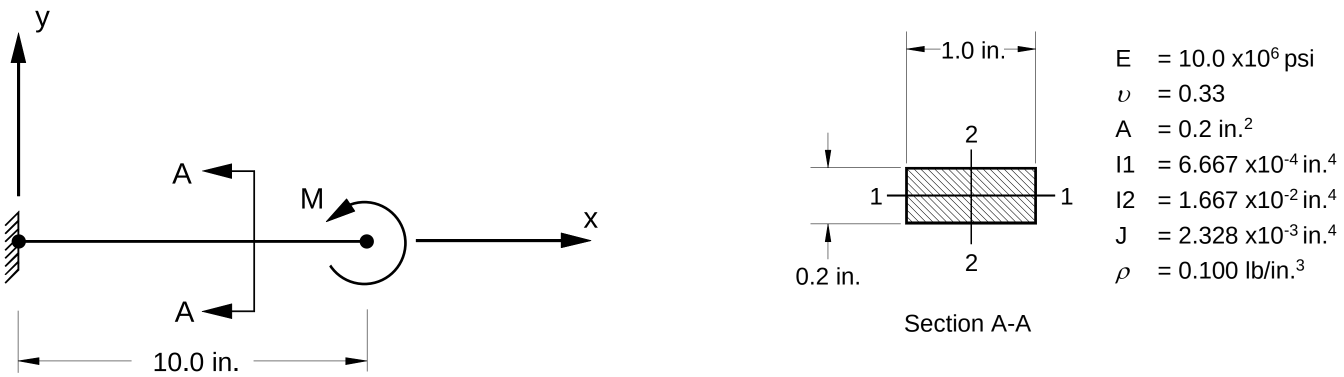


Figure 15-1. 2-D Cantilever Beam Example Problem with an End Moment.

Listing 15-1. Model Input File for the Cantilever Beam Problem with an End Moment.

```
$
$ NONLINEAR STATIC SOLUTION.
$
SOL NONLINEAR STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = VERY LARGE ROTATION OF A CANTILEVER BEAM
$
DISPLACEMENT = ALL
$
SUBCASE 1
 LABEL = MOMENT AT FREE END ABOUT Z-DIR
 LOAD = 1
 NLPARM = 1
 SPC = 1
$
BEGIN BULK
$
$ TURN ON LARGE DISPLACEMENT EFFECTS.
$
PARAM, LGDISP, ON
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  20,  ,  ITER,  5,  ,  ,  YES
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
$
$ BAR ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  1.667E-2,  6.667E-4,  2.328E-3,
,  -0.5,  0.1,  0.5,  0.1,  -0.5,  -0.1,  0.5,  -0.1
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ FIXED AT ONE END, MOVEMENT CONSTRAINED TO X-Y PLANE ONLY.
$
SPC1,  1,  123456,  1
SPC1,  1,     345,  1,  THRU,  11
$
$ END MOMENT (Z-DIRECTION).
$
MOMENT,  1,  11,  0,  4189.,  0.,  0.,  1.
ENDDATA
```

Note that the orientation vector defining the bar element coordinate system must be defined parallel to the axis of rotation. Since the bar rotates about the global z-axis, the element y-axis is defined to be the global z-axis. This is important because the updated element coordinate system will be calculated using the undeformed element y-axis definition.

The `NLPARM` entry controls the nonlinear iteration and intermediate output. Here, we have requested that the 4189 inch-pound load be divided into 20 increments and that intermediate output be supplied for each increment. The maximum displacements from the Model Results Output File are shown in tabular form in Listing 15-2. Maximum vector results are output for each subcase. Note that the units for rotation are in radians. The deflected shapes are plotted in Figure 15-2.

Listing 15-2. Load Increment Maximum Displacements and Rotations.

```
                                          M A X I M U M   D I S P L A C E M E N T S

     LOAD         LOAD               T1             T2             T3             R1             R2             R3
   INCREMENT     FACTOR
       1     5.000000E-02     1.632789E-01  1.557983E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.141592E-01
       2     1.000000E-01     6.435680E-01  3.040089E+00  0.000000E+00  0.000000E+00  0.000000E+00  6.283185E-01
       3     1.500000E-01     1.412885E+00  4.375353E+00  0.000000E+00  0.000000E+00  0.000000E+00  9.424777E-01
       4     2.000000E-01     2.426751E+00  5.502288E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.256637E+00
       5     2.500000E-01     3.627253E+00  6.372747E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.570796E+00
       6     3.000000E-01     4.947011E+00  6.954843E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.884956E+00
       7     3.500000E-01     6.313747E+00  7.234680E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.199115E+00
       8     4.000000E-01     7.655110E+00  7.216829E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.513274E+00
       9     4.500000E-01     8.903427E+00  6.923490E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.827433E+00
      10     5.000000E-01     1.000000E+01  6.392453E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.141593E+00
      11     5.500000E-01     1.089868E+01  5.814916E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.455752E+00
      12     6.000000E-01     1.156842E+01  5.315670E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.769911E+00
      13     6.500000E-01     1.199474E+01  4.911833E+00  0.000000E+00  0.000000E+00  0.000000E+00  4.084071E+00
      14     7.000000E-01     1.217989E+01  4.579621E+00  0.000000E+00  0.000000E+00  0.000000E+00  4.398230E+00
      15     7.500000E-01     1.214183E+01  4.257288E+00  0.000000E+00  0.000000E+00  0.000000E+00  4.712389E+00
      16     8.000000E-01     1.191213E+01  4.005218E+00  0.000000E+00  0.000000E+00  0.000000E+00  5.026549E+00
      17     8.500000E-01     1.153297E+01  3.785962E+00  0.000000E+00  0.000000E+00  0.000000E+00  5.340708E+00
      18     9.000000E-01     1.105341E+01  3.528039E+00  0.000000E+00  0.000000E+00  0.000000E+00  5.654867E+00
      19     9.500000E-01     1.052547E+01  3.379959E+00  0.000000E+00  0.000000E+00  0.000000E+00  5.969026E+00
      20     1.000000E+00     9.999999E+00  3.236068E+00  0.000000E+00  0.000000E+00  0.000000E+00  6.283186E+00
```

Table 15-1 shows a comparison between Autodesk Inventor Nastran and the theoretical end rotation. The formula is:

$$\theta_{end} = \frac{M\ell}{EI}$$

where,

θ_{end} is the end rotation

M is the end moment

E is Young's Modulus

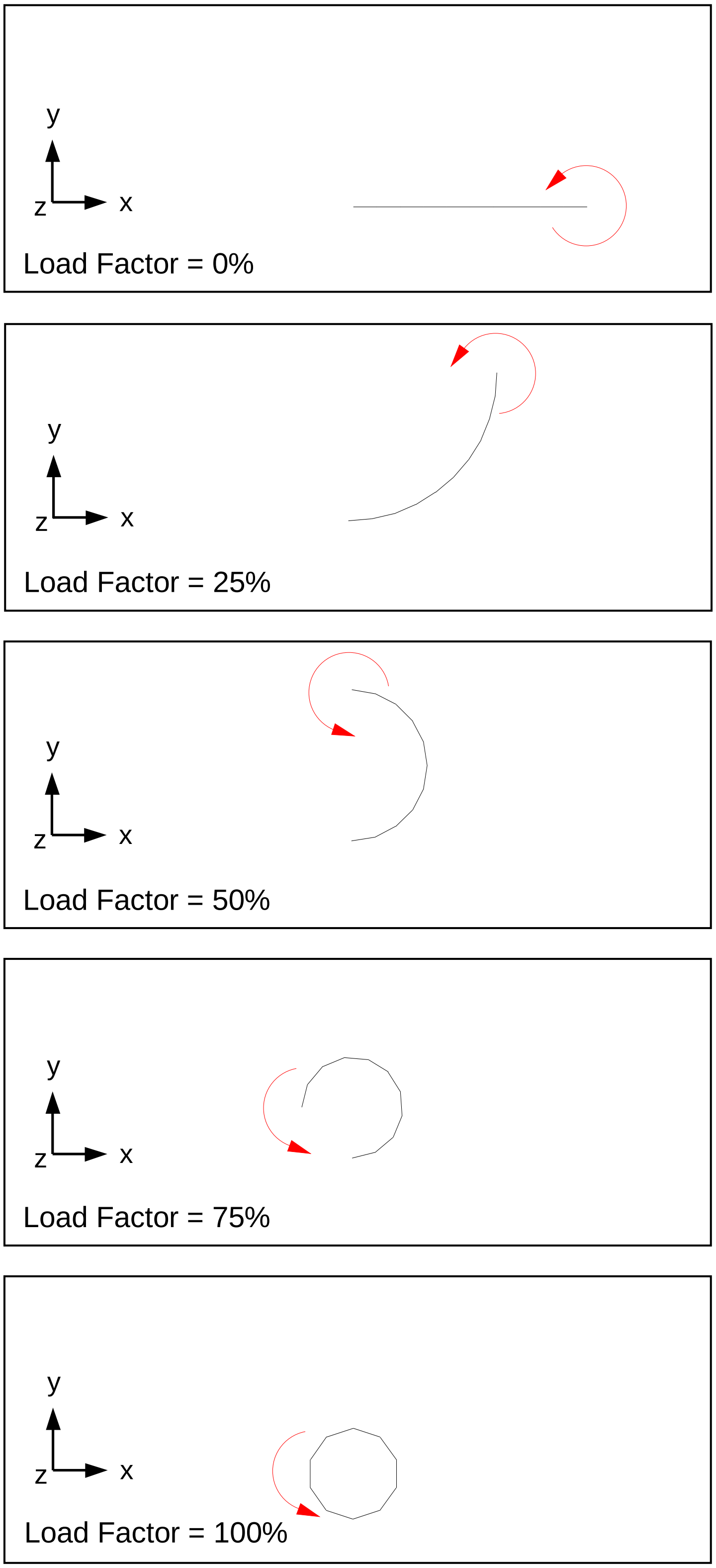
I is the moment of intertia about the applicable plane

ℓ is the length of the beam

Table 15-1. Comparison of Theoretical Versus Predicted Beam End Rotations.

Load Increment	Load Factor (%)	Theoretical (degrees)	Autodesk Inventor Nastran (degrees)	Difference (%)
5	25	90.0	90.0	0.0
10	50	180.0	180.0	0.0
15	75	270.0	270.0	0.0
20	100	360.0	360.0	0.0

Figure 15-2. Deformed Shapes of a Cantilever Beam with an End Moment.



15.3.2 Large Displacements

The next problem is another example of very large displacement and rotation. Initially, the beam in Figure 15-3 is subjected to an axial force $P = 0.9P_{cr}$ and a small lateral force $P_i = 0.1P_{cr}$ (initial disturbance) at the free end. The lateral force is subsequently removed and large rotations of the beam are produced when the axial force is increased above the critical value P_{cr}. The critical load is calculated from Reference 18 using:

$$P_{cr} = \frac{\pi^2 EI}{4\ell^2}$$

Listing 15-3 contains the Model Input File.

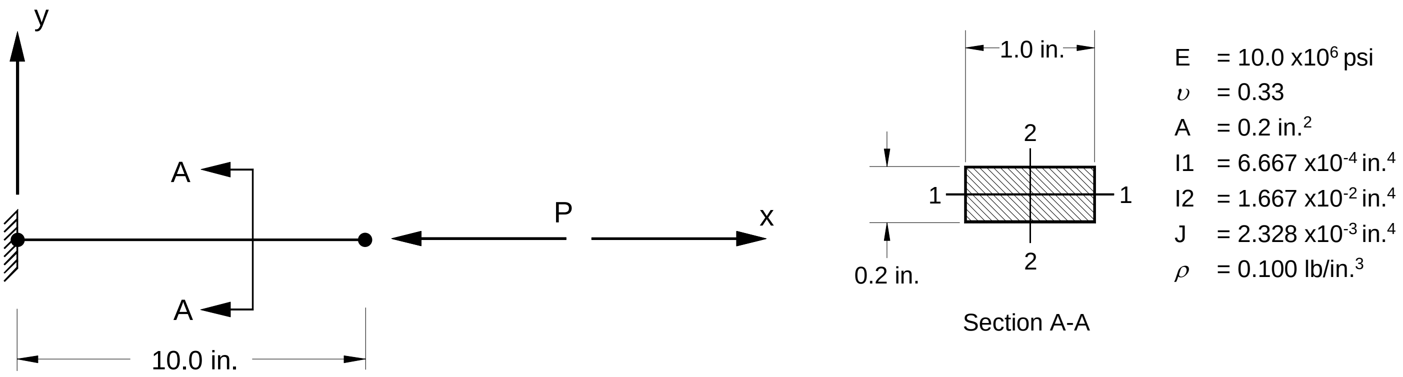


Figure 15-3. 2-D Cantilever Beam Example Problem with an Axial End Force.

Listing 15-3. Model Input File for the Cantilever Beam Problem with an Axial End Force.

```
$
$ NONLINEAR STATIC SOLUTION.
$
SOL NONLINEAR STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = POST BUCKLING ANALYSIS OF A CANTILEVER BEAM
$
DISPLACEMENT = ALL
FORCE = ALL
$
SPC = 1
NLPARM = 1
SUBCASE 1
 LABEL = P/Pcr = 0.900 PLUS LATERAL
 LOAD = 1
SUBCASE 2
 LABEL = P/Pcr = 1.152
 LOAD = 2
SUBCASE 3
 LABEL = P/Pcr = 1.518
 LOAD = 3
SUBCASE 4
 LABEL = P/Pcr = 2.541
 LOAD = 4
SUBCASE 5
 LABEL = P/Pcr = 4.029
 LOAD = 5
SUBCASE 6
 LABEL = P/Pcr = 9.116
 LOAD = 6
$
BEGIN BULK
$
$ TURN ON LARGE DISPLACEMENT EFFECTS.
$
PARAM, LGDISP, ON
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  10,  ,  ITER,  1,  ,  ,  NO
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
$
$ BAR ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  1.667E-2,  6.667E-4,  2.328E-3,
,  -0.5,  0.1,  0.5,  0.1,  -0.5,  -0.1,  0.5,  -0.1
```

Listing 15-3. Model Input File for the Cantilever Beam Problem with an Axial End Force. (Continued)

```
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ FIXED AT ONE END, MOVEMENT CONSTRAINED TO X-Y PLANE ONLY.
$
SPC1,  1,  123456,  1
SPC1,  1,     345,  1,  THRU,  11
$
$ LOADING (Pcr = 164.5).
$
FORCE,  1,  11,  0,  164.5,  -0.900,   0.1,  0.
FORCE,  2,  11,  0,  164.5,  -1.152,   0.,   0.
FORCE,  3,  11,  0,  164.5,  -1.518,   0.,   0.
FORCE,  4,  11,  0,  164.5,  -2.541,   0.,   0.
FORCE,  5,  11,  0,  164.5,  -4.029,   0.,   0.
FORCE,  6,  11,  0,  164.5,  -9.116,   0.,   0.
ENDDATA
```

In this example, the subcase structure is used to control the load application with the `NLPARM` entry used to control the nonlinear iteration and intermediate output. The analysis is performed in six subcases with each subcase divided into 10 increments. The first subcase contains the initial loading, which triggers the large bending beyond the critical buckling load. The lateral force is then removed with each of the remaining subcases corresponding to an increasing axial force ratio, P/P_{cr}, of 1.152, 1.518, 2.541, 4.029, and 9.116. The `ITER` option of `KMETHOD` on the `NLPARM` Bulk Data entry is selected with `KSTEP` set to 1, specifying the full Newton’s method. The maximum displacements from the Model Results Output File are shown in tabular form in Listing 15-4. Theoretical results given in Reference 18 at each P/P_{cr} ratio are compared to predicted values in Table 15-2. The deflected shapes are plotted in Figure 15-4.

Listing 15-4. Subcase and Load Increment Maximum Displacements and Rotations.

```
P/Pcr = 0.900 PLUS LATERAL                                          SUBCASE 1

                                          M A X I M U M   D I S P L A C E M E N T S

    LOAD        LOAD              T1            T2            T3            R1            R2            R3
  INCREMENT    FACTOR
      1     1.000000E-01     5.626600E-04  9.024624E-02  0.000000E+00  0.000000E+00  0.000000E+00  1.358794E-02
      2     2.000000E-01     2.552055E-03  1.999392E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.022102E-02
      3     3.000000E-01     7.031550E-03  3.360746E-01  0.000000E+00  0.000000E+00  0.000000E+00  5.100405E-02
      4     4.000000E-01     1.598085E-02  5.093251E-01  0.000000E+00  0.000000E+00  0.000000E+00  7.762938E-02
      5     5.000000E-01     3.329118E-02  7.366284E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.128003E-01
      6     6.000000E-01     6.711409E-02  1.045930E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.610273E-01
      7     7.000000E-01     1.356572E-01  1.484171E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.300512E-01
      8     8.000000E-01     2.811141E-01  2.126161E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.328186E-01
      9     9.000000E-01     5.915713E-01  3.052338E+00  0.000000E+00  0.000000E+00  0.000000E+00  4.857224E-01
     10     1.000000E+00     1.173620E+00  4.215686E+00  0.000000E+00  0.000000E+00  0.000000E+00  6.897444E-01
```

Listing 15-4. Subcase and Load Increment Maximum Displacements and Rotations. (Continued)

```
P/Pcr = 1.152                                                       SUBCASE 2

                                            M A X I M U M   D I S P L A C E M E N T S

      LOAD        LOAD               T1            T2            T3            R1            R2            R3
    INCREMENT    FACTOR
        1     1.000000E-01     1.238127E+00  4.319003E+00  0.000000E+00  0.000000E+00  0.000000E+00  7.095212E-01
        2     2.000000E-01     1.314203E+00  4.436674E+00  0.000000E+00  0.000000E+00  0.000000E+00  7.321863E-01
        3     3.000000E-01     1.404031E+00  4.570193E+00  0.000000E+00  0.000000E+00  0.000000E+00  7.581309E-01
        4     4.000000E-01     1.509937E+00  4.720654E+00  0.000000E+00  0.000000E+00  0.000000E+00  7.877135E-01
        5     5.000000E-01     1.634144E+00  4.888358E+00  0.000000E+00  0.000000E+00  0.000000E+00  8.211911E-01
        6     6.000000E-01     1.778443E+00  5.072444E+00  0.000000E+00  0.000000E+00  0.000000E+00  8.586464E-01
        7     7.000000E-01     1.943816E+00  5.270653E+00  0.000000E+00  0.000000E+00  0.000000E+00  8.999279E-01
        8     8.000000E-01     2.130151E+00  5.479375E+00  0.000000E+00  0.000000E+00  0.000000E+00  9.446312E-01
        9     9.000000E-01     2.336150E+00  5.694045E+00  0.000000E+00  0.000000E+00  0.000000E+00  9.921378E-01
       10     1.000000E+00     2.559498E+00  5.909754E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.041701E+00

  P/Pcr = 1.518                                                       SUBCASE 3

                                            M A X I M U M   D I S P L A C E M E N T S

      LOAD        LOAD               T1            T2            T3            R1            R2            R3
    INCREMENT    FACTOR
        1     1.000000E-01     3.074797E+00  6.352419E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.148729E+00
        2     2.000000E-01     3.555460E+00  6.703099E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.242440E+00
        3     3.000000E-01     4.004936E+00  6.984954E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.325958E+00
        4     4.000000E-01     4.426052E+00  7.213414E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.401322E+00
        5     5.000000E-01     4.821328E+00  7.399455E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.469981E+00
        6     6.000000E-01     5.193009E+00  7.551187E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.533003E+00
        7     7.000000E-01     5.543090E+00  7.674782E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.591209E+00
        8     8.000000E-01     5.873355E+00  7.775047E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.645243E+00
        9     9.000000E-01     6.185400E+00  7.855789E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.695620E+00
       10     1.000000E+00     6.480659E+00  7.920077E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.742766E+00

  P/Pcr = 2.541                                                       SUBCASE 4

                                            M A X I M U M   D I S P L A C E M E N T S

      LOAD        LOAD               T1            T2            T3            R1            R2            R3
    INCREMENT    FACTOR
        1     1.000000E-01     7.227439E+00  8.032060E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.860181E+00
        2     2.000000E-01     7.875935E+00  8.072446E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.960562E+00
        3     3.000000E-01     8.444208E+00  8.065437E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.047726E+00
        4     4.000000E-01     8.946105E+00  8.026574E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.124319E+00
        5     5.000000E-01     9.392528E+00  7.966190E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.192270E+00
        6     6.000000E-01     9.792158E+00  7.891328E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.253035E+00
        7     7.000000E-01     1.015199E+01  7.806881E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.307739E+00
        8     8.000000E-01     1.047770E+01  7.716299E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.357268E+00
        9     9.000000E-01     1.077398E+01  7.622038E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.402336E+00
       10     1.000000E+00     1.104469E+01  7.525861E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.443524E+00

  P/Pcr = 4.029                                                       SUBCASE 5

                                            M A X I M U M   D I S P L A C E M E N T S

      LOAD        LOAD               T1            T2            T3            R1            R2            R3
    INCREMENT    FACTOR
        1     1.000000E-01     1.139931E+01  7.385070E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.497474E+00
        2     2.000000E-01     1.171466E+01  7.245492E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.545407E+00
        3     3.000000E-01     1.199709E+01  7.108755E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.588254E+00
        4     4.000000E-01     1.225167E+01  6.975835E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.626763E+00
        5     5.000000E-01     1.248250E+01  6.847281E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.661534E+00
        6     6.000000E-01     1.269292E+01  6.723368E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.693062E+00
        7     7.000000E-01     1.288566E+01  6.604190E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.721756E+00
        8     8.000000E-01     1.306300E+01  6.489723E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.747958E+00
        9     9.000000E-01     1.322683E+01  6.379867E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.771959E+00
       10     1.000000E+00     1.337876E+01  6.274479E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.794003E+00

  P/Pcr = 9.116                                                       SUBCASE 6

                                            M A X I M U M   D I S P L A C E M E N T S

      LOAD        LOAD               T1            T2            T3            R1            R2            R3
    INCREMENT    FACTOR
        1     1.000000E-01     1.382518E+01  5.945448E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.857236E+00
        2     2.000000E-01     1.418606E+01  5.658976E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.906079E+00
        3     3.000000E-01     1.448583E+01  5.407911E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.944554E+00
        4     4.000000E-01     1.474027E+01  5.186232E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.975339E+00
        5     5.000000E-01     1.496002E+01  4.989041E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.000293E+00
        6     6.000000E-01     1.515253E+01  4.812396E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.020740E+00
        7     7.000000E-01     1.532318E+01  4.653128E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.037652E+00
        8     8.000000E-01     1.547595E+01  4.508673E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.051753E+00
        9     9.000000E-01     1.561387E+01  4.376944E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.063596E+00
       10     1.000000E+00     1.573928E+01  4.256233E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.073603E+00
```

Table 15-2a. Comparison of Theoretical Versus Predicted Beam End X-Displacements.

P/P_{cr}	Autodesk Inventor Nastran X_{end}/L	Theoretical X_{end}/L	Difference (%)
1.152	0.744	0.741	0.4
1.518	0.352	0.349	0.9
2.541	0.105	0.107	1.9
4.029	-0.338	-0.340	0.6
9.116	-0.574	-0.577	0.5

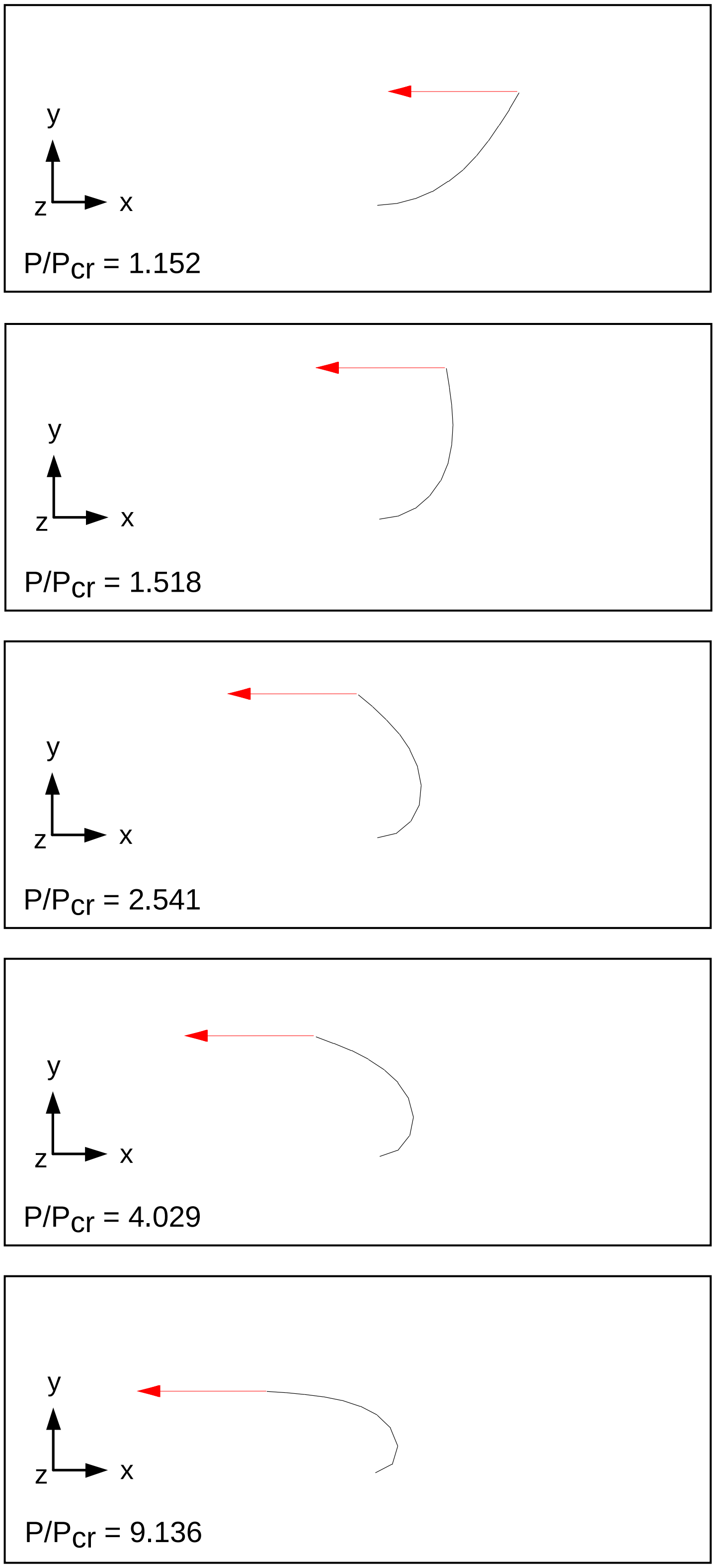
Table 15-2b. Comparison of Theoretical Versus Predicted Beam End Y-Displacements.

P/P_{cr}	Autodesk Inventor Nastran Y_{end}/L	Theoretical Y_{end}/L	Difference (%)
1.152	0.591	0.593	0.3
1.518	0.792	0.792	0.0
2.541	0.753	0.750	0.4
4.029	0.627	0.625	0.3
9.116	0.426	0.421	1.2

Table 15-2c. Comparison of Theoretical Versus Predicted Beam End Rotations.

P/P_{cr}	Autodesk Inventor Nastran θ_{end}	Theoretical θ_{end}	Difference (%)
1.152	59.7	60.0	0.5
1.518	99.9	100.0	0.1
2.541	140.0	140.0	0.0
4.029	160.1	160.0	0.1
9.116	176.1	176.0	0.1

Figure 15-4. Deformed Shapes of a Cantilever Beam with an Axial End Force.



15.3.3 Nonlinear-Elastic Material

The next problem is an example of material nonlinearity. The beam in Figure 15-5 is made from a nonlinear elastic material and subjected to a 1000 pound axial force. The load is divided into 20 increments.

The material behavior is stress dependent and represented graphically in Figure 15-6. The stress dependence is input using `MATS1` and `TABLES1` Bulk Data entries. The `MATS1` Bulk Data entry must reference an isotropic material. Listing 15-5 contains the Model Input File.

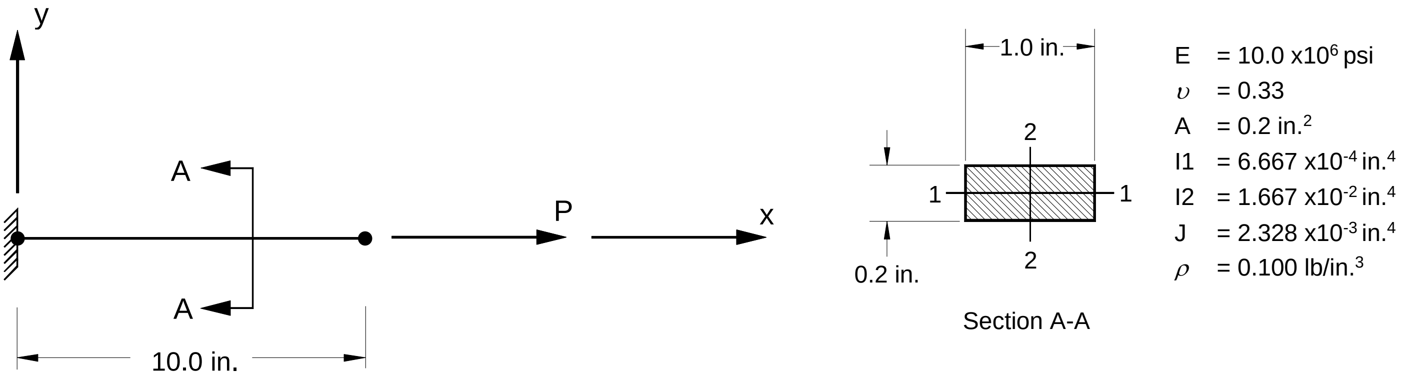


Figure 15-5. 2-D Cantilever Beam Example Problem with a Nonlinear Elastic Material.

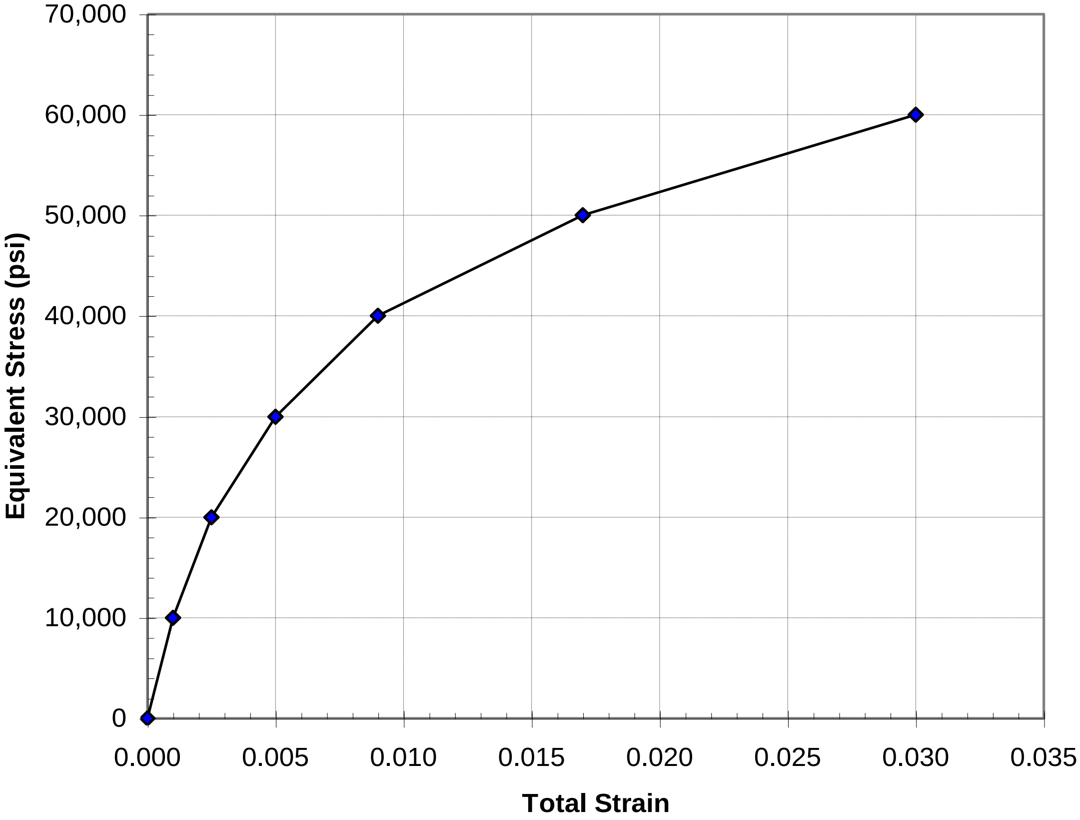


Figure 15-6. `TABLES1` Bulk Data Entry Equivalent Stress vs. Total Strain Input Data.

Listing 15-5. Model Input File for Cantilever Beam Problem with a Nonlinear Elastic Material.

```
$
$ NONLINEAR STATIC SOLUTION.
$
SOL NONLINEAR STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = AXIAL LOADED CANTILEVER BEAM WITH NONLINEAR ELASTIC MATERIAL
$
DISPLACEMENT = ALL
FORCE = ALL
STRESS = ALL
$
SUBCASE 1
 LABEL = TENSILE LOAD IN X-DIR
 LOAD = 1
 NLPARM = 1
 SPC = 1
$
BEGIN BULK
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  20,  ,  ,  ,  ,  ,  YES
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
$
$ BAR ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  1.667E-2,  6.667E-4,  2.328E-3,
,  -0.5,  0.1,  0.5,  0.1,  -0.5,  -0.1,  0.5,  -0.1
$
$ LINEAR ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ NONLINEAR-ELASTIC ELEMENT MATERIAL PROPERTIES.
$
MATS1,  100,  10,  NLELAST
$
$ STRESS/STRAIN DATA.
$
TABLES1,  10,
  ,  0.,  0.,  1.E-3,  1.E+4,  2.5E-3,  2.E+4,  5.E-3,  3.E+4,
  ,  9.E-3,  4.E+4,  1.7E-2,  5.E+4,  3.E-2,  6.E+4,  ENDT
```

Listing 15-5. Model Input File for the Cantilever Beam Problem with a Nonlinear Elastic Material. (Continued)

```
$
$ FIXED AT ONE END, FREE TO TRANSLATE IN X-DIR AT OTHER END.
$
SPC1,  1,  123456,   1
SPC1,  1,   23456,   2,  THRU,  11
$
$ TENSILE LOAD (X-DIRECTION).
$
FORCE,  1,  11,  0,  1.E+4,  1.,  0.,  0.
ENDDATA
```

The maximum displacements from the Model Results Output File are shown in tabular form in Listing 15-6 and graphically in Figure 15-7. Bar element equivalent stress is plotted against effective strain in Figure 15-8. Since this is a simple case of axial loading, the input and output stress-strain curves are the same.

Listing 15-6. Load Increment Maximum Displacements.

```
                                        M A X I M U M   D I S P L A C E M E N T S

      LOAD         LOAD              T1            T2            T3            R1            R2            R3
    INCREMENT     FACTOR
        1     5.000000E-02     2.500000E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        2     1.000000E-01     5.000000E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        3     1.500000E-01     7.500000E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        4     2.000000E-01     1.000000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        5     2.500000E-01     1.375000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        6     3.000000E-01     1.750000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        7     3.500000E-01     2.125000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        8     4.000000E-01     2.500000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        9     4.500000E-01     3.125000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       10     5.000000E-01     3.750000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       11     5.500000E-01     4.375000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       12     6.000000E-01     5.000000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       13     6.500000E-01     6.000000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       14     7.000000E-01     7.000000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       15     7.500000E-01     8.000000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       16     8.000000E-01     9.000000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       17     8.500000E-01     1.100000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       18     9.000000E-01     1.300000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       19     9.500000E-01     1.500000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       20     1.000000E+00     1.700000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
```

Figure 15-7. Maximum Displacement vs. Load Factor.

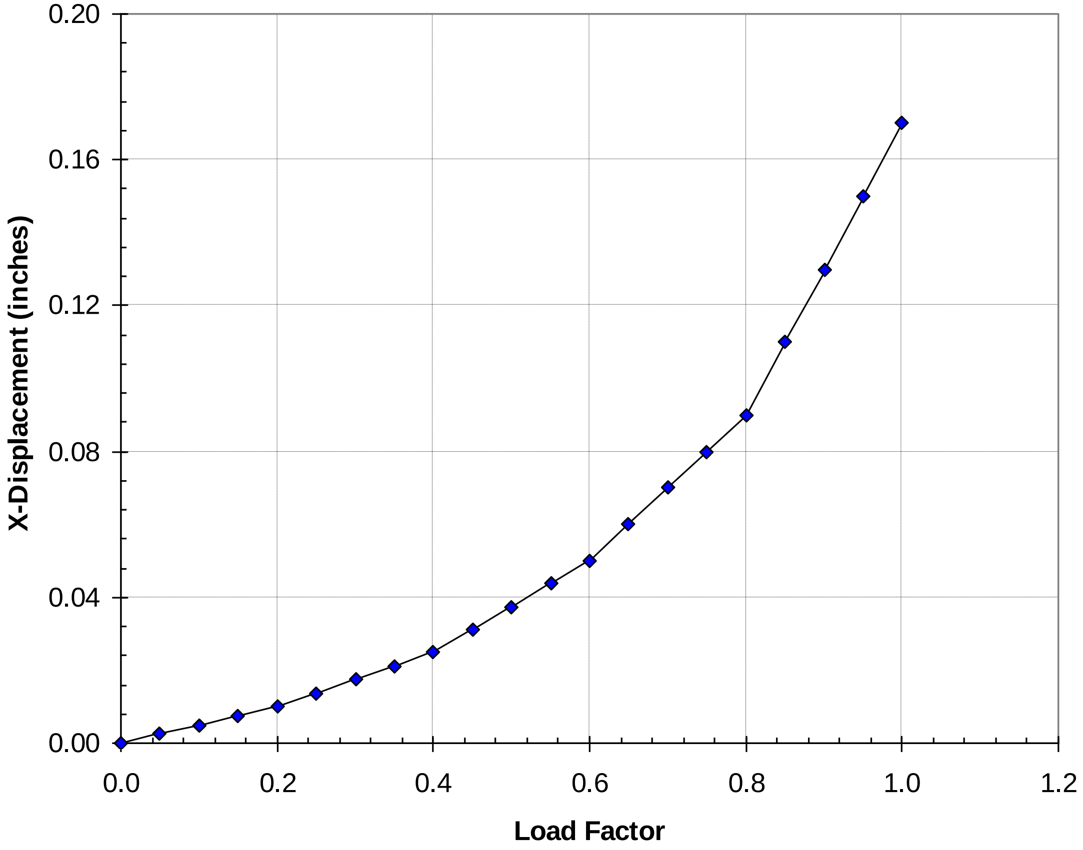
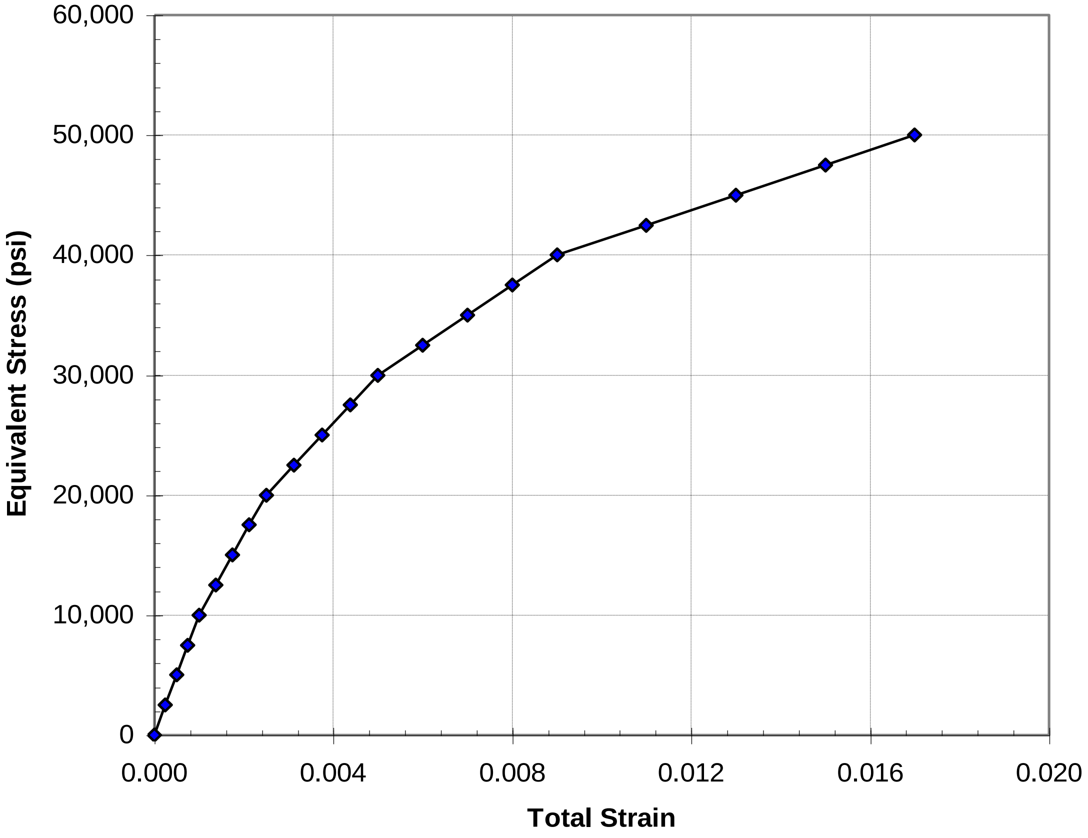


Figure 15-8. Bar Element Equivalent Stress vs. Total Strain.



15.3.4 Thermal-Elastic Material

The next problem is an example thermal-elastic material nonlinearity. The beam in Figure 15-9 is made from a temperature dependent material and subjected to a 1000 pound axial force at initial temperature of 100 °F. The temperature then increased to 200 °F while the load is held constant. The temperature increment is divided into 10 increments.

There are two methods for analyzing this problem. Both methods yield the same results as will be shown. The material behavior is temperature dependent and represented graphically in Figure 15-10. In the first method, the temperature dependence is input using `MATS1`, `TABLES1`, and `TABLEST` Bulk Data entries. The `MATS1` Bulk Data entry must reference an isotropic material. Listing 15-7 contains the Model Input File.

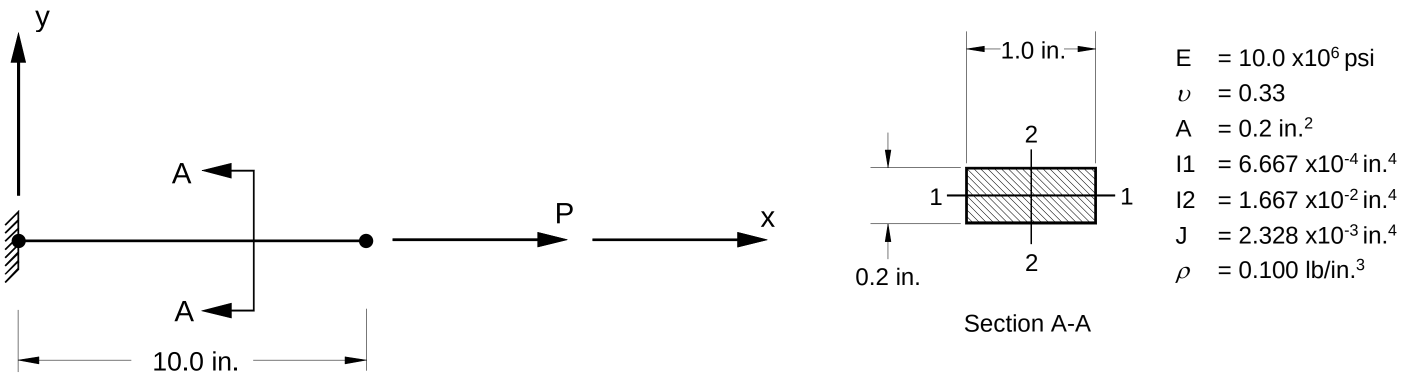


Figure 15-9. 2-D Cantilever Beam Example Problem with a Temperature Dependent Material.

Figure 15-10a. `TABLES1` Bulk Data Entry Equivalent Stress vs. Total Strain Input Data at 100 °F.

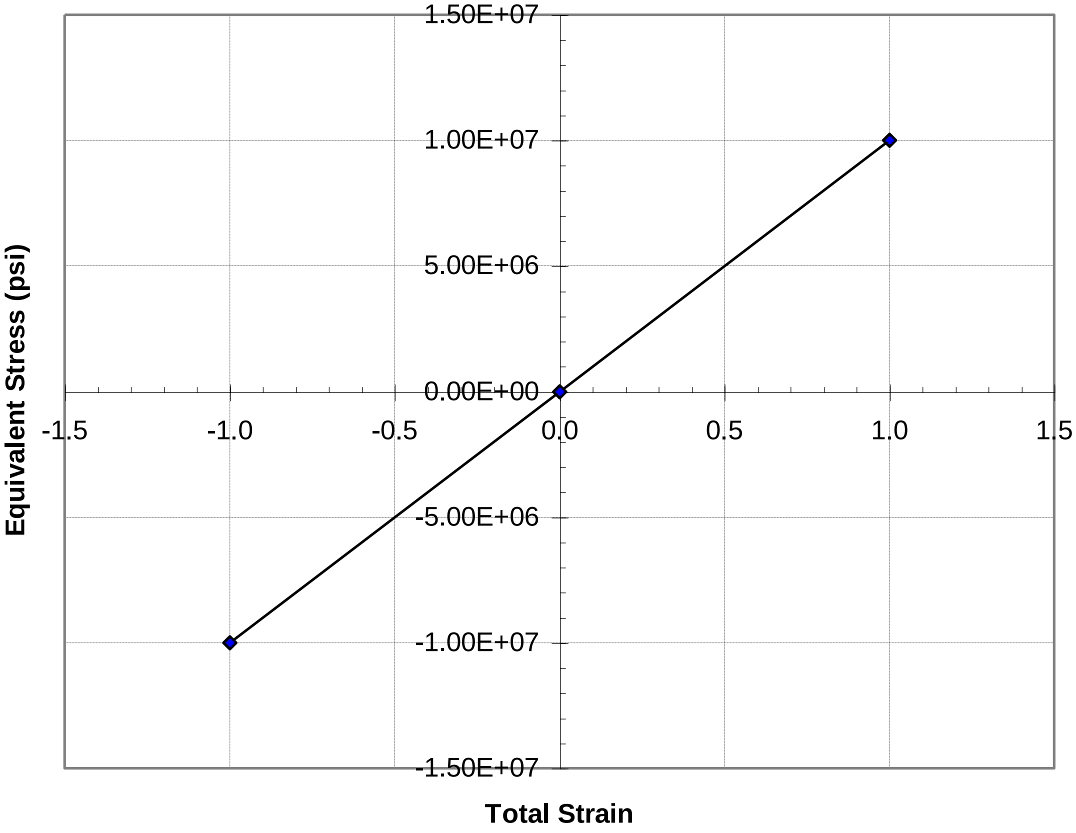
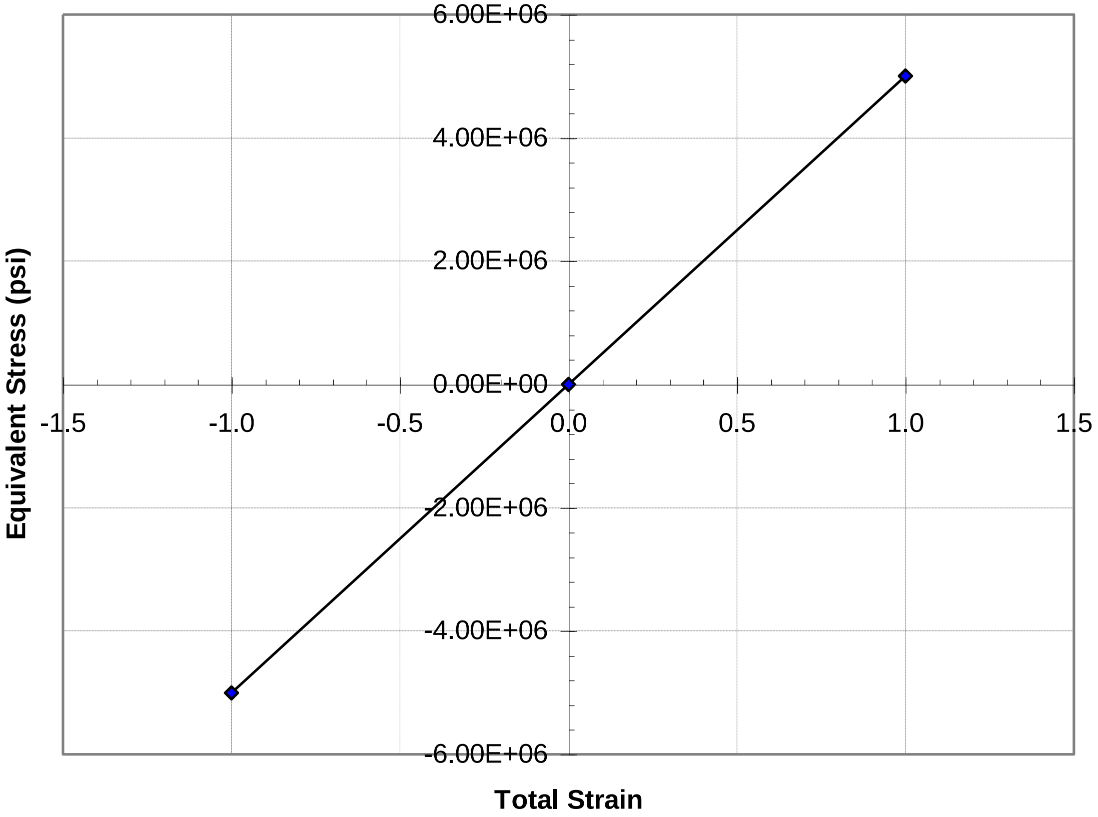


Figure 15-10b. `TABLES1` **Bulk Data Entry Equivalent Stress vs. Total Strain Input Data at 200 °F.**



Listing 15-7. Model Input File for the Cantilever Beam Problem with a Temperature Dependent Material.

```
$
$ NONLINEAR STATIC SOLUTION.
$
SOL NONLINEAR STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = AXIAL LOADED CANTILEVER BEAM WITH TEMPERATURE DEPENDENT MATERIAL
$
DISPLACEMENT = ALL
FORCE = ALL
STRESS = ALL
$
SPC = 1
TEMPERATURE(INITIAL) = 100
SUBCASE 1
 LABEL = TENSILE LOAD IN X-DIR, TEMPERATURE AT 100 DEG.
 LOAD = 1
 NLPARM = 1
SUBCASE 2
 LABEL = TENSILE LOAD IN X-DIR, TEMPERATURE AT 200 DEG.
 TEMPERATURE(MATERIAL) = 200
 LOAD = 1
 NLPARM = 2
$
BEGIN BULK
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,   1,  ,  ,  ,  ,  ,  YES
NLPARM,  2,  10,  ,  ,  ,  ,  ,  YES
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
$
$ BAR ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  1.667E-2,  6.667E-4,  2.328E-3,
,  -0.5,  0.1,  0.5,  0.1,  -0.5,  -0.1,  0.5,  -0.1
$
$ LINEAR ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
```

Listing 15-7. Model Input File for the Cantilever Beam Problem with a Temperature Dependent Material. (Continued)

```
$
$ NONLINEAR-ELASTIC ELEMENT MATERIAL PROPERTIES.
$
MATS1,  100,  10,  NLELAST
$
$ TEMPERATURE DEPENDENT STRESS/STRAIN DATA.
$
TABLEST,  10,
,  0.,  20,  100.,  30,  200.,  40,  ENDT
TABLES1,  20,
, -1., -1.E+7,  0.,  0.,  1.,  1.E+7,  ENDT
TABLES1,  30,
, -1., -1.E+7,  0.,  0.,  1.,  1.E+7,  ENDT
TABLES1,  40,
, -1., -5.E+6,  0.,  0.,  1.,  5.E+6,  ENDT
$
$ FIXED AT ONE END, FREE TO TRANSLATE IN X-DIR AT OTHER END.
$
SPC1,  1,  123456,   1
SPC1,  1,   23456,   2,  THRU,  11
$
$ TENSILE LOAD (X-DIRECTION).
$
FORCE,  1,  11,  0,  1.E+4,  1.,  0.,  0.
$
$ UNIFORM TEMPERATURE.
$
TEMPD,  100,  100.
TEMPD,  200,  200.
ENDDATA
```

The maximum displacements from the Model Results Output File are shown in tabular form in Listing 15-8 and graphically in Figure 15-11. Bar element total strain is plotted against load factor in Figure 15-12. Note that for subcase 2, a load factor of 0.0 corresponds to a temperature of 100 °F and at 1.0, a temperature of 200 °F.

Listing 15-8. Subcase 2 Load Increment Maximum Displacements.

```
                                        M A X I M U M   D I S P L A C E M E N T S

     LOAD         LOAD             T1            T2            T3            R1            R2            R3
  INCREMENT      FACTOR
        1     1.000000E-01     5.263125E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        2     2.000000E-01     5.554209E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        3     3.000000E-01     5.881059E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        4     4.000000E-01     6.247772E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        5     5.000000E-01     6.664291E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        6     6.000000E-01     7.140140E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        7     7.000000E-01     7.688979E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        8     8.000000E-01     8.330279E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        9     9.000000E-01     9.086483E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       10     1.000000E+00     9.994857E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
```

Figure 15-11. Maximum Displacement vs. Load Factor.

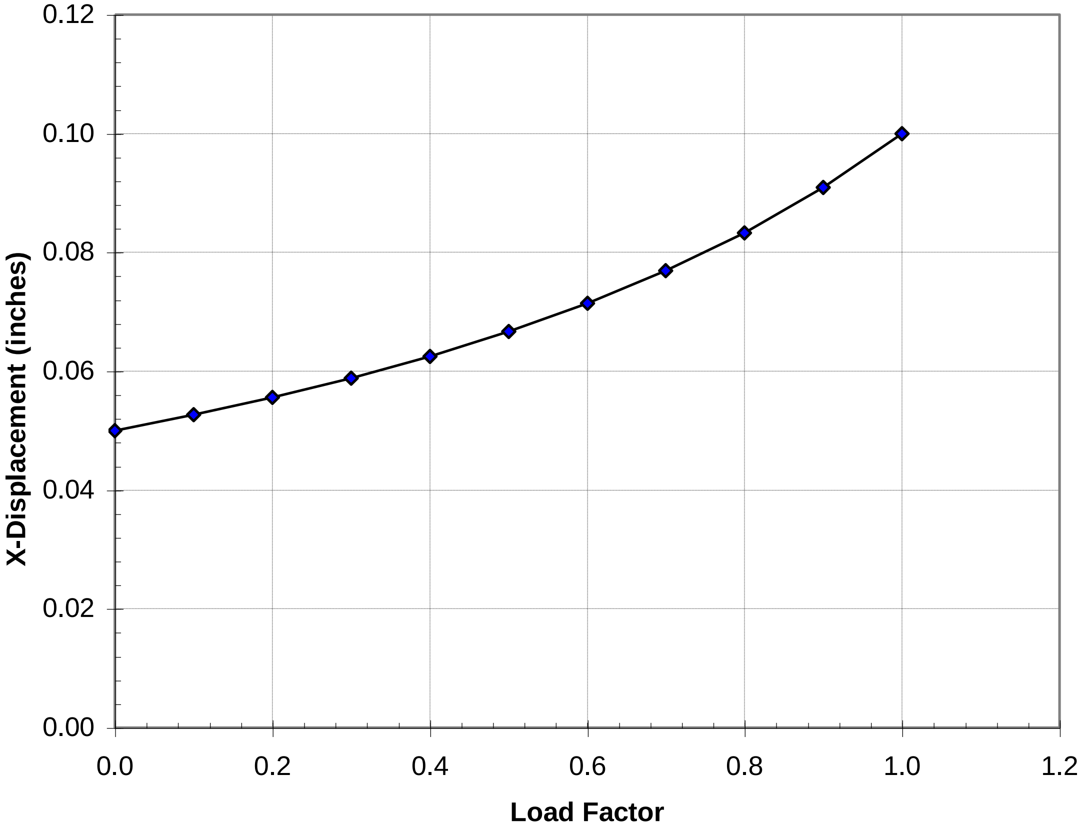
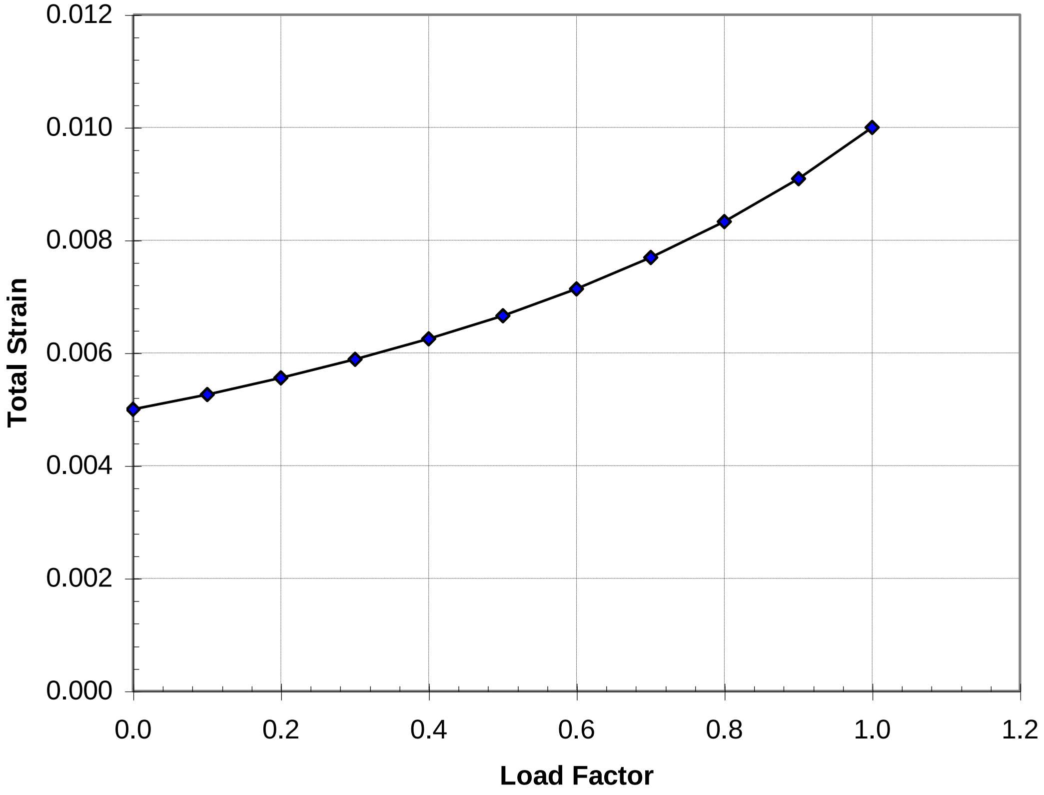


Figure 15-12. Bar Element Total Strain vs. Load Factor.



In the second method, the temperature dependence is input using `MATT1` and `TABLEM2` Bulk Data entries. The `MATT1` Bulk Data entry must reference an isotropic material. Listing 15-9 contains the Model Input File.

Listing 15-9. Model Input File for the Cantilever Beam Problem with a Temperature Dependent Material –Alternate Method.

```
$
$ NONLINEAR STATIC SOLUTION.
$
SOL NONLINEAR STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = AXIAL LOADED CANTILEVER BEAM WITH TEMPERATURE DEPENDENT MATERIAL
$
DISPLACEMENT = ALL
FORCE = ALL
STRESS = ALL
$
SPC = 1
TEMPERATURE(INITIAL) = 100
SUBCASE 1
 LABEL = TENSILE LOAD IN X-DIR, TEMPERATURE AT 100 DEG.
 LOAD = 1
 NLPARM = 1
SUBCASE 2
 LABEL = TENSILE LOAD IN X-DIR, TEMPERATURE AT 200 DEG.
 TEMPERATURE(MATERIAL) = 200
 LOAD = 1
 NLPARM = 2
$
BEGIN BULK
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,   1,  ,  ,  ,  ,  ,  YES
NLPARM,  2,  10,  ,  ,  ,  ,  ,  YES
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
```

Listing 15-9. Model Input File for the Cantilever Beam Problem with a Temperature Dependent Material –Alternate Method. (Continued)

```
$
$ BAR ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  1.667E-2,  6.667E-4,  2.328E-3,
,  -0.5,  0.1,  0.5,  0.1,  -0.5,  -0.1,  0.5,  -0.1
$
$ LINEAR ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ MATERIAL TEMPERATURE DEPENDENCE.
$
MATT1,  100,  10,  10
$
TABLEM2,  10,
,  0.,  1.,  100.,  1.,  200.,  0.5,  ENDT
$
$ FIXED AT ONE END, FREE TO TRANSLATE IN X-DIR AT OTHER END.
$
SPC1,  1,  123456,   1
SPC1,  1,   23456,   2,  THRU,  11
$
$ TENSILE LOAD (X-DIRECTION).
$
FORCE,  1,  11,  0,  1.E+4,  1.,  0.,  0.
$
$ UNIFORM TEMPERATURE.
$
TEMPD,  100,  100.
TEMPD,  200,  200.
ENDDATA
```

The maximum displacements from the Model Results Output File are shown in tabular form in Listing 15-10 and graphically in Figure 15-13. Bar element total strain is plotted against load factor in Figure 15-14. Note that both methods yield the same results.

Listing 15-10. Subcase 2 Load Increment Maximum Displacements.

```
                                      M A X I M U M   D I S P L A C E M E N T S

     LOAD        LOAD            T1             T2             T3             R1             R2             R3
   INCREMENT    FACTOR
      1     1.000000E-01     5.263125E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      2     2.000000E-01     5.554209E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      3     3.000000E-01     5.881059E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      4     4.000000E-01     6.247772E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      5     5.000000E-01     6.664291E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      6     6.000000E-01     7.140140E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      7     7.000000E-01     7.688979E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      8     8.000000E-01     8.330279E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      9     9.000000E-01     9.086483E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
     10     1.000000E+00     9.994857E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
```

Figure 15-13. Maximum Displacement vs. Load Factor.

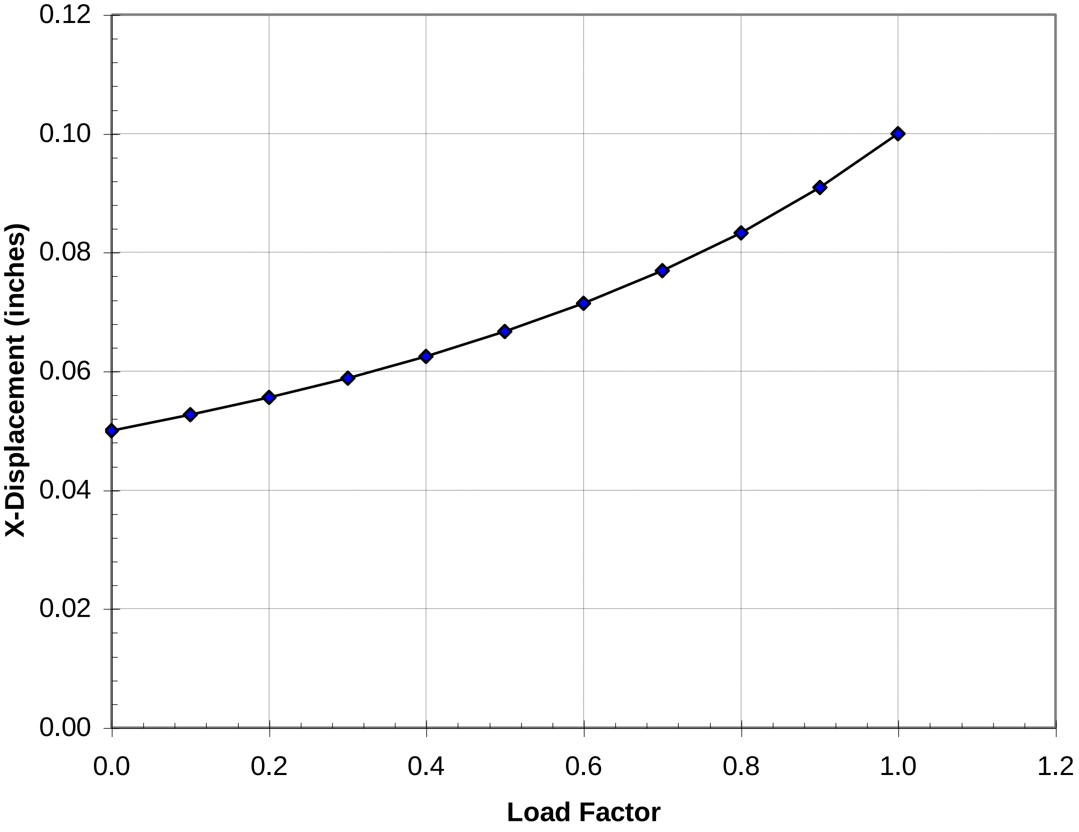
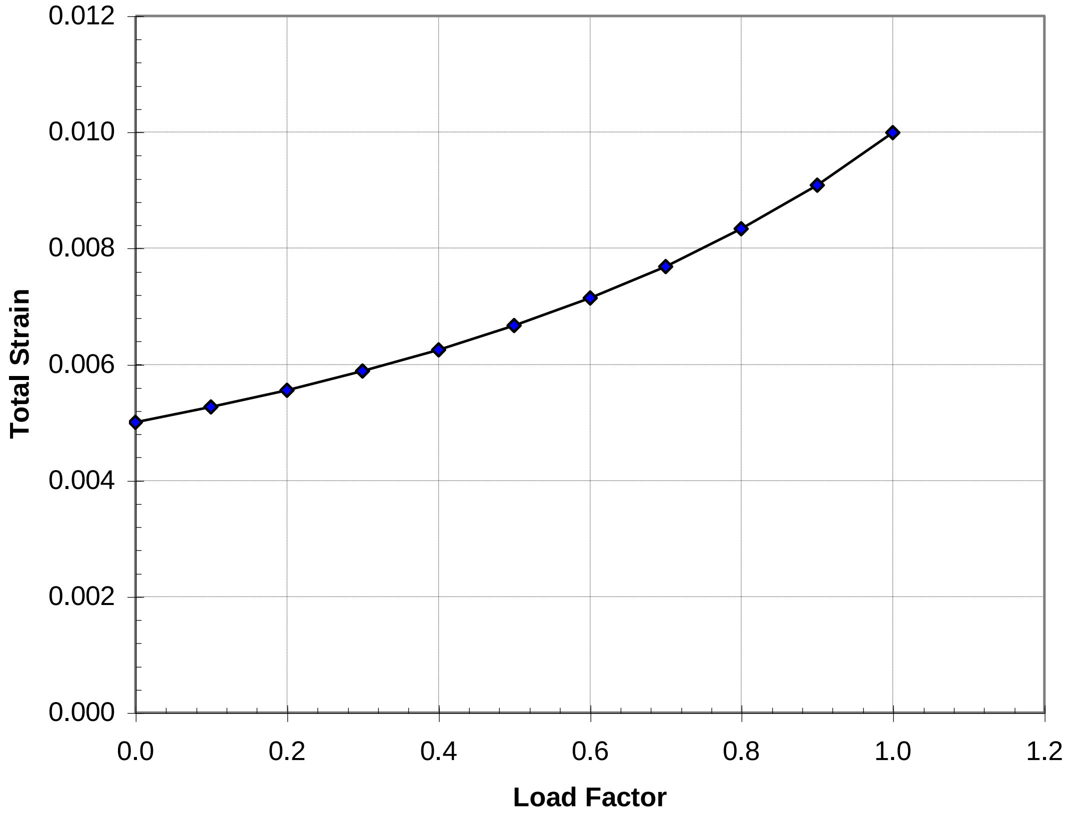


Figure 15-14. Bar Element Total Strain vs. Load Factor.



15.3.5 Elastic-Plastic Material

The next problem is another example of material nonlinearity. The beam in Figure 15-15 is made from an elastic-plastic material and subjected to a 1000 pound axial force. The load is gradually increased in subcase 1 to its full value, then decreased in subcase 2 to zero. The loading is divided into 20 increments for each subcase. The `W` option of `CONV` on the `NLPARM` Bulk Data entry is selected specifying the work convergence criteria (default is load and work convergence criteria).

The material behavior is stress dependent and represented graphically in Figure 15-16. The initial yield point (Y_1) and plasticity modulus (H) are input using the `MATS1` Bulk Data entry. In this example, the initial yield point of the material is 3.0E+4 psi and the plasticity modulus is 1.0E+6 psi. The plasticity modulus is related to the tangent modulus, E_T, by:

$$H = \frac{E_T}{1 - \frac{E_T}{E}}$$

Listing 15-11 contains the Model Input File.

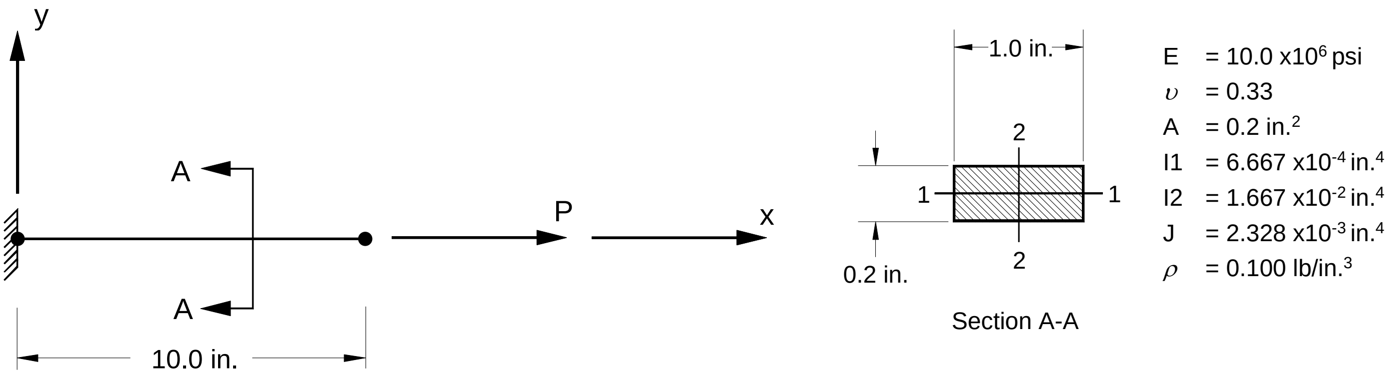
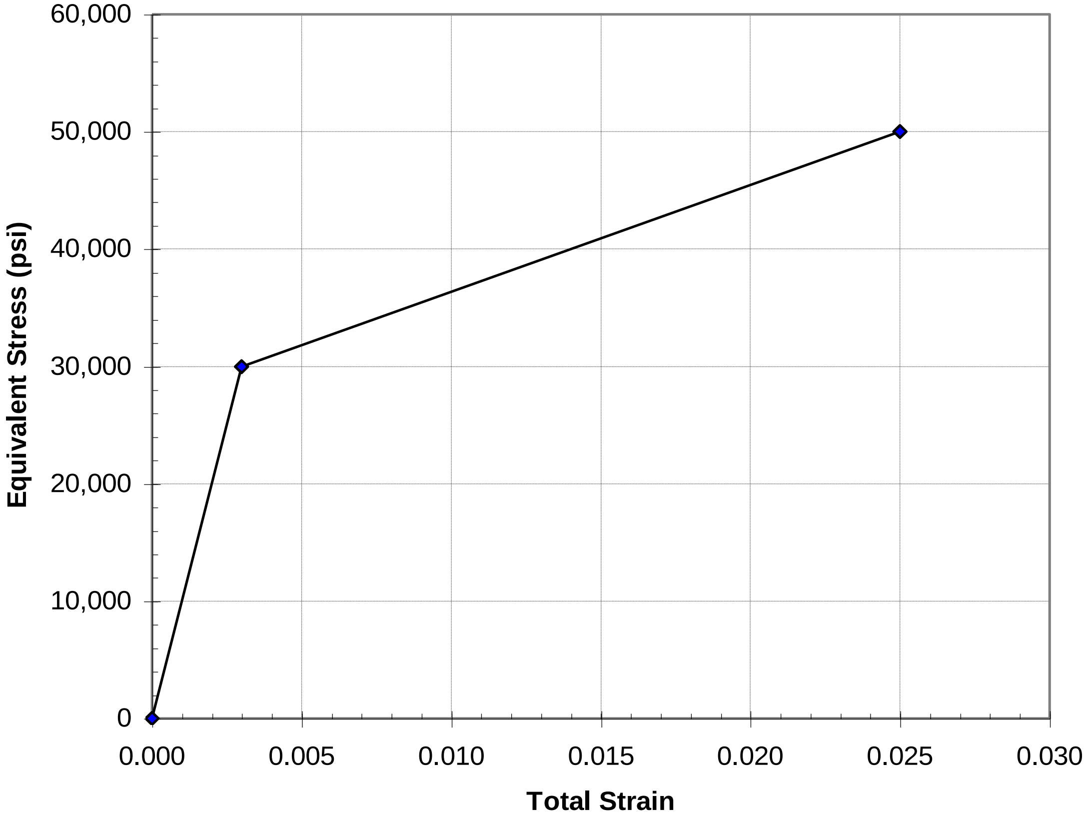


Figure 15-15. 2-D Cantilever Beam Example Problem with an Elastic-Plastic Material.

Figure 15-16. `MATS1` **Bulk Data Entry Equivalent Stress vs. Total Strain Input Data.**



Listing 15-11. Model Input File for the Cantilever Beam Problem with an Elastic-Plastic Material.

```
$
$ NONLINEAR STATIC SOLUTION.
$
SOL NONLINEAR STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = AXIAL LOADED CANTILEVER BEAM WITH ELASTIC-PLASTIC MATERIAL
$
DISPLACEMENT = ALL
FORCE = ALL
STRESS = ALL
$
SUBCASE 1
 LABEL = TENSILE LOAD IN X-DIR
 LOAD = 1
 NLPARM = 1
 SPC = 1
SUBCASE 2
 LABEL = UNLOAD
 NLPARM = 1
 SPC = 1
$
BEGIN BULK
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  20,  ,  ,  ,  ,  W,  YES
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
$
$ BAR ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  1.667E-2,  6.667E-4,  2.328E-3,
,  -0.5,  0.1,  0.5,  0.1,  -0.5,  -0.1,  0.5,  -0.1
$
$ LINEAR ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ ELASTIC-PLASTIC ELEMENT MATERIAL PROPERTIES.
$
MATS1,  100,  ,  PLASTIC,  1.E+6,  ,  ,  3.0E+4
```

Listing 15-11. Model Input File for the Cantilever Beam Problem with an Elastic-Plastic Material. (Continued)

```
$
$ FIXED AT ONE END, FREE TO TRANSLATE IN X-DIR AT OTHER END.
$
SPC1,  1,  123456,   1
SPC1,  1,   23456,   2,  THRU,  11
$
$ TENSILE LOAD (X-DIRECTION).
$
FORCE,  1,  11,  0,  1.E+4,  1.,  0.,  0.
ENDDATA
```

The maximum displacements from the Model Results Output File are shown in tabular form in Listing 15-12 and graphically in Figure 15-17. Bar element equivalent stress is plotted against total strain in Figure 15-18 and against effective plastic strain in Figure 15-19. A major difference between a nonlinear elastic and elastic-plastic material is that a nonlinear elastic material does not accumulate effective plastic strain, and when unloaded, will return to its initial configuration. In this example, the material is loaded past the initial yield point and well into the plastic range of the material. Plastic strain is accumulated and a residual strain of 2.0E-2 inch/inch exists after the beam is unloaded.

Listing 15-12. Load Increment Maximum Displacements.

```
 TENSILE LOAD IN X-DIR                                               SUBCASE 1

                                   M A X I M U M   D I S P L A C E M E N T S

     LOAD         LOAD              T1            T2            T3            R1            R2            R3
   INCREMENT     FACTOR
       1      5.000000E-02     2.500000E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       2      1.000000E-01     5.000000E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       3      1.500000E-01     7.500000E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       4      2.000000E-01     1.000000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       5      2.500000E-01     1.250000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       6      3.000000E-01     1.500000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       7      3.500000E-01     1.750000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       8      4.000000E-01     2.000000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       9      4.500000E-01     2.250000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      10      5.000000E-01     2.500000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      11      5.500000E-01     2.750000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      12      6.000000E-01     3.000000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      13      6.500000E-01     5.750000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      14      7.000000E-01     8.500000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      15      7.500000E-01     1.125000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      16      8.000000E-01     1.400000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      17      8.500000E-01     1.675000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      18      9.000000E-01     1.950000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      19      9.500000E-01     2.225000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      20      1.000000E+00     2.500000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00

 UNLOAD                                                              SUBCASE 2

                                   M A X I M U M   D I S P L A C E M E N T S

     LOAD         LOAD              T1            T2            T3            R1            R2            R3
   INCREMENT     FACTOR
       1      9.500000E-01     2.475000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       2      9.000000E-01     2.450000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       3      8.500000E-01     2.425000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       4      8.000000E-01     2.400000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       5      7.500000E-01     2.375000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       6      7.000000E-01     2.350000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       7      6.500000E-01     2.325000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       8      6.000000E-01     2.300000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       9      5.500000E-01     2.275000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      10      5.000000E-01     2.250000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      11      4.500000E-01     2.225000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      12      4.000000E-01     2.200000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      13      3.500000E-01     2.175000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      14      3.000000E-01     2.150000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      15      2.500000E-01     2.125000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      16      2.000000E-01     2.100000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      17      1.500000E-01     2.075000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      18      1.000000E-01     2.050000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      19      5.000000E-02     2.025000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      20      0.000000E-00     2.000000E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
```

Figure 15-17. Maximum Displacement vs. Load Factor.

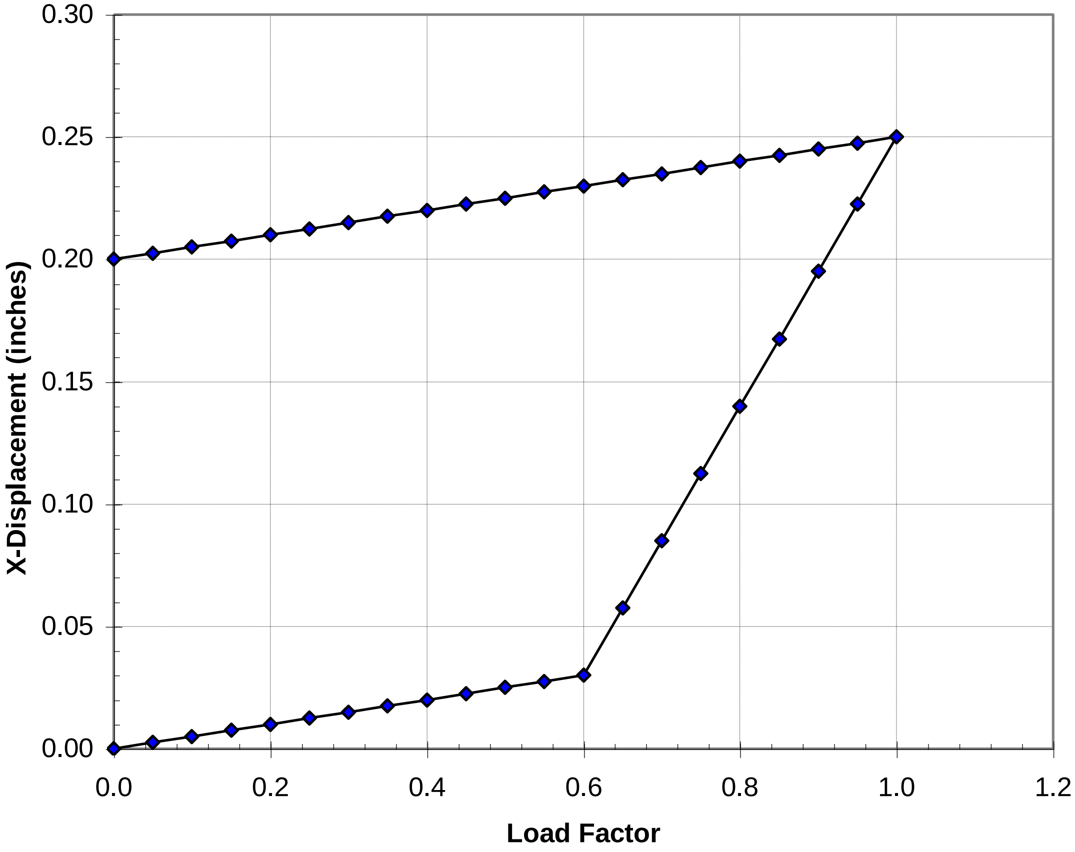


Figure 15-18. Bar Element Equivalent Stress vs. Total Strain.

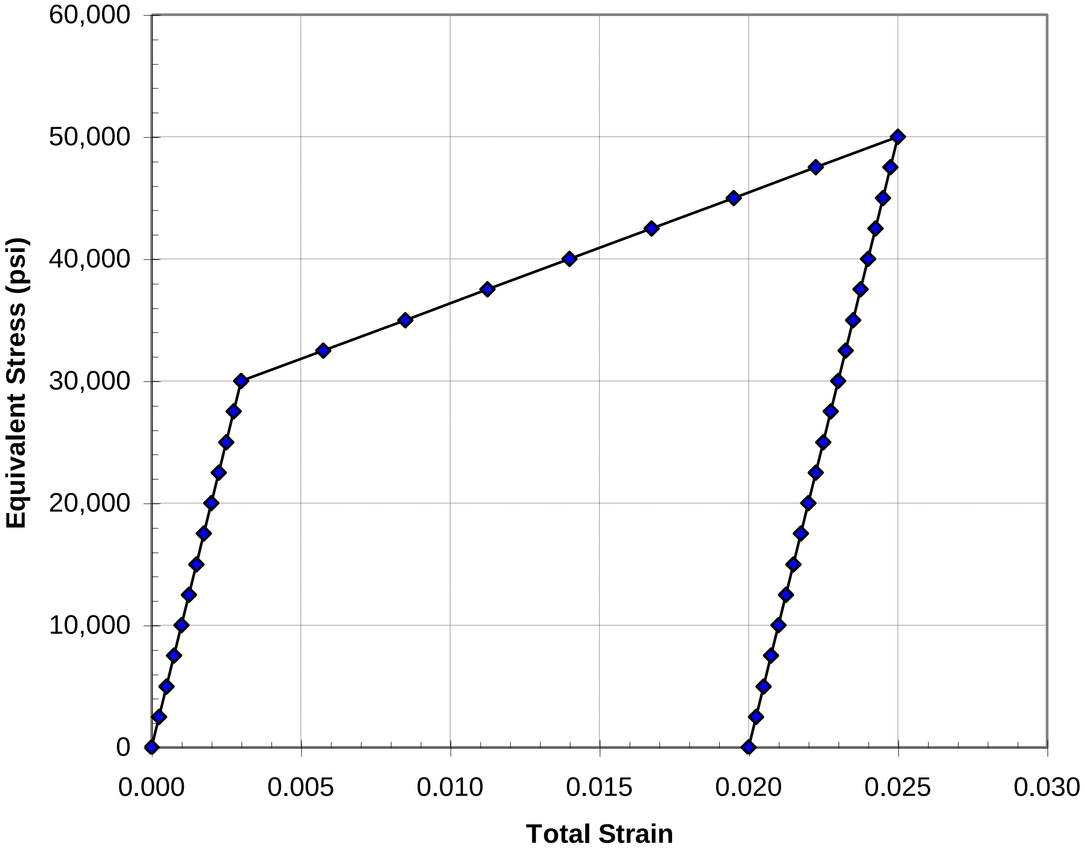
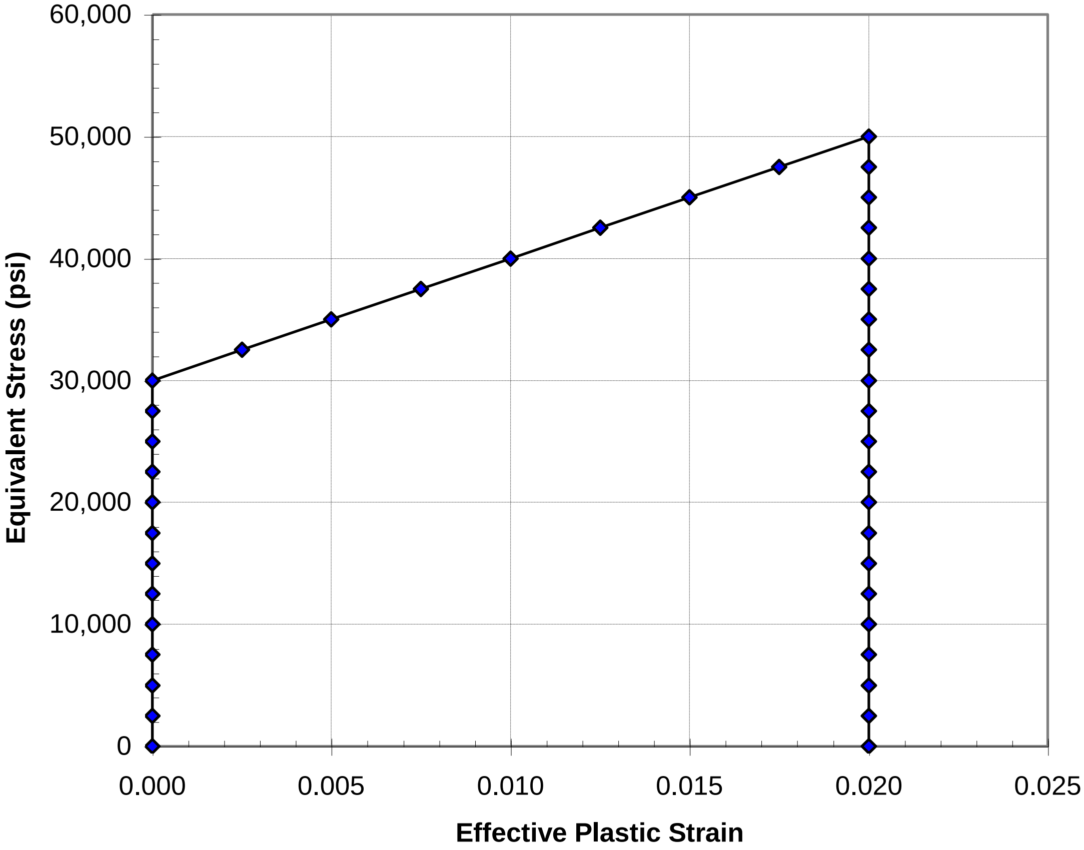


Figure 15-19. Bar Element Equivalent Stress vs. Effective Plastic Strain.



15.3.6 Combined Large Displacement and Nonlinear-Elastic Material

The next problem is an example of combined geometric and material nonlinearity. Figure 15-20 shows a cable which is clamped (fixed) at one end and attached with a pulley (free to translate in the x-direction) at the other end. In subcase 1, the cable is initially lose (modeled using nonlinear elastic material nonlinearity) and is tighten by the addition of a tensile load at the pulley. After the slack has been removed, subcase 2 through subcase 4 gradually apply a mid-span load causing the cable to sag a large amount. The sagging cable is then tightened again in subcase 5 with the increase of the tensile load at the pulley and the cable straightens out again (not fully though). Note that the subcase structure is used to initially add a very small amount of shear load, which is then gradually increased thus preventing divergence and increasing solution efficiency. Listing 15-13 contains the Model Input File.

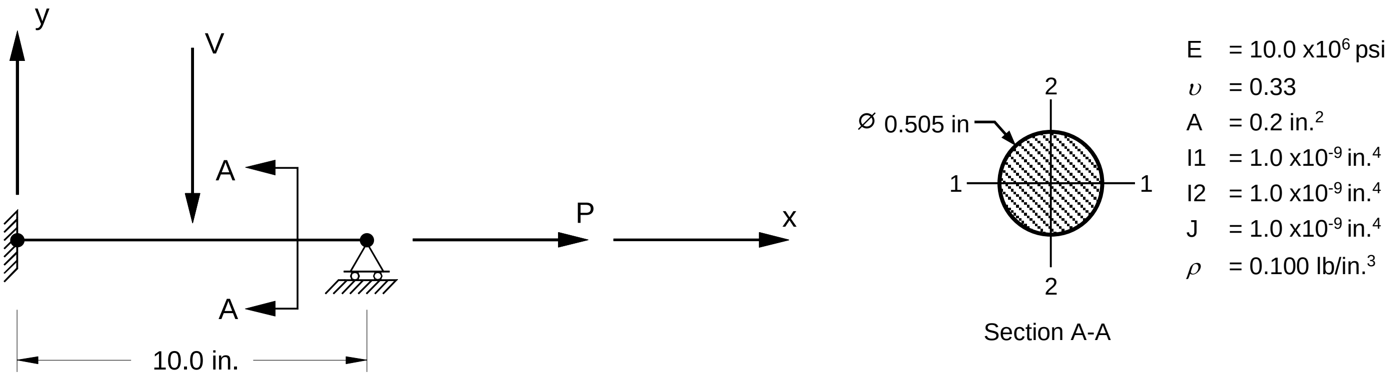


Figure 15-20. 2-D Cable Example Problem.

Listing 15-13. Model Input File for the Cable Example Problem.

```
$
$ NONLINEAR STATIC SOLUTION.
$
SOL NONLINEAR STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = SLACK CABLE LOADED OUT OF PLANE THEN PULLED TAUGHT
$
DISPLACEMENT = ALL
FORCE = ALL
STRESS = ALL
$
SPC = 1
SUBCASE 1
 LABEL = INITIAL AXIAL LOAD IN CABLE ADDED WITH P AT 10% -SLACK IS REMOVED
 LOAD = 10
 NLPARM = 1
SUBCASE 2
 LABEL = MID-SPAN LOAD ADDED AT 0.1% OF V, AXIAL LOAD AT 10% OF P
 LOAD = 20
 NLPARM = 1
SUBCASE 3
 LABEL = MID-SPAN LOAD AT 1% OF V, AXIAL LOAD AT 10% OF P
 LOAD = 30
 NLPARM = 1
SUBCASE 4
 LABEL = MID-SPAN LOAD AT 100% OF V, AXIAL LOAD AT 10% OF P
 LOAD = 40
 NLPARM = 1
SUBCASE 5
 LABEL = MID-SPAN LOAD AT 100% OF V, AXIAL LOAD AT 100% OF P
 LOAD = 50
 NLPARM = 1
$
BEGIN BULK
$
$ TURN ON LARGE DISPLACEMENT EFFECTS.
$
PARAM, LGDISP, ON
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  10,  ,  ,  ,  50,  P,  YES
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ CABLE MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES.
$
PBAR,  10,  100,  0.20,  1.E-9,  1.E-9,  1.E-9
```

Listing 15-13. Model Input File for the Cable Example Problem. (Continued)

```
$
$ LINEAR ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ NONLINEAR-ELASTIC ELEMENT MATERIAL PROPERTIES.
$
MATS1,  100,  10,  NLELAST
$
$ STRESS/STRAIN DATA.
$
TABLES1,  10,
  ,  0.,  0.,  1.E-3,  1.E+3,  1.,  1.E+7,  ENDT
$
$ FIXED AT BOTH ENDS -ONE END FREE TO TRANSLATE IN X-DIR.
$
SPC1,  1,  123456,   1
SPC1,  1,   23456,  11
SPC1,  1,     345,   1,  THRU,  11
$
$ AXIAL TENSILE LOAD (X-DIRECTION).
$
FORCE,  1,  11,  0,  1.E+5,  1.,   0.,  0.
$
$ MID-SPAN OUT OF PLANE LOAD (Y-DIRECTION).
$
FORCE,  2,   6,  0,  5.E+3,  0.,  -1.,  0.
$
$ SCALE TOTAL LOADS TO GET SUBCASE LOADING.
$
LOAD,  10,  1.,  0.,     2,  0.1,  1
LOAD,  20,  1.,  0.001,  2,  0.1,  1
LOAD,  30,  1.,  0.01,   2,  0.1,  1
LOAD,  40,  1.,  1.,     2,  0.1,  1
LOAD,  50,  1.,  1.,     2,  1.,   1
ENDDATA
```

The cable is modeled using a nonlinear elastic material, which has very little stiffness until a stress level of 1000 psi is reached (slack removed). The cable moments of inertia are made fictitiously small to represent an actual cable. While great amount of bending does occur, the initial bending stiffness is primary due to the differential stiffness contribution generated from the initial prestress in the cable (subcase 1). The maximum displacements from the Model Results Output File are shown in tabular form in Listing 15-14. The deflected shapes are plotted in Figure 15-21.

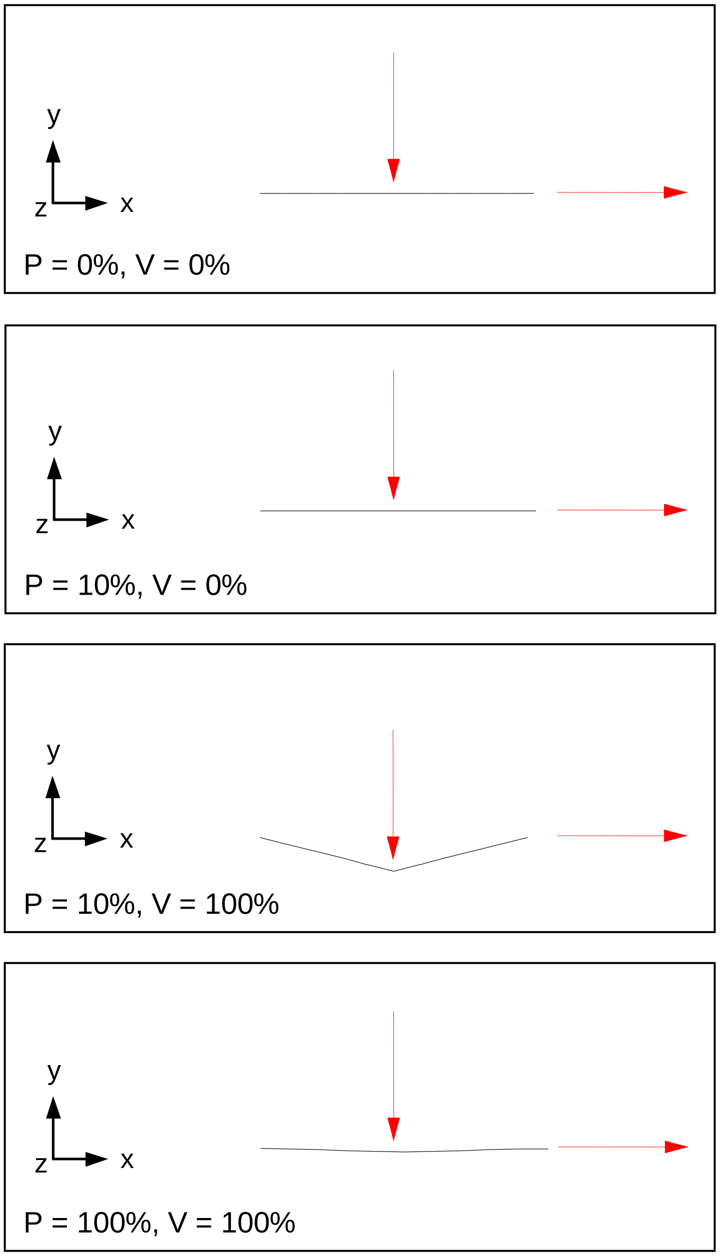
Listing 15-14. Subcase and Load Increment Maximum Displacements and Rotations.

```
  INITIAL AXIAL LOAD IN CABLE ADDED WITH P AT 10% -SLACK IS REMOVED  SUBCASE 1

                                   M A X I M U M   D I S P L A C E M E N T S

     LOAD        LOAD              T1            T2            T3            R1            R2            R3
   INCREMENT    FACTOR
       1     1.000000E-01     1.399680E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       2     2.000000E-01     1.899190E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       3     3.000000E-01     2.398740E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       4     4.000000E-01     2.898290E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       5     5.000000E-01     3.397840E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       6     6.000000E-01     3.897390E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       7     7.000000E-01     4.396940E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       8     8.000000E-01     4.896490E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       9     9.000000E-01     5.396040E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      10     1.000000E+00     5.895590E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
```

Listing 15-14. Subcase and Load Increment Maximum Displacements and Rotations. (Continued)

```
 MID-SPAN LOAD ADDED AT 0.1% OF V, AXIAL LOAD AT 10% OF P           SUBCASE 2

                                          M A X I M U M   D I S P L A C E M E N T S

     LOAD        LOAD              T1            T2            T3            R1            R2            R3
   INCREMENT    FACTOR
       1     1.000000E-01     5.895590E-02  1.191090E-04  0.000000E+00  0.000000E+00  0.000000E+00  2.413803E-05
       2     2.000000E-01     5.895589E-02  2.442823E-04  0.000000E+00  0.000000E+00  0.000000E+00  4.885105E-05
       3     3.000000E-01     5.895587E-02  3.699497E-04  0.000000E+00  0.000000E+00  0.000000E+00  7.354801E-05
       4     4.000000E-01     5.895585E-02  4.956765E-04  0.000000E+00  0.000000E+00  0.000000E+00  9.823953E-05
       5     5.000000E-01     5.895582E-02  6.214117E-04  0.000000E+00  0.000000E+00  0.000000E+00  1.229303E-04
       6     6.000000E-01     5.895579E-02  7.471482E-04  0.000000E+00  0.000000E+00  0.000000E+00  1.476210E-04
       7     7.000000E-01     5.895575E-02  8.728849E-04  0.000000E+00  0.000000E+00  0.000000E+00  1.723117E-04
       8     8.000000E-01     5.895570E-02  9.986216E-04  0.000000E+00  0.000000E+00  0.000000E+00  1.970026E-04
       9     9.000000E-01     5.895565E-02  1.124358E-03  0.000000E+00  0.000000E+00  0.000000E+00  2.216935E-04
      10     1.000000E+00     5.895559E-02  1.250095E-03  0.000000E+00  0.000000E+00  0.000000E+00  2.463845E-04

 MID-SPAN LOAD AT 1% OF V, AXIAL LOAD AT 10% OF P                   SUBCASE 3

                                          M A X I M U M   D I S P L A C E M E N T S

     LOAD        LOAD              T1            T2            T3            R1            R2            R3
   INCREMENT    FACTOR
       1     1.000000E-01     5.895505E-02  2.328704E-03  0.000000E+00  0.000000E+00  0.000000E+00  4.641798E-04
       2     2.000000E-01     5.895378E-02  3.455821E-03  0.000000E+00  0.000000E+00  0.000000E+00  6.865740E-04
       3     3.000000E-01     5.895199E-02  4.586895E-03  0.000000E+00  0.000000E+00  0.000000E+00  9.088407E-04
       4     4.000000E-01     5.894968E-02  5.718445E-03  0.000000E+00  0.000000E+00  0.000000E+00  1.131064E-03
       5     5.000000E-01     5.894687E-02  6.850061E-03  0.000000E+00  0.000000E+00  0.000000E+00  1.353281E-03
       6     6.000000E-01     5.894355E-02  7.981686E-03  0.000000E+00  0.000000E+00  0.000000E+00  1.575498E-03
       7     7.000000E-01     5.893972E-02  9.113311E-03  0.000000E+00  0.000000E+00  0.000000E+00  1.797715E-03
       8     8.000000E-01     5.893538E-02  1.024494E-02  0.000000E+00  0.000000E+00  0.000000E+00  2.019932E-03
       9     9.000000E-01     5.893054E-02  1.137656E-02  0.000000E+00  0.000000E+00  0.000000E+00  2.242150E-03
      10     1.000000E+00     5.892520E-02  1.250818E-02  0.000000E+00  0.000000E+00  0.000000E+00  2.464369E-03

 MID-SPAN LOAD AT 100% OF V, AXIAL LOAD AT 10% OF P                 SUBCASE 4

                                          M A X I M U M   D I S P L A C E M E N T S

     LOAD        LOAD              T1            T2            T3            R1            R2            R3
   INCREMENT    FACTOR
       1     1.000000E-01     5.524412E-02  1.369622E-01  0.000000E+00  0.000000E+00  0.000000E+00  2.690013E-02
       2     2.000000E-01     4.545541E-02  2.611414E-01  0.000000E+00  0.000000E+00  0.000000E+00  5.130230E-02
       3     3.000000E-01     2.961242E-02  3.848461E-01  0.000000E+00  0.000000E+00  0.000000E+00  7.564257E-02
       4     4.000000E-01     7.806385E-03  5.078039E-01  0.000000E+00  0.000000E+00  0.000000E+00  9.987711E-02
       5     5.000000E-01     1.985301E-02  6.299070E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.240078E-01
       6     6.000000E-01     5.321625E-02  7.508961E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.479919E-01
       7     7.000000E-01     9.210474E-02  8.706718E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.718358E-01
       8     8.000000E-01     1.363304E-01  9.888429E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.954441E-01
       9     9.000000E-01     1.856641E-01  1.105347E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.188300E-01
      10     1.000000E+00     2.398641E-01  1.220022E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.419702E-01

 MID-SPAN LOAD AT 100% OF V, AXIAL LOAD AT 100% OF P                SUBCASE 5

                                          M A X I M U M   D I S P L A C E M E N T S

     LOAD        LOAD              T1            T2            T3            R1            R2            R3
   INCREMENT    FACTOR
       1     1.000000E-01     1.839034E-02  6.590703E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.260254E-01
       2     2.000000E-01     1.091632E-01  4.515147E-01  0.000000E+00  0.000000E+00  0.000000E+00  8.447664E-02
       3     3.000000E-01     1.710144E-01  3.437879E-01  0.000000E+00  0.000000E+00  0.000000E+00  6.287747E-02
       4     4.000000E-01     2.241714E-01  2.773184E-01  0.000000E+00  0.000000E+00  0.000000E+00  4.946210E-02
       5     5.000000E-01     2.734531E-01  2.333803E-01  0.000000E+00  0.000000E+00  0.000000E+00  4.060874E-02
       6     6.000000E-01     3.210294E-01  2.018570E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.423515E-02
       7     7.000000E-01     3.680206E-01  1.763287E-01  0.000000E+00  0.000000E+00  0.000000E+00  2.886501E-02
       8     8.000000E-01     4.140753E-01  1.583103E-01  0.000000E+00  0.000000E+00  0.000000E+00  2.521486E-02
       9     9.000000E-01     4.598664E-01  1.435603E-01  0.000000E+00  0.000000E+00  0.000000E+00  2.220493E-02
      10     1.000000E+00     5.054500E-01  1.313985E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.971218E-02
```

Figure 15-21. Deformed Shapes of a Cable Under Multiple Loading.



15.3.7 Combined Gap Contact with Large Displacement and Rotation

The next problem is an example of contact with large displacement and rotation. The cantilever beam in Figure 15-22 is subjected to a shear load at its free end. The beam deflects normally until hitting a rigid support, which is modeled using a gap element. After contacting the support, the beam continues to deflect resulting in a reaction force in the gap element. Listing 15-15 contains the Model Input File.

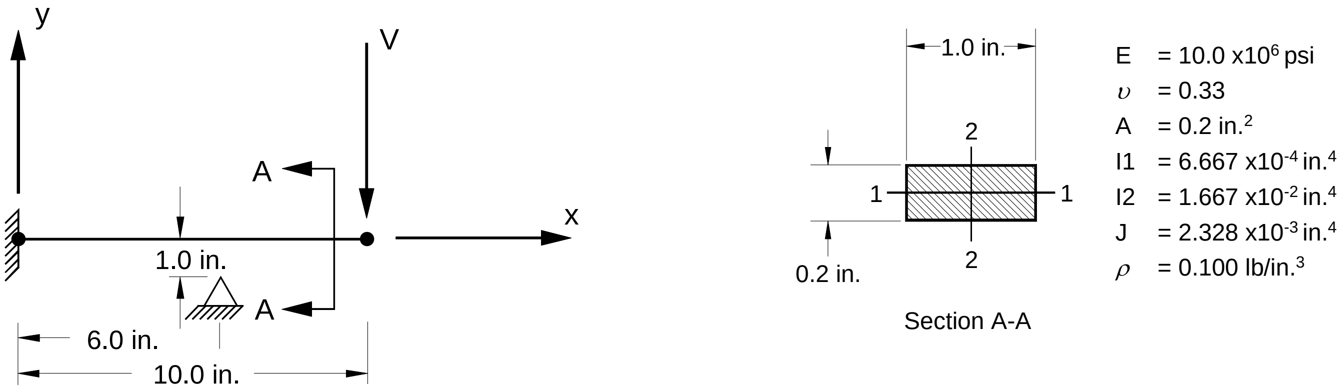


Figure 15-22. 2-D Cantilever Beam Example Problem with Contact.

Listing 15-15. Model Input File for the Cantilever Beam Problem with Contact.

```
$
$ NONLINEAR STATIC SOLUTION.
$
SOL NONLINEAR STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = SHEAR LOADED CANTILEVER BEAM WITH CONTACT
$
DISPLACEMENT = ALL
FORCE = ALL
STRESS = ALL
$
SUBCASE 1
 LABEL = POINT LOAD AT FREE END (SHEAR)
 LOAD = 1
 NLPARM = 1
 SPC = 1
$
BEGIN BULK
$
$ TURN ON LARGE DISPLACEMENT EFFECTS.
$
PARAM, LGDISP, ON
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  10,  ,  ,  ,  ,  P,  YES
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
GRID, 12,  0,   6., -1.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
$
$ GAP ELEMENT.
$
CGAP, 11,  20,   7,  12,  1.,  0.,  0.
$
$ BAR ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  1.667E-2,  6.667E-4,  2.328E-3,
,  -0.5,  -0.1,  0.5,  -0.1,  0.5,  0.1,  -0.5,  0.1
$
$ LINEAR ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ GAP ELEMENT PROPERTIES.
$
PGAP,  20,  1.,  0.,  1.E+7
```

Listing 15-15. Model Input File for the Cantilever Beam Problem with Contact. (Continued)

```
$
$ FIXED AT ONE END, MOVEMENT CONSTRAINED TO X-Y PLANE ONLY.
$
SPC1,  1,  123456,   1
SPC1,  1,     345,   2,  THRU,  11
SPC1,  1,  123456,  12
$
$ POINT LOAD AT FREE END (SHEAR).
$
FORCE,  1,  11,  0,  1.5E+2,  0., -1.,  0.
ENDDATA
```

The maximum displacements from the Model Results Output File are shown in tabular form in Listing 15-16. The gap axial force and deflection are plotted in Figures 15-23 and 15-24, respectively. The bar element stresses at the fixed end of the beam are plotted in Figure 15-25. The deflected shapes are plotted in Figure 15-26.

In this example, the gap element is defined with an initial opening of 1 inch, as specified on the `CGAP` Bulk Data entry. At a load factor of 40%, the beam has deflected 1 inch just above the contact point, the gap closes, and contact is made. At this point, axial load begins to develop in the gap element and a discontinuity occurs in the bar element stresses. Since no friction has been defined on the `PGAP` Bulk Data entry, no shear forces are developed when contact is made.

Listing 15-16. Load Increment Maximum Displacements and Rotations.

```
                                    M A X I M U M   D I S P L A C E M E N T S

     LOAD         LOAD              T1            T2            T3            R1            R2            R3
  INCREMENT     FACTOR
       1     1.000000E-01     3.334812E-02  7.456519E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.119740E-01
       2     2.000000E-01     1.297465E-01  1.466728E+00  0.000000E+00  0.000000E+00  0.000000E+00  2.209795E-01
       3     3.000000E-01     2.794321E-01  2.143234E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.245490E-01
       4     4.000000E-01     3.464905E-01  2.362401E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.700575E-01
       5     5.000000E-01     3.809492E-01  2.448332E+00  0.000000E+00  0.000000E+00  0.000000E+00  3.983580E-01
       6     6.000000E-01     4.168033E-01  2.531740E+00  0.000000E+00  0.000000E+00  0.000000E+00  4.260638E-01
       7     7.000000E-01     4.538512E-01  2.612513E+00  0.000000E+00  0.000000E+00  0.000000E+00  4.531351E-01
       8     8.000000E-01     4.919300E-01  2.690641E+00  0.000000E+00  0.000000E+00  0.000000E+00  4.795643E-01
       9     9.000000E-01     5.308484E-01  2.766058E+00  0.000000E+00  0.000000E+00  0.000000E+00  5.053217E-01
      10     1.000000E+00     5.704526E-01  2.838782E+00  0.000000E+00  0.000000E+00  0.000000E+00  5.304044E-01
```

Figure 15-23. Gap Element Axial Force vs. Load Factor.

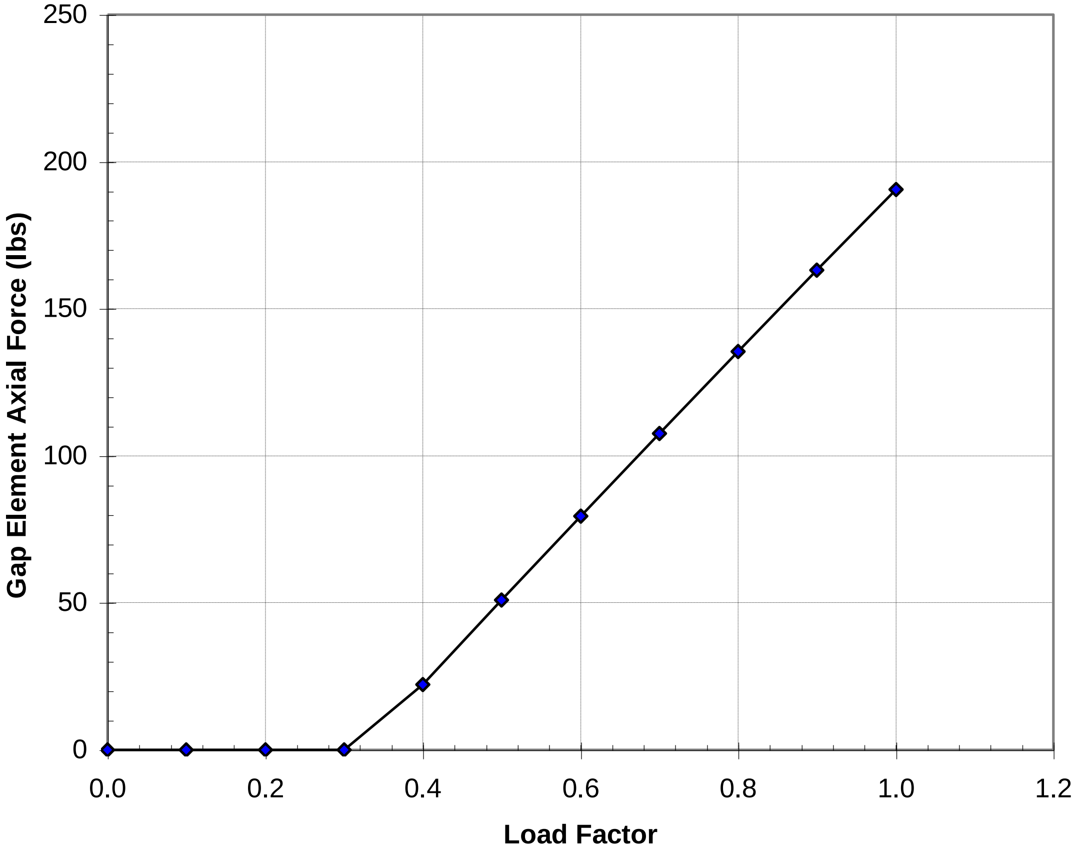


Figure 15-24. Gap Element Axial Displacement vs. Load Factor.

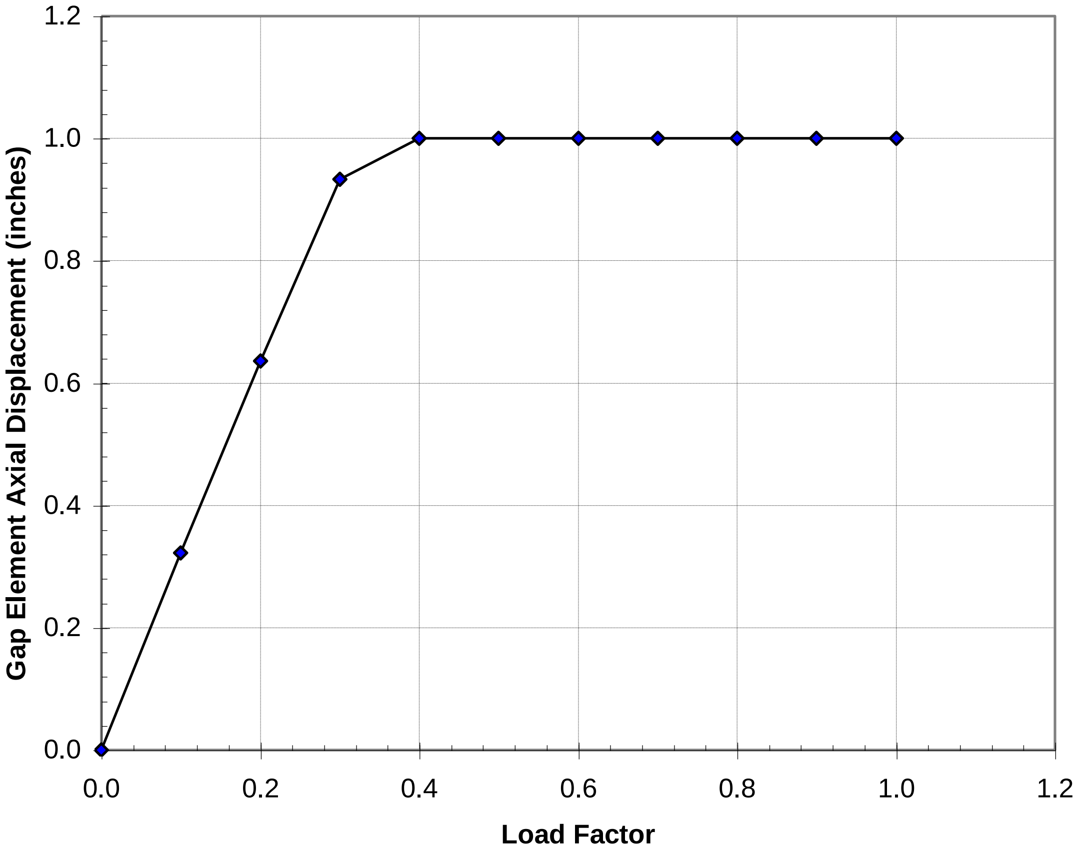


Figure 15-25. Bar Element Stresses at Fixed End vs. Load Factor.

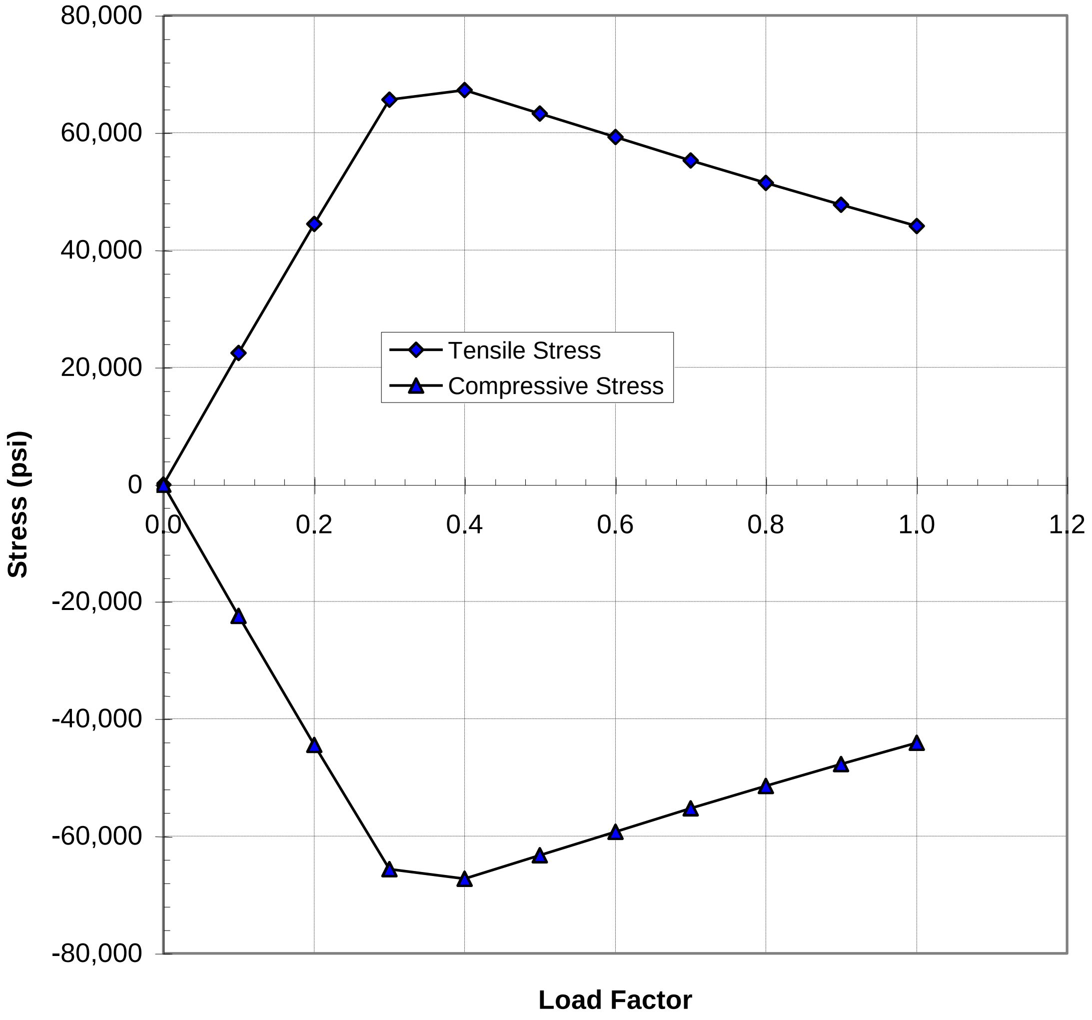
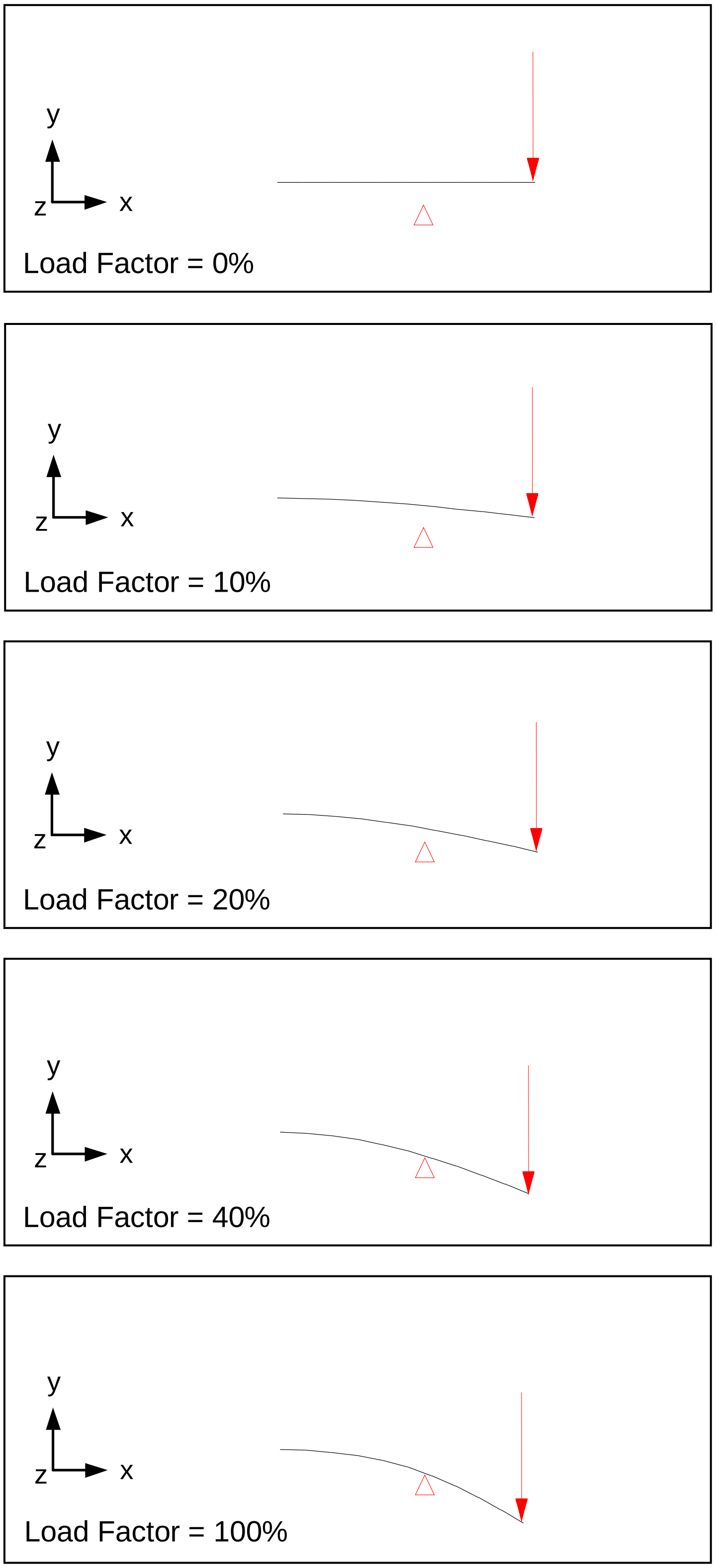


Figure 15-26. Deformed Shapes of a Cantilever Beam with Contact.



15.3.8 Gap Contact with Friction

The next problem is an example of contact with friction. The cantilever beam in Figure 15-27 is subjected to axial and shear loads at its free end. A gap element with friction is positioned just under the shear load at the free end. The first subcase applies the shear load, which closes the gap and activates the gap element friction. The second subcase adds an axial load. The axial load is resisted by the gap element friction, which is directly proportional to the applied shear load. As the axial load is incremented, it reaches a point where it overcomes the frictional force resulting in the beam deflecting axially. Listing 15-17 contains the Model Input File.

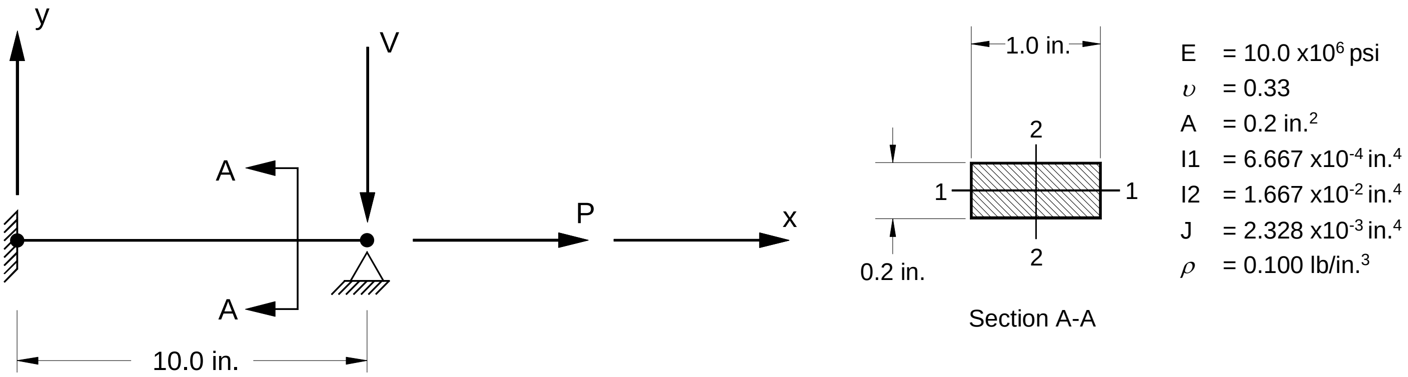


Figure 15-27. 2-D Cantilever Beam Example Problem with Contact and Friction.

Listing 15-17. Model Input File for the Cantilever Beam Problem with Contact and Friction.

```
$
$ NONLINEAR STATIC SOLUTION.
$
SOL NONLINEAR STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = AXIAL LOADED CANTILEVER BEAM WITH CONTACT AND FRICTION
$
DISPLACEMENT = ALL
FORCE = ALL
STRESS = ALL
$
SPC = 1
SUBCASE 1
 LABEL = POINT LOAD AT FREE END (SHEAR)
 LOAD = 1
 NLPARM = 1
SUBCASE 2
 LABEL = POINT LOADS AT FREE END (AXIAL AND SHEAR)
 LOAD = 3
 NLPARM = 2
$
BEGIN BULK
$
$ TURN ON LARGE DISPLACEMENT EFFECTS.
$
PARAM, LGDISP, ON
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,   2,  ,  ,  ,  ,  ,  YES
NLPARM,  2,  10,  ,  ,  ,  ,  ,  YES
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,   0.,  0.,  0
GRID,  2,  0,   1.,   0.,  0.,  0
GRID,  3,  0,   2.,   0.,  0.,  0
GRID,  4,  0,   3.,   0.,  0.,  0
GRID,  5,  0,   4.,   0.,  0.,  0
GRID,  6,  0,   5.,   0.,  0.,  0
GRID,  7,  0,   6.,   0.,  0.,  0
GRID,  8,  0,   7.,   0.,  0.,  0
GRID,  9,  0,   8.,   0.,  0.,  0
GRID, 10,  0,   9.,   0.,  0.,  0
GRID, 11,  0,  10.,   0.,  0.,  0
GRID, 12,  0,  10.,  -0.1, 0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
$
$ GAP ELEMENT.
$
CGAP, 11,  20,  11,  12,  1.,  0.,  0.
$
$ BAR ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  1.667E-2,  6.667E-4,  2.328E-3,
,  -0.5,  0.1,  0.5,  0.1,  -0.5,  -0.1,  0.5,  -0.1
$
$ LINEAR ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
```

Listing 15-17. Model Input File for the Cantilever Beam Problem with Contact and Friction. (Continued)

```
$
$ GAP ELEMENT PROPERTIES.
$
PGAP,  20,  ,  ,  1.E+9,  ,  ,  0.1
$
$ FIXED AT ONE END, MOVEMENT CONSTRAINED TO X-Y PLANE ONLY.
$
SPC1,  1,  123456,  1,  12
SPC1,  1,     345,  2,  THRU,  11
$
$ POINT LOAD AT FREE END (SHEAR).
$
FORCE,  1,  11,  0,  5.E+3,  0., -1.,  0.
$
$ POINT LOAD AT FREE END (AXIAL).
$
FORCE,  2,  11,  0,  1.E+3,  1.,  0.,  0.
$
$ COMBINE LOADS.
$
LOAD,  3,  1.,  1.,  1,  1.,  2
ENDDATA
```

The maximum displacements from the Model Results Output File are shown in tabular form in Listing 15-18. The gap shear force and deflection are plotted in Figures 15-28 and 15-29, respectively. The bar element forces at the fixed end of the beam are plotted in Figure 15-30.

Since the gap element friction force is related to the gap axial force by:

$$F_{\text{friction}} = \mu F_{\text{axial}}$$

And the applied shear load has a constant value of 5000 pounds, the applied axial load must exceed 500 pounds before the static frictional force is overcome. This occurs in subcase 2, increment 6 when the applied axial load is incremented from 500 pounds to 600 pounds.

Listing 15-18. Load Increment Maximum Displacements and Rotations.

```
                                   M A X I M U M   D I S P L A C E M E N T S

   LOAD         LOAD             T1             T2             T3             R1             R2             R3
 INCREMENT     FACTOR
     1     1.000000E-01     9.980040E-07  5.000000E-06  0.000000E+00  0.000000E+00  0.000000E+00  7.499625E-07
     2     2.000000E-01     1.996008E-06  5.000000E-06  0.000000E+00  0.000000E+00  0.000000E+00  7.499250E-07
     3     3.000000E-01     2.994012E-06  5.000000E-06  0.000000E+00  0.000000E+00  0.000000E+00  7.498875E-07
     4     4.000000E-01     3.992016E-06  5.000000E-06  0.000000E+00  0.000000E+00  0.000000E+00  7.498501E-07
     5     5.000000E-01     4.990020E-06  5.000000E-06  0.000000E+00  0.000000E+00  0.000000E+00  7.498127E-07
     6     6.000000E-01     5.002450E-04  5.000000E-06  0.000000E+00  0.000000E+00  0.000000E+00  7.325524E-07
     7     7.000000E-01     1.000000E-03  5.000000E-06  0.000000E+00  0.000000E+00  0.000000E+00  7.166867E-07
     8     8.000000E-01     1.500000E-03  5.000000E-06  0.000000E+00  0.000000E+00  0.000000E+00  7.034875E-07
     9     9.000000E-01     2.000000E-03  5.000000E-06  0.000000E+00  0.000000E+00  0.000000E+00  6.920503E-07
    10     1.000000E+00     2.500000E-03  5.000000E-06  0.000000E+00  0.000000E+00  0.000000E+00  6.818862E-07
```

Figure 15-28. Gap Element Shear Force vs. Load Factor.

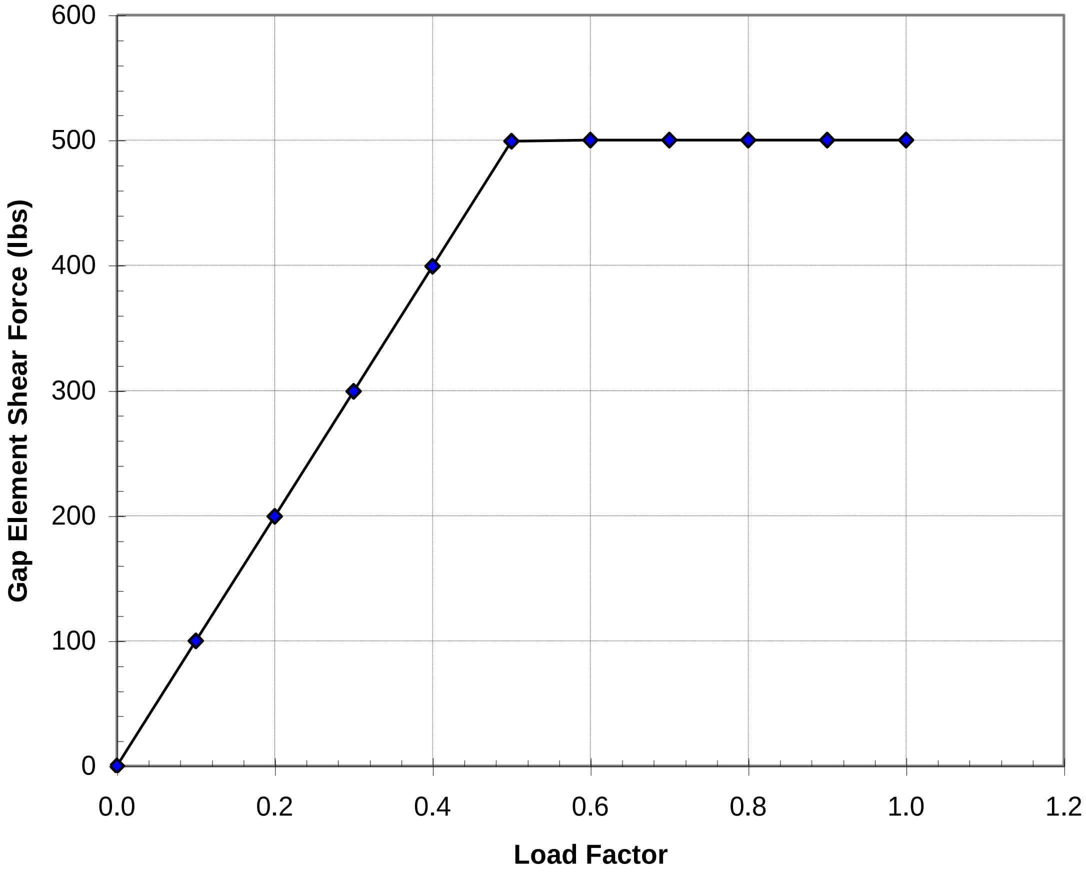


Figure 15-29. Gap Element Shear Displacement vs. Load Factor.

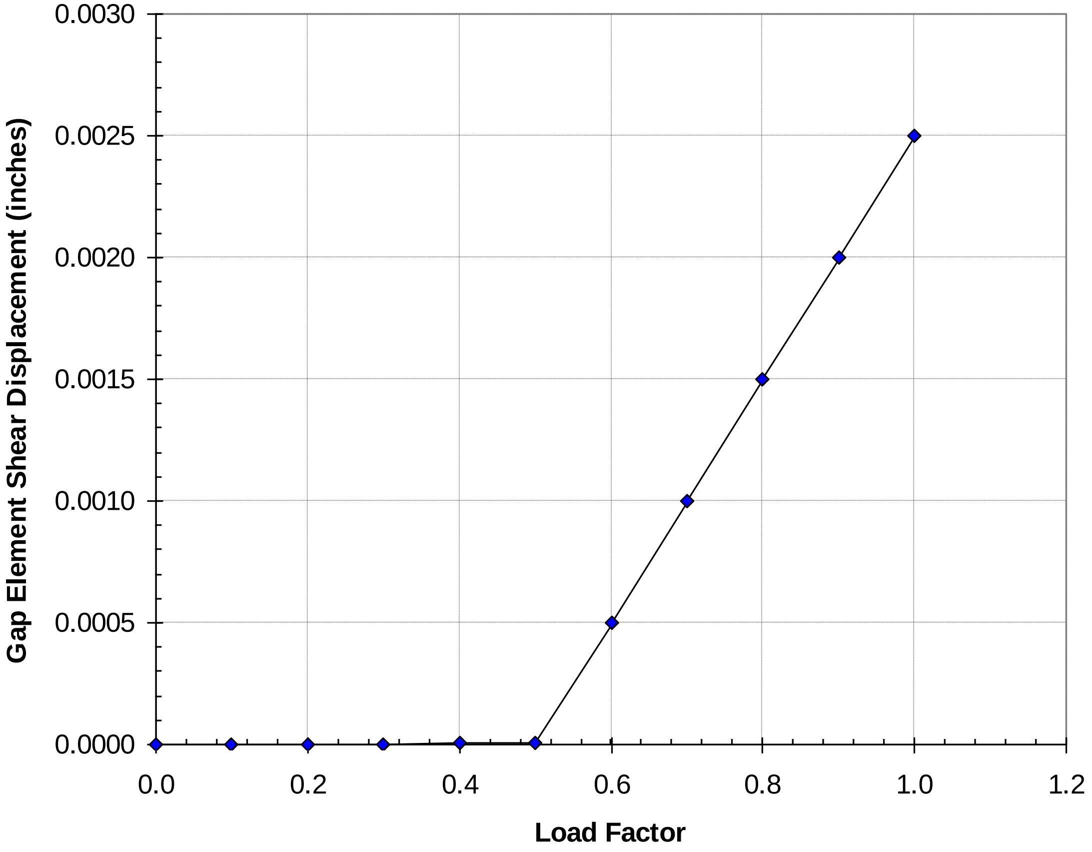
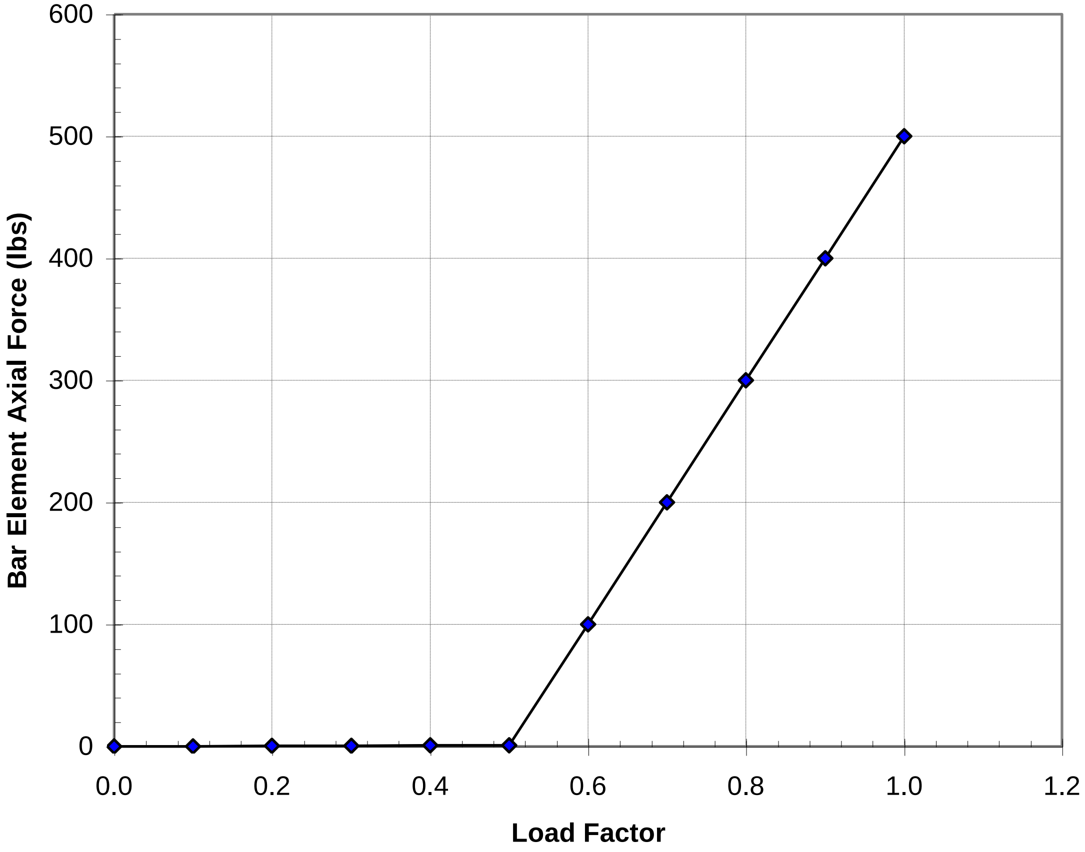


Figure 15-30. Bar Element Axial Force at Fixed End vs. Load Factor.



15.3.9 Slide Line Contact

The next problem is another example of contact using the slide line contact element. The upper cantilever beam in Figure 15-31 supports a distributed load over half its span. The lower beam is simply supported. The upper beam deflects normally until contacting the lower beam. Listing 15-19 contains the Model Input File.

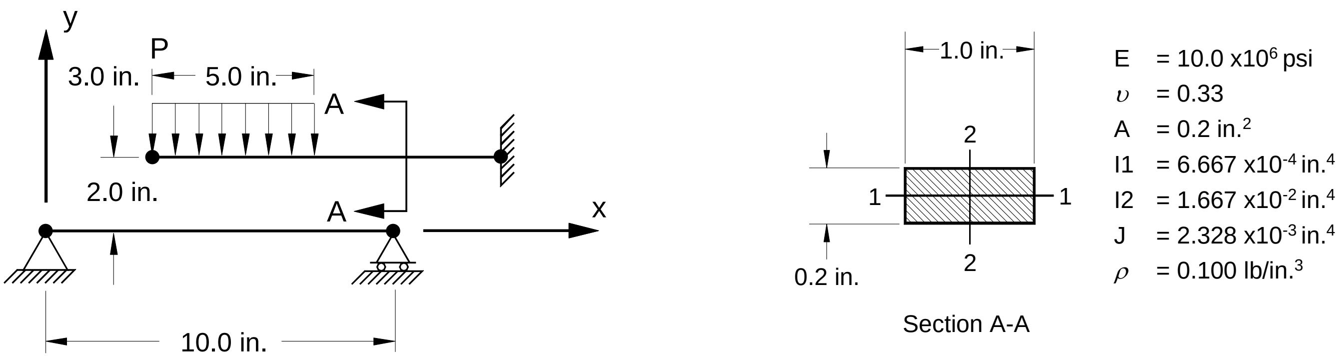


Figure 15-31. 2-D Cantilever Beam Example Problem with Slide Line Contact.

Listing 15-19. Model Input File for the Cantilever Beam Problem with Slide Line Contact.

```
$
$ NONLINEAR STATIC SOLUTION.
$
SOL NONLINEAR STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = DISTRIBUTED LOADED CANTILEVER BEAM WITH SLIDE LINE CONTACT
$
DISPLACEMENT = ALL
STRESS = ALL
FORCE = ALL
$
SUBCASE 1
 LABEL = DISTRIBUTED LOAD ON UPPER BEAM
 LOAD = 1
 NLPARM = 1
 SPC = 1
$
BEGIN BULK
$
$ TURN ON LARGE DISPLACEMENT EFFECTS.
$
PARAM, LGDISP, ON
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  20,  ,  ,  ,  ,  ,  YES
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
$ LOWER BEAM.
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
$
$ UPPER BEAM.
$
GRID, 11,  0,  10.,  0.,  0.,  0
GRID, 12,  0,   3.,  2.,  0.,  0
GRID, 13,  0,   4.,  2.,  0.,  0
GRID, 14,  0,   5.,  2.,  0.,  0
GRID, 15,  0,   6.,  2.,  0.,  0
GRID, 16,  0,   7.,  2.,  0.,  0
GRID, 17,  0,   8.,  2.,  0.,  0
GRID, 18,  0,   9.,  2.,  0.,  0
GRID, 19,  0,  10.,  2.,  0.,  0
GRID, 20,  0,  11.,  2.,  0.,  0
GRID, 21,  0,  12.,  2.,  0.,  0
GRID, 22,  0,  13.,  2.,  0.,  0
$
$ BEAMS MODELED WITH BAR ELEMENTS.
$
$ LOWER BEAM.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
```

Listing 15-19. Model Input File for the Cantilever Beam Problem with Slide Line Contact. (Continued)

```
$
$ UPPER BEAM.
$
CBAR, 11,  20,  12,  13,  0.,  0.,  1.
CBAR, 12,  20,  13,  14,  0.,  0.,  1.
CBAR, 13,  20,  14,  15,  0.,  0.,  1.
CBAR, 14,  20,  15,  16,  0.,  0.,  1.
CBAR, 15,  20,  16,  17,  0.,  0.,  1.
CBAR, 16,  20,  17,  18,  0.,  0.,  1.
CBAR, 17,  20,  18,  19,  0.,  0.,  1.
CBAR, 18,  20,  19,  20,  0.,  0.,  1.
CBAR, 19,  20,  20,  21,  0.,  0.,  1.
CBAR, 20,  20,  21,  22,  0.,  0.,  1.
$
$ BAR ELEMENT MATERIAL AND SECTION PROPERTIES.
$
$ UPPER BEAM (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  1.667E-2,  6.667E-4,  2.328E-3,
,  -0.5,  0.1,  0.5,  0.1,  -0.5,  -0.1,  0.5,  -0.1
$
$ LOWER BEAM (1.0" X 0.1" CROSS-SECTION).
$
PBAR,  20,  100,  0.2,  8.333E-3,  3.333E-5,  3.133E-4,
,  -0.5,  -0.05,  0.5,  -0.05,  0.5,  0.05,  -0.5,  0.05
$
$ LINEAR ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ SLIDE LINE ELEMENT DEFINITION (SYMMETRIC PENETRATION).
$
BCONP,  21,  2,  1,  ,  ,  ,  2
$
$ LOWER SLINE LINE SEGMENTS (1.0" WIDTH).
$
BLSEG,  1,  1,  THRU, 11
BWIDTH,  1,  1.
$
$ UPPER SLINE LINE SEGMENTS (1.0" WIDTH).
$
BLSEG,  2,  22,  THRU, 12,  BY, -1
BWIDTH,  2,  1.
$
$ SLIDE LINE OUTPUT REQUEST.
$
BOUTPUT,  21,  ALL
$
$ UPPER BEAM FIXED AT ONE END, LOWER BEAM PINNED AT ONE END AND
$ FREE TO SLIDE AT THE OTHER, MOVEMENT CONSTRAINED TO X-Y PLANE
$ ONLY.
$
SPC1,  1,   12345,   1
SPC1,  1,     345,   2,  THRU,  10
SPC1,  1,    2345,  11
SPC1,  1,     345,  12,  THRU,  21
SPC1,  1,  123456,  22
$
$ DISTRIBUTED LOAD IN NEGATIVE Y-DIRECTION ON HALF OF UPPER BEAM.
$
PLOAD1,  1,  11,  FZE,  FR,  0.,  100.,  1.,  100.
PLOAD1,  1,  12,  FZE,  FR,  0.,  100.,  1.,  100.
PLOAD1,  1,  13,  FZE,  FR,  0.,  100.,  1.,  100.
PLOAD1,  1,  14,  FZE,  FR,  0.,  100.,  1.,  100.
PLOAD1,  1,  15,  FZE,  FR,  0.,  100.,  1.,  100.
ENDDATA
```

The upper beam y-displacement and contact stress at the beam tip is plotted Figures 15-32 and 15-33, respectively. The bar element stresses at the fixed end of the upper beam and mid-span of the lower beam are plotted in Figure 15-34. The deflected shapes are plotted in Figure 15-35.

The slide line element is defined using the `BCONP` Bulk Data entry, which references a primary and secondary region defined using the `BLSEG` Bulk Data entry. The `BLSEG` entry references consecutive grid points, which form a line or region. Each consecutive pair of grid points defines a segment. In this example, the width of each segment is 1.0 inches, defined via the `BWIDTH` Bulk Data entry. The slide line coordinate system z-axis is specified on the `BCONP` entry and defines the slide line contact plane. The normal direction for a slide line segment is formed from the cross product of the vector from primary node 1 to primary node 2 and the slide line plane vector. The direction of the slide line plane vector and the grid point ordering of the primary line is defined so that the normal direction points toward the secondary region. For symmetric penetration the normals of the primary and secondary segments must face each other. In this example this is accomplished by ordering the grid points counterclockwise since the slide line plane vector is the z-direction of the basic coordinate system (default on the `BCONP` entry).

Figure 15-32. Upper Beam Tip Y-Displacement vs. Load Factor.

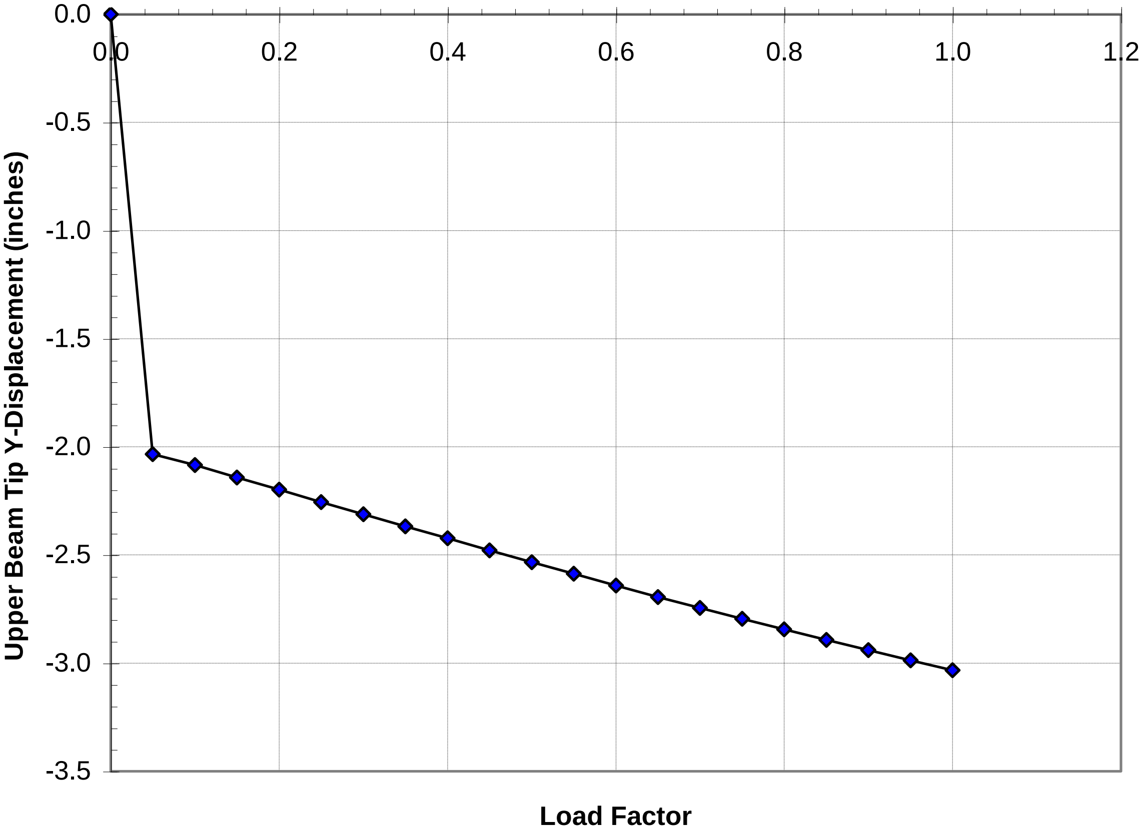


Figure 15-33. Upper Beam Tip Contact Stress vs. Load Factor.

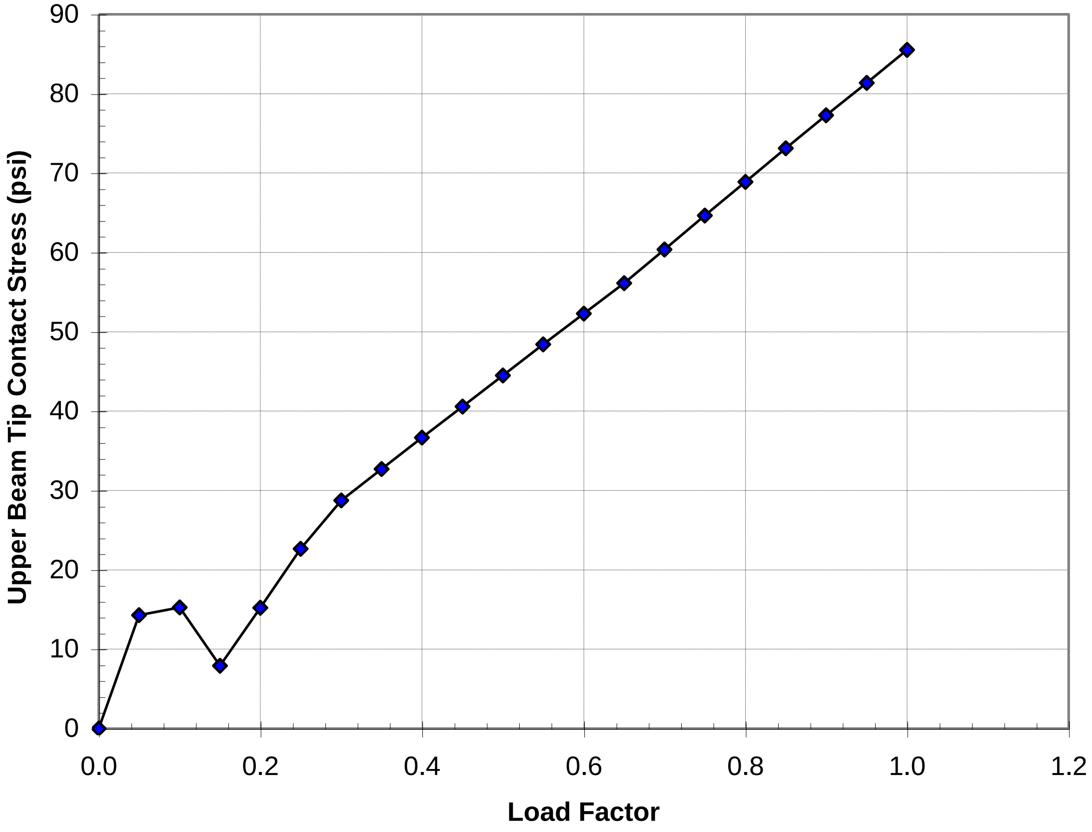


Figure 15-34. Bar Element Stresses at Upper Beam Fixed End vs. Load Factor.

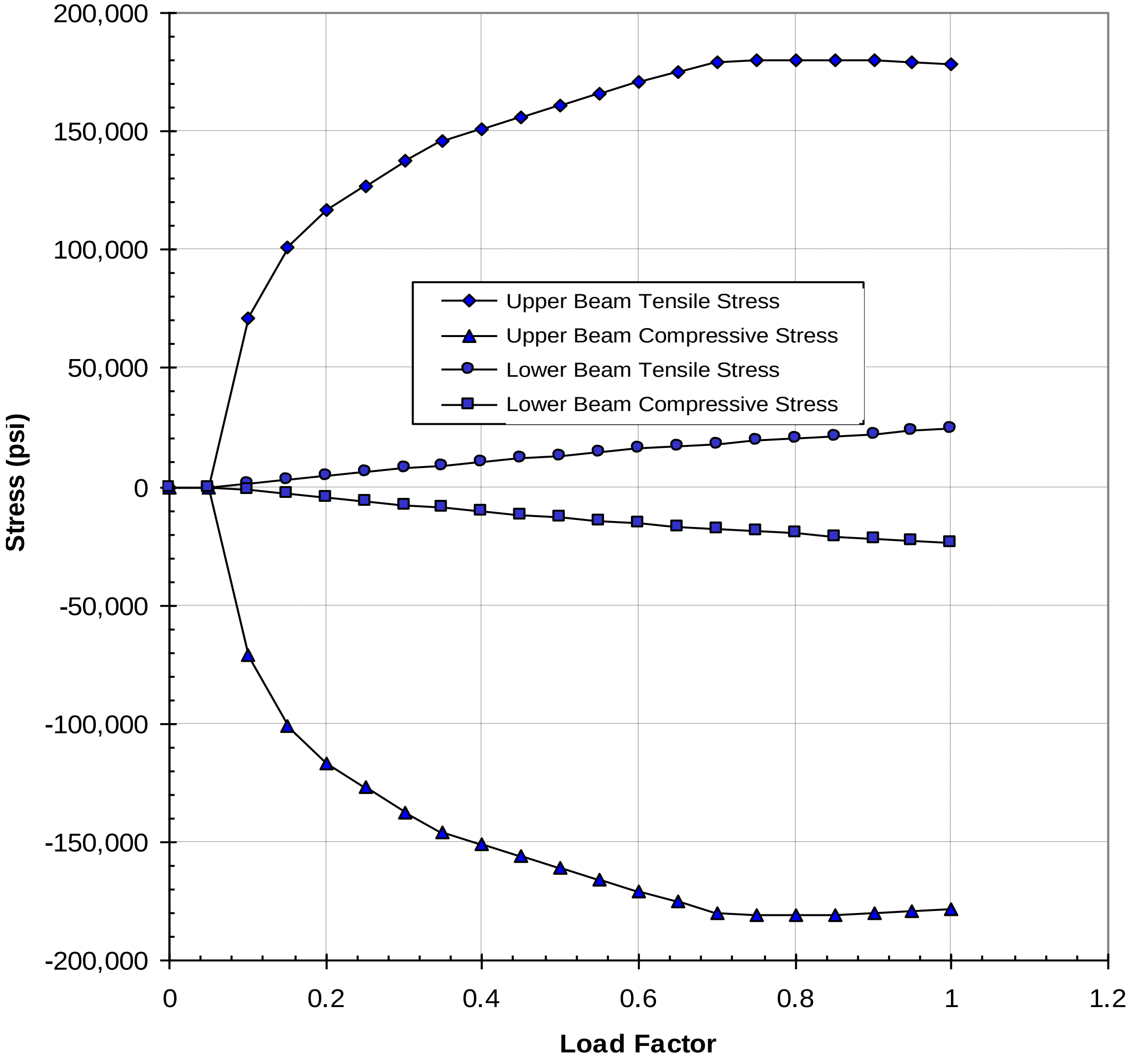
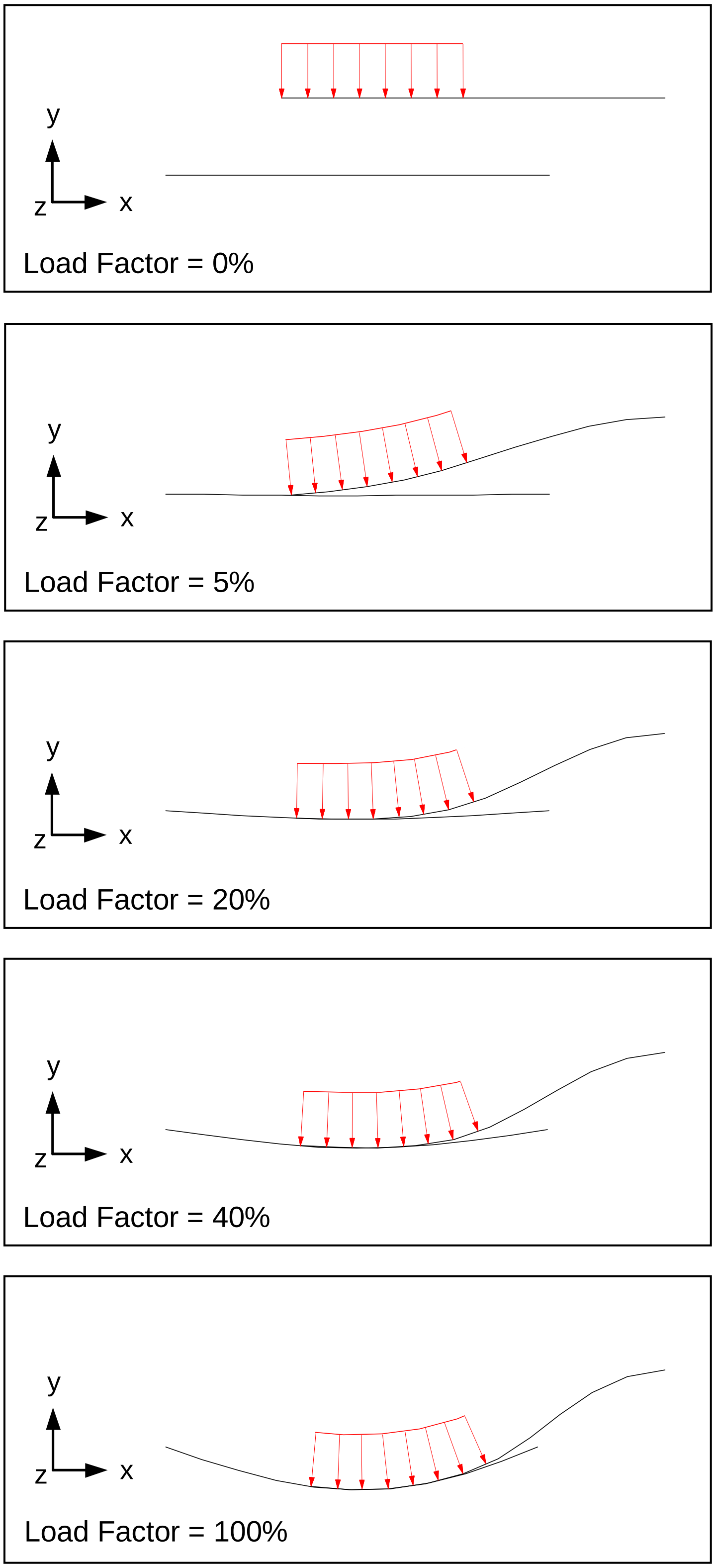
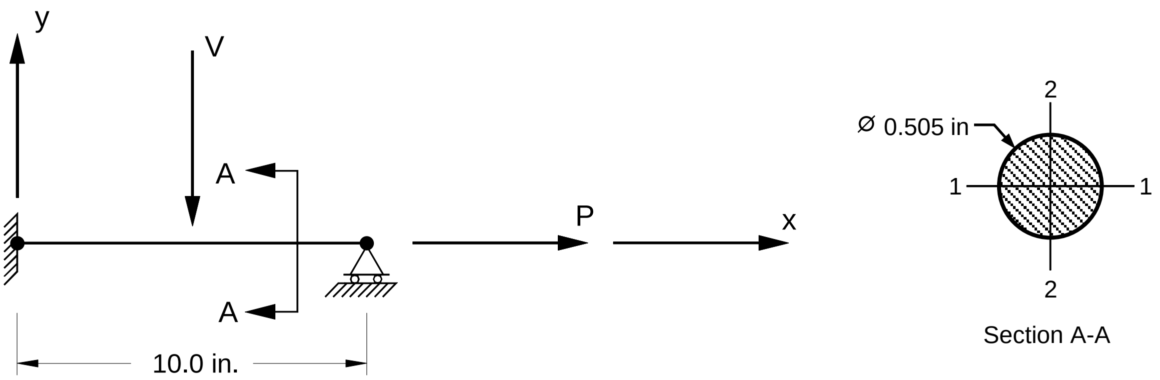


Figure 15-35. Deformed Shapes of a Cantilever Beam with Slide Line Contact.



15.3.10 Tension Only Cable

The next problem is another example of geometric nonlinearity using the Autodesk Inventor Nastran cable element. The cable element is a tension only element with optional bending stiffness. Figure 15-36 shows a cable which is clamped (fixed) at one end and attached with a pulley (free to translate in the x-direction) at the other end. In subcase 1, the cable is initially lose and is tighten by the addition of a tensile load at the pulley. After the slack has been removed, subcase 2 through subcase 4 gradually apply a mid-span load causing the cable to sag a large amount. The sagging cable is then tightened again in subcase 5 with the increase of the tensile load at the pulley and the cable straightens out again (not fully though). Note that the subcase structure is used to initially add a very small amount of shear load, which is then gradually increased thus preventing divergence and increasing solution efficiency. Listing 15-20 contains the Model Input File.



$E = 10.0 \times 10^6$ psi
$\upsilon = 0.33$
$A = 0.2$ in.^{2}
$I1 = 1.0 \times 10^{-9}$ in.^{4}
$I2 = 1.0 \times 10^{-9}$ in.^{4}
$J = 1.0 \times 10^{-9}$ in.^{4}
$\rho = 0.100$ lb/in.^{3}

Figure 15-36. 2-D Cable Example Problem.

Listing 15-20. Model Input File for the Cable Example Problem.

```
$
$ NONLINEAR STATIC SOLUTION.
$
SOL NONLINEAR STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = SLACK CABLE LOADED OUT OF PLANE THEN PULLED TAUGHT
$
DISPLACEMENT = ALL
FORCE = ALL
STRESS = ALL
$
SPC = 1
SUBCASE 1
 LABEL = INITIAL AXIAL LOAD IN CABLE ADDED WITH P AT 10% -SLACK IS REMOVED
 LOAD = 10
 NLPARM = 1
SUBCASE 2
 LABEL = MID-SPAN LOAD ADDED AT 0.1% OF V, AXIAL LOAD AT 10% OF P
 LOAD = 20
 NLPARM = 1
SUBCASE 3
 LABEL = MID-SPAN LOAD AT 1% OF V, AXIAL LOAD AT 10% OF P
 LOAD = 30
 NLPARM = 1
SUBCASE 4
 LABEL = MID-SPAN LOAD AT 100% OF V, AXIAL LOAD AT 10% OF P
 LOAD = 40
 NLPARM = 1
SUBCASE 5
 LABEL = MID-SPAN LOAD AT 100% OF V, AXIAL LOAD AT 100% OF P
 LOAD = 50
 NLPARM = 1
$
BEGIN BULK
$
$ TURN ON LARGE DISPLACEMENT EFFECTS.
$
PARAM, LGDISP, ON
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  10,  ,  ,  ,  50,  P,  YES
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ CABLE MODELED WITH CABLE ELEMENTS.
$
CCABLE,  1,  10,   1,   2,  0.,  0.,  1.
CCABLE,  2,  10,   2,   3,  0.,  0.,  1.
CCABLE,  3,  10,   3,   4,  0.,  0.,  1.
CCABLE,  4,  10,   4,   5,  0.,  0.,  1.
CCABLE,  5,  10,   5,   6,  0.,  0.,  1.
CCABLE,  6,  10,   6,   7,  0.,  0.,  1.
CCABLE,  7,  10,   7,   8,  0.,  0.,  1.
CCABLE,  8,  10,   8,   9,  0.,  0.,  1.
CCABLE,  9,  10,   9,  10,  0.,  0.,  1.
CCABLE, 10,  10,  10,  11,  0.,  0.,  1.
```

Listing 15-20. Model Input File for the Cable Example Problem. (Continued)

```
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES.
$
PCABLE,  10,  100,  0.,  0.,  0.2
$
$ LINEAR ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ FIXED AT BOTH ENDS -ONE END FREE TO TRANSLATE IN X-DIR.
$
SPC1,  1,  123456,   1
SPC1,  1,   23456,  11
SPC1,  1,     345,   1,  THRU,  11
$
$ AXIAL TENSILE LOAD (X-DIRECTION).
$
FORCE,  1,  11,  0,  1.E+5,  1.,   0.,  0.
$
$ MID-SPAN OUT OF PLANE LOAD (Y-DIRECTION).
$
FORCE,  2,   6,  0,  5.E+3,  0.,  -1.,  0.
$
$ SCALE TOTAL LOADS TO GET SUBCASE LOADING.
$
LOAD,  10,  1.,  0.,     2,  0.1,  1
LOAD,  20,  1.,  0.001,  2,  0.1,  1
LOAD,  30,  1.,  0.01,   2,  0.1,  1
LOAD,  40,  1.,  1.,     2,  0.1,  1
LOAD,  50,  1.,  1.,     2,  1.,   1
ENDDATA
```

The cable element must reference a linear isotropic material, but may be temperature dependent. Both thermal and inertia element loads are supported.

The cable element, when subjected to lateral loading, requires a small amount of bending stiffness. The default bending stiffness is based on the square of the area of a circular cross-section. While great amount of bending does occur, the initial bending stiffness is primary due to the differential stiffness contribution generated from the initial prestress in the cable (subcase 1). The maximum displacements from the Model Results Output File are shown in tabular form in Listing 15-21. The deflected shapes are plotted in Figure 15-37.

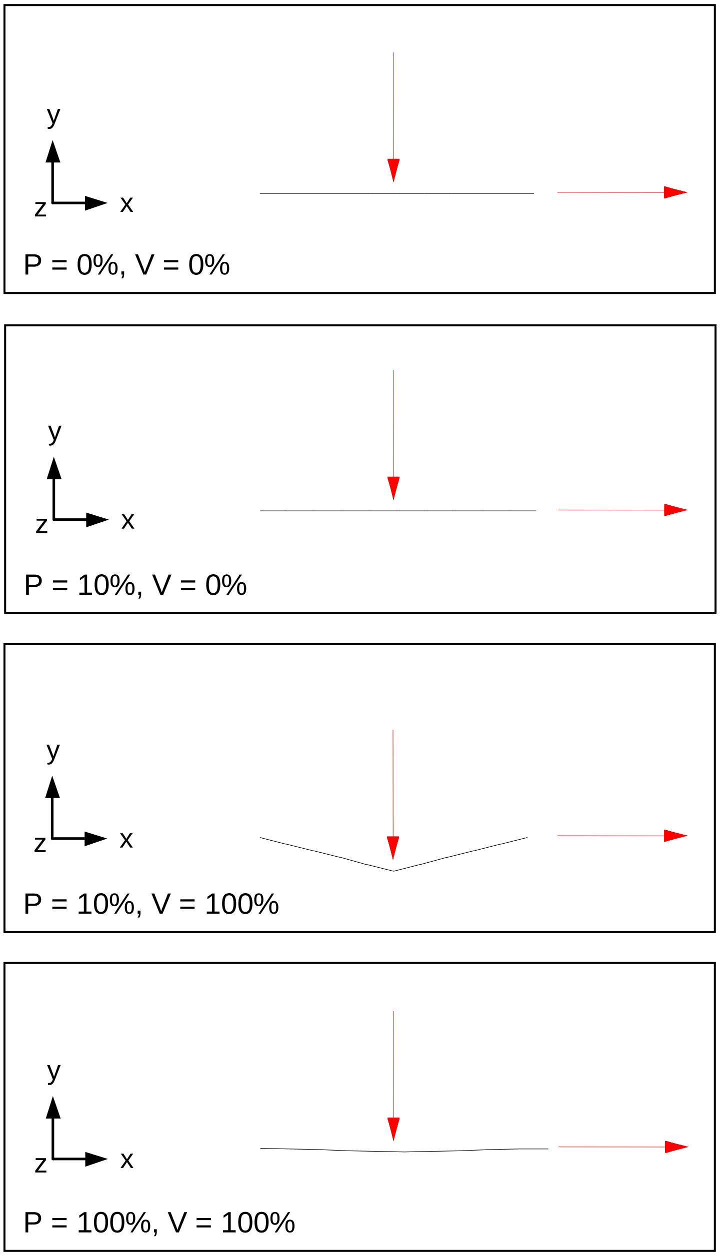
Listing 15-21. Subcase and Load Increment Maximum Displacements and Rotations.

```
 INITIAL AXIAL LOAD IN CABLE ADDED WITH P AT 10% -SLACK IS REMOVED   SUBCASE 1

                                          M A X I M U M   D I S P L A C E M E N T S

    LOAD        LOAD              T1            T2            T3            R1            R2            R3
  INCREMENT    FACTOR
      1     1.000000E-01     5.000000E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      2     2.000000E-01     1.000000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      3     3.000000E-01     1.500000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      4     4.000000E-01     2.000000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      5     5.000000E-01     2.500000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      6     6.000000E-01     3.000000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      7     7.000000E-01     3.500000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      8     8.000000E-01     4.000000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      9     9.000000E-01     4.500000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
     10     1.000000E+00     5.000000E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
```

Listing 15-21. Subcase and Load Increment Maximum Displacements and Rotations. (Continued)

```
MID-SPAN LOAD ADDED AT 0.1% OF V, AXIAL LOAD AT 10% OF P             SUBCASE 2

                                          M A X I M U M   D I S P L A C E M E N T S

    LOAD        LOAD             T1            T2            T3            R1            R2            R3
  INCREMENT    FACTOR
      1     1.000000E-01     5.000000E-02  4.599373E-05  0.000000E+00  0.000000E+00  0.000000E+00  1.292988E-05
      2     2.000000E-01     5.000000E-02  9.286826E-05  0.000000E+00  0.000000E+00  0.000000E+00  2.604774E-05
      3     3.000000E-01     5.000000E-02  1.397576E-04  0.000000E+00  0.000000E+00  0.000000E+00  3.916815E-05
      4     4.000000E-01     5.000000E-02  1.866471E-04  0.000000E+00  0.000000E+00  0.000000E+00  5.228859E-05
      5     5.000000E-01     4.999999E-02  2.335366E-04  0.000000E+00  0.000000E+00  0.000000E+00  6.540904E-05
      6     6.000000E-01     4.999999E-02  2.804262E-04  0.000000E+00  0.000000E+00  0.000000E+00  7.852949E-05
      7     7.000000E-01     4.999998E-02  3.273157E-04  0.000000E+00  0.000000E+00  0.000000E+00  9.164993E-05
      8     8.000000E-01     4.999998E-02  3.742053E-04  0.000000E+00  0.000000E+00  0.000000E+00  1.047704E-04
      9     9.000000E-01     4.999997E-02  4.210948E-04  0.000000E+00  0.000000E+00  0.000000E+00  1.178908E-04
     10     1.000000E+00     4.999996E-02  4.679843E-04  0.000000E+00  0.000000E+00  0.000000E+00  1.310113E-04

MID-SPAN LOAD AT 1% OF V, AXIAL LOAD AT 10% OF P                     SUBCASE 3

                                          M A X I M U M   D I S P L A C E M E N T S

    LOAD        LOAD             T1            T2            T3            R1            R2            R3
  INCREMENT    FACTOR
      1     1.000000E-01     4.999995E-02  8.828237E-04  0.000000E+00  0.000000E+00  0.000000E+00  2.475708E-04
      2     2.000000E-01     4.999983E-02  1.304708E-03  0.000000E+00  0.000000E+00  0.000000E+00  3.656336E-04
      3     3.000000E-01     4.999962E-02  1.726710E-03  0.000000E+00  0.000000E+00  0.000000E+00  4.837168E-04
      4     4.000000E-01     4.999933E-02  2.148714E-03  0.000000E+00  0.000000E+00  0.000000E+00  6.018000E-04
      5     5.000000E-01     4.999897E-02  2.570716E-03  0.000000E+00  0.000000E+00  0.000000E+00  7.198829E-04
      6     6.000000E-01     4.999853E-02  2.992717E-03  0.000000E+00  0.000000E+00  0.000000E+00  8.379656E-04
      7     7.000000E-01     4.999800E-02  3.414718E-03  0.000000E+00  0.000000E+00  0.000000E+00  9.560480E-04
      8     8.000000E-01     4.999740E-02  3.836718E-03  0.000000E+00  0.000000E+00  0.000000E+00  1.074130E-03
      9     9.000000E-01     4.999672E-02  4.258716E-03  0.000000E+00  0.000000E+00  0.000000E+00  1.192212E-03
     10     1.000000E+00     4.999595E-02  4.680714E-03  0.000000E+00  0.000000E+00  0.000000E+00  1.310294E-03

MID-SPAN LOAD AT 100% OF V, AXIAL LOAD AT 10% OF P                   SUBCASE 4

                                          M A X I M U M   D I S P L A C E M E N T S

    LOAD        LOAD             T1            T2            T3            R1            R2            R3
  INCREMENT    FACTOR
      1     1.000000E-01     4.942530E-02  5.109248E-02  0.000000E+00  0.000000E+00  0.000000E+00  1.429679E-02
      2     2.000000E-01     4.787843E-02  9.741249E-02  0.000000E+00  0.000000E+00  0.000000E+00  2.725629E-02
      3     3.000000E-01     4.538477E-02  1.438168E-01  0.000000E+00  0.000000E+00  0.000000E+00  4.024897E-02
      4     4.000000E-01     4.193677E-02  1.901387E-01  0.000000E+00  0.000000E+00  0.000000E+00  5.321722E-02
      5     5.000000E-01     3.753335E-02  2.362959E-01  0.000000E+00  0.000000E+00  0.000000E+00  6.614271E-02
      6     6.000000E-01     3.218672E-02  2.824505E-01  0.000000E+00  0.000000E+00  0.000000E+00  7.907912E-02
      7     7.000000E-01     2.594701E-02  3.282489E-01  0.000000E+00  0.000000E+00  0.000000E+00  9.191215E-02
      8     8.000000E-01     1.869709E-02  3.742844E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.048308E-01
      9     9.000000E-01     1.056318E-02  4.200367E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.176788E-01
     10     1.000000E+00     4.546152E-03  4.653870E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.304138E-01

MID-SPAN LOAD AT 100% OF V, AXIAL LOAD AT 100% OF P                  SUBCASE 5

                                          M A X I M U M   D I S P L A C E M E N T S

    LOAD        LOAD             T1            T2            T3            R1            R2            R3
  INCREMENT    FACTOR
      1     1.000000E-01     6.976324E-02  3.398500E-01  0.000000E+00  0.000000E+00  0.000000E+00  9.294212E-02
      2     2.000000E-01     1.246073E-01  2.684120E-01  0.000000E+00  0.000000E+00  0.000000E+00  7.179141E-02
      3     3.000000E-01     1.745826E-01  2.227584E-01  0.000000E+00  0.000000E+00  0.000000E+00  5.838510E-02
      4     4.000000E-01     2.224723E-01  1.907897E-01  0.000000E+00  0.000000E+00  0.000000E+00  4.908088E-02
      5     5.000000E-01     2.693440E-01  1.670149E-01  0.000000E+00  0.000000E+00  0.000000E+00  4.221547E-02
      6     6.000000E-01     3.155724E-01  1.488335E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.698013E-02
      7     7.000000E-01     3.613846E-01  1.349679E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.307577E-02
      8     8.000000E-01     4.070960E-01  1.223913E-01  0.000000E+00  0.000000E+00  0.000000E+00  2.954238E-02
      9     9.000000E-01     4.525827E-01  1.129070E-01  0.000000E+00  0.000000E+00  0.000000E+00  2.671430E-02
     10     1.000000E+00     4.979398E-01  1.048321E-01  0.000000E+00  0.000000E+00  0.000000E+00  2.446734E-02
```

Figure 15-37. Deformed Shapes of a Cable Under Multiple Loading.



The next problem is another application of the cable element. The cantilever beam in Figure 15-38 is supported by 3 vertical cables and is subjected to a shear load at its free end. The left cable is preloaded to 1000 pounds. The middle cable is initially loose with 1.0 inch of slack. The right cable has a defined tensile allowable of 7.E+4 psi, above which it will fail completely. The first third of the applied loading (0.0 to 1.0 load factor) extends the beam axially while the left cable deflects the beam upward at its attach point deflecting the tip downward slightly. The next third (1.0 to 2.0 load factor) introduces a shear load at the beam tip, which results in only a slight deflection due to the support of the right cable. As the load is increased a point is reached (1.5 load factor) where the allowable for the right cable is exceeded and the cable snaps. The load path is then transferred to beam bending until the 1.0 inch slack in the middle cable is taken up and the middle cable begins to load.

Listing 15-22 contains the Model Input File. Stress for each cable is plotted in Figure 15-39. The beam tip deflection is plotted in Figure 15-40.

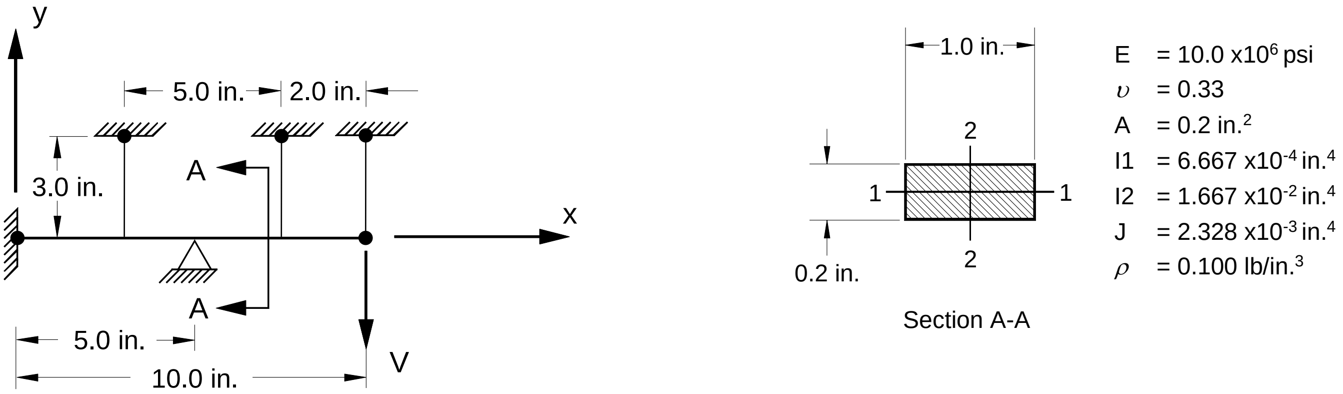


Figure 15-38. 2-D Cantilever Beam Example Problem with Cable Elements.

Listing 15-22. Model Input File for the Cantilever Beam Problem with Cable Elements.

```
$
$ NONLINEAR STATIC SOLUTION.
$
SOL NONLINEAR STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = CANTILEVER BEAM SUPPORTED BY CABLES
$
DISPLACEMENT = ALL
FORCE = ALL
STRESS = ALL
$
SPC = 1
SUBCASE 1
 LABEL = INITIAL AXIAL LOAD AT FREE END OF THE BEAM
 LOAD = 10
 NLPARM = 1
SUBCASE 2
 LABEL = END LOAD AT 10% OF V, AXIAL LOAD AT 100% OF P
 LOAD = 20
 NLPARM = 1
SUBCASE 3
 LABEL = END LOAD AT 100% OF V, AXIAL LOAD AT 100% OF P
 LOAD = 30
 NLPARM = 1
$
BEGIN BULK
$
$ TURN ON LARGE DISPLACEMENT EFFECTS.
$
PARAM, LGDISP, ON
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  10,  ,  ,  ,  ,  P,  YES
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
GRID, 12,  0,   3.,  3.,  0.,  0
GRID, 13,  0,   8.,  3.,  0.,  0
GRID, 14,  0,  10.,  3.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
$
$ CABLE ELEMENTS.
$
CCABLE, 11,  20,   4,  12
CCABLE, 12,  30,   9,  13
CCABLE, 13,  40,  11,  14
```

Listing 15-22. Model Input File for the Cantilever Beam Problem with Cable Elements. (Continued)

```
$
$ BAR ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  1.667E-2,  6.667E-4,  2.328E-3,
,  -0.5,  -0.1,  0.5,  -0.1,  0.5,  0.1,  -0.5,  0.1
$
$ LINEAR ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ CABLE ELEMENT PROPERTIES (0.1" DIAMETER).
$
PCABLE,  20,  100,  0., 1000.,  7.854E-3
PCABLE,  30,  100,  1.,    0.,  7.854E-3
PCABLE,  40,  100,  0.,    0.,  7.854E-3,  ,  7.E+4
$
$ PINNED AT ONE END AND MIDPOINT, MOVEMENT CONSTRAINED TO$ X-Y PLANE
$ ONLY.
$
SPC1,  1,   12345,   1
SPC1,  1,     345,   2,  THRU,  11
SPC1,  1,  123456,  12,  THRU,  14
SPC1,  1,   12345,   6
$
$ AXIAL TENSILE LOAD (X-DIRECTION).
$
FORCE,  1,  11,  0,  1.E+4,  1.,   0.,  0.
$
$ POINT LOAD AT FREE END (SHEAR).
$
FORCE,  2,  11,  0,  1.E+4,  0.,  -1.,  0.
$
$ ADD COMPONENT LOADS TO GET SUBCASE LOADING.
$
LOAD,  10,  1.,  1.,  1,  0.,   2
LOAD,  20,  1.,  1.,  1,  0.1,  2
LOAD,  30,  1.,  1.,  1,  1.,   2
ENDDATA
```

Figure 13-39. Cable Element Stress vs. Load Factor.

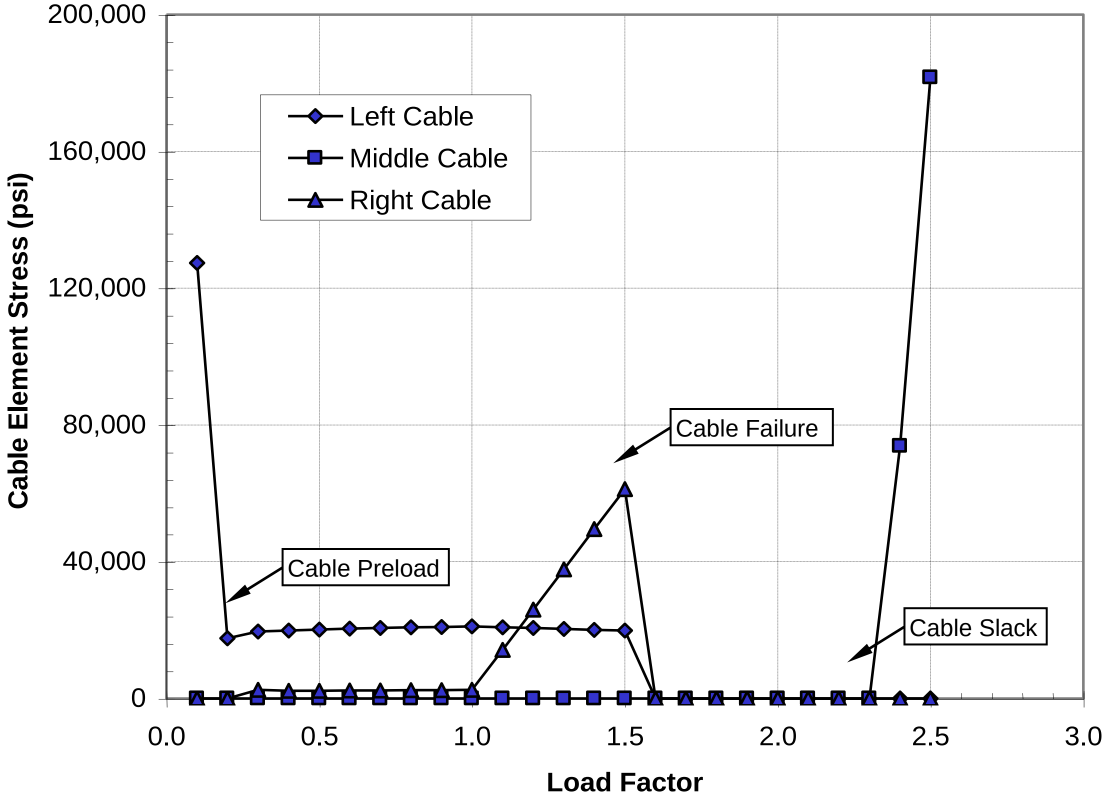
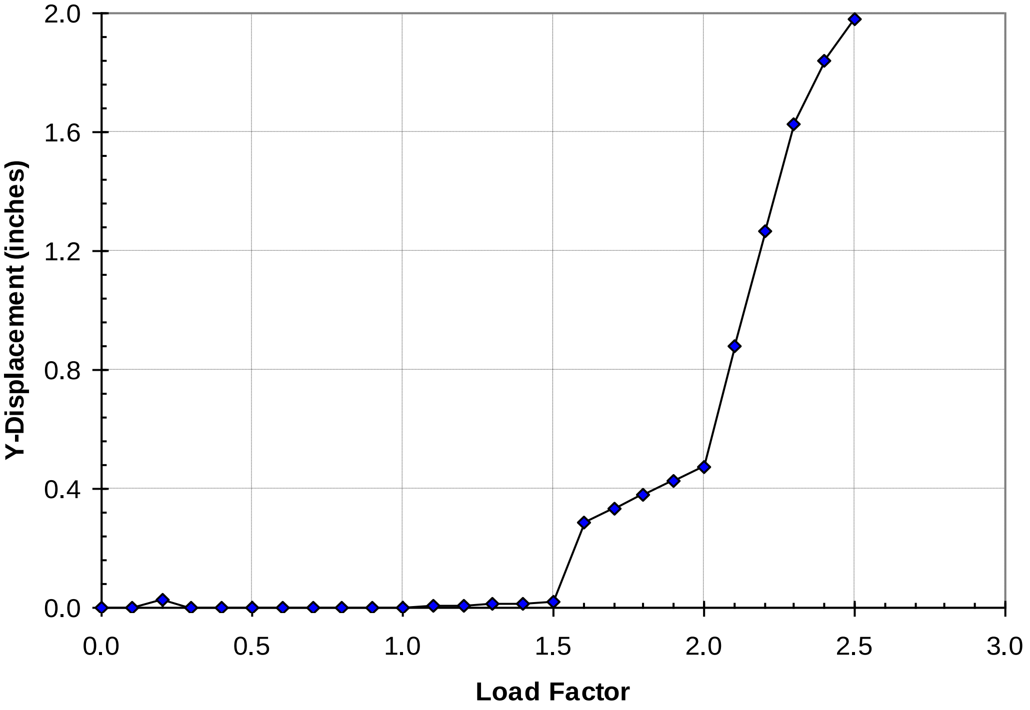


Figure 13-40. Beam Tip Displacement vs. Load Factor.



15.3.11 Creep and Viscoelastic Material

The next problem is an example of creep analysis using a generalized viscoelastic material. The beam in Figure 15-41 is made from an elastic material and subjected to a 4000 pound axial force at 1000 °F for 6000 hours. The load is ramped up over 5 increments in the first subcase without any creep effects. The second subcase maintains the load while adding the creep effects over 20 increments with each increment representing 300 hours.

The material behavior is both stress and time dependent and is represented using:

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

where for this example,

$$A(\sigma) = (6.985\text{E-}6)\sigma^{2.444}$$

$$R(\sigma) = (7.032\text{E-}4)e^{0.1072\sigma}$$

$$K(\sigma) = (6.73\text{E-}9)[\sinh(0.1479\,\sigma)]^{3.0}$$

The above creep law is defined using the `CREEP` Bulk Data entry where the input parameters are typically obtained empirically. For this example time is in hours and applied load is in kilo pounds. Listing 15-23 contains the Model Input File.

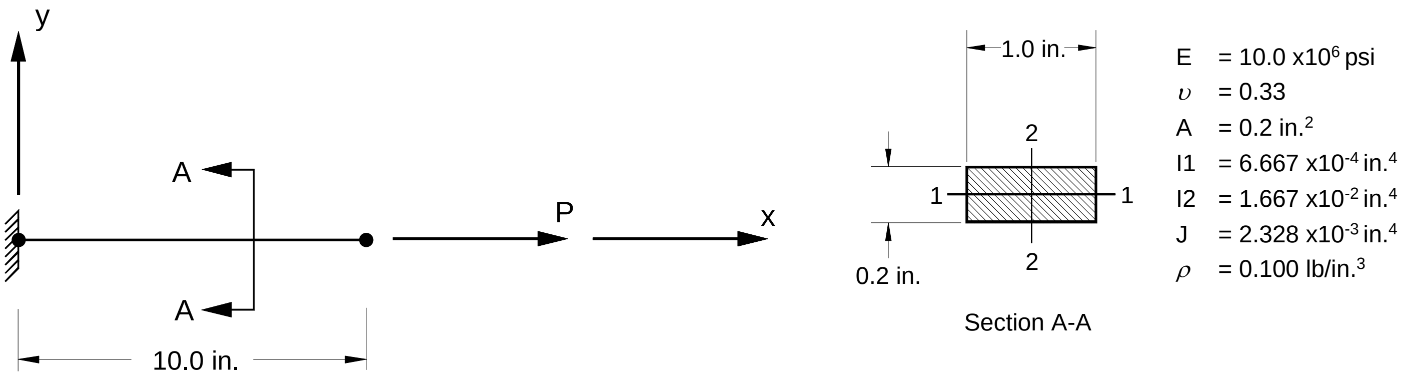


Figure 15-41. 2-D Cantilever Beam Example Problem with Creep Effects.

Listing 15-23. Model Input File for Cantilever Beam Problem with Creep Effects.

```
$
$ NONLINEAR STATIC SOLUTION.
$
SOL NONLINEAR STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = AXIAL LOADED CANTILEVER BEAM WITH CREEP EFFECTS
$
DISPLACEMENT = ALL
STRESS = ALL
FORCE = ALL
TEMPERATURE(INITIAL) = 1
SUBCASE 1
 LABEL = TENSILE LOAD IN X-DIR WITHOUT CREEP
 TEMPERATURE(MATERIAL) = 1
 LOAD = 1
 NLPARM = 1
 SPC = 1
SUBCASE 2
 LABEL = TENSILE LOAD IN X-DIR WITH CREEP
 TEMPERATURE(MATERIAL) = 1
 LOAD = 1
 NLPARM = 2
 SPC = 1
$
BEGIN BULK
$
$ TURN ON LARGE DISPLACEMENT EFFECTS.
$
PARAM, LGDISP, ON
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,   5,    0.,  ,  ,  ,  ,  YES
NLPARM,  2,  20,  300.,  ,  ,  ,  ,  YES
$
$ FAHRENHEIT TO ABSOLUTE TEMPERATURE CONVERSION FACTOR.
$
PARAM, TABS, 459.69
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
$
$ BAR ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  1.667E-2,  6.667E-4,  2.328E-3
```

Listing 15-23. Model Input File for the Cantilever Beam Problem with Creep Effects. (Continued)

```
$
$ LINEAR ELEMENT MATERIAL PROPERTIES (MA956 IN KSI).
$
MAT1,  100,  3.4E+4,  ,  0.33,  0.3
$
$ CREEP CHARACTERISTICS.
$
CREEP,  100,  1000.,  ,  CRLAW,
,  111,  6.985E-6,  2.444,  7.032E-4,  0.1072,  6.73E-9,  0.1479,  3.0
$
$ FIXED AT ONE END, FREE TO TRANSLATE IN X-DIR AT OTHER END.
$
SPC1,  1,  123456,   1
SPC1,  1,   23456,   2,  THRU,  11
$
$ TENSILE LOAD (X-DIRECTION IN KIPS).
$
FORCE,  1,  11,  0,  4.,  1.,  0.,  0.
$
$ INITIAL TEMPERATURE DISTRIBUTION.
$
TEMPD,  1,  1000.
ENDDATA
```

The maximum displacements from the Model Results Output File are shown in tabular form in Listing 15-24 and graphically in Figure 15-42. Bar element equivalent stress is plotted against effective creep strain in Figure 15-43 and time is plotted against total strain in Figure 15-44.

Listing 15-24. Load Increment Maximum Displacements.

```
TENSILE LOAD IN X-DIR WITHOUT CREEP                                 SUBCASE 1

                                    M A X I M U M   D I S P L A C E M E N T S

     LOAD        LOAD              T1            T2            T3            R1            R2            R3
  INCREMENT     FACTOR
       1     2.000000E-01     1.176471E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       2     4.000000E-01     2.352941E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       3     6.000000E-01     3.529412E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       4     8.000000E-01     4.705882E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       5     1.000000E+00     5.882353E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
```

Listing 15-24. Load Increment Maximum Displacements. (Continued)

```
 TENSILE LOAD IN X-DIR WITH CREEP                                          SUBCASE 2

                                           M A X I M U M   D I S P L A C E M E N T S

       LOAD         LOAD             T1             T2             T3             R1             R2             R3
    INCREMENT      FACTOR
         1      5.000000E-02     8.396330E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
         2      1.000000E-01     1.201145E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
         3      1.500000E-01     1.428504E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
         4      2.000000E-01     1.612447E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
         5      2.500000E-01     1.782919E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
         6      3.000000E-01     1.948112E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
         7      3.500000E-01     2.112452E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
         8      4.000000E-01     2.276483E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
         9      4.500000E-01     2.440406E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        10      5.000000E-01     2.604289E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        11      5.500000E-01     2.768159E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        12      6.000000E-01     2.932023E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        13      6.500000E-01     3.095886E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        14      7.000000E-01     3.259749E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        15      7.500000E-01     3.423611E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        16      8.000000E-01     3.587473E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        17      8.500000E-01     3.751335E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        18      9.000000E-01     3.915197E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        19      9.500000E-01     4.079059E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        20      1.000000E+00     4.244452E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
```

Figure 15-42. Maximum Displacement vs. Load Factor.

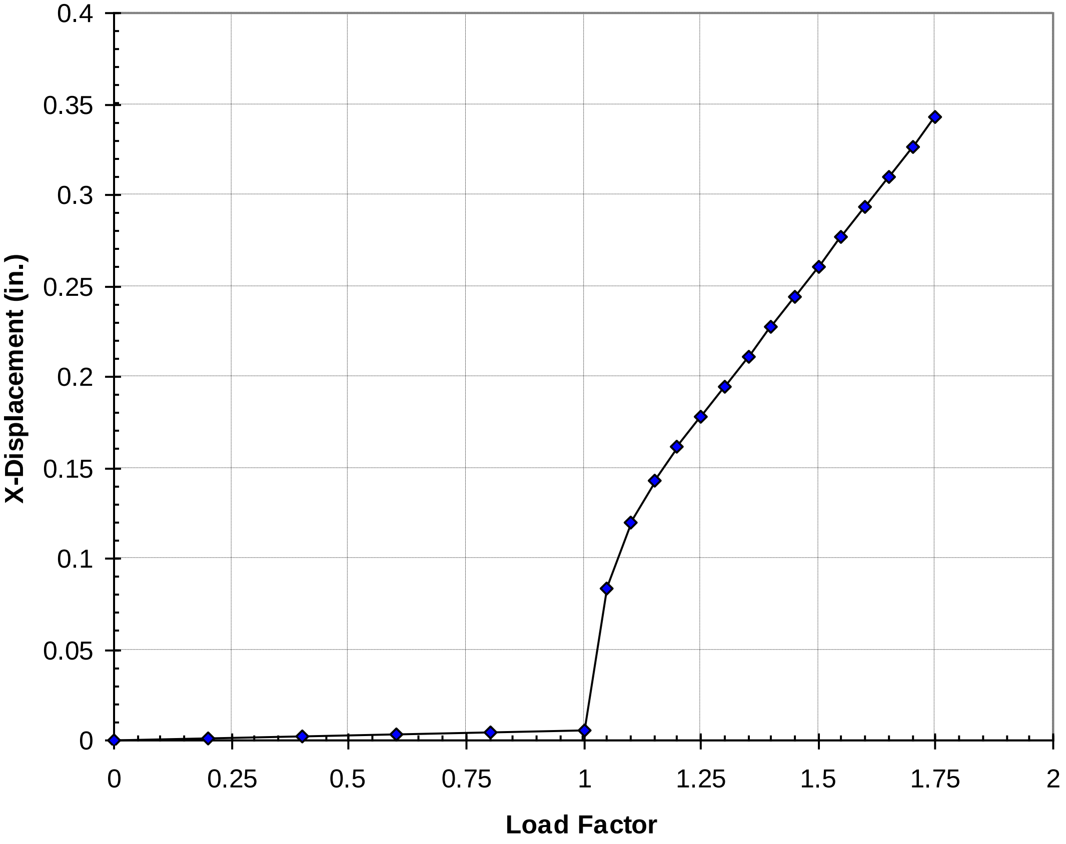


Figure 15-43. Bar Element Equivalent Stress vs. Effective Creep Strain.

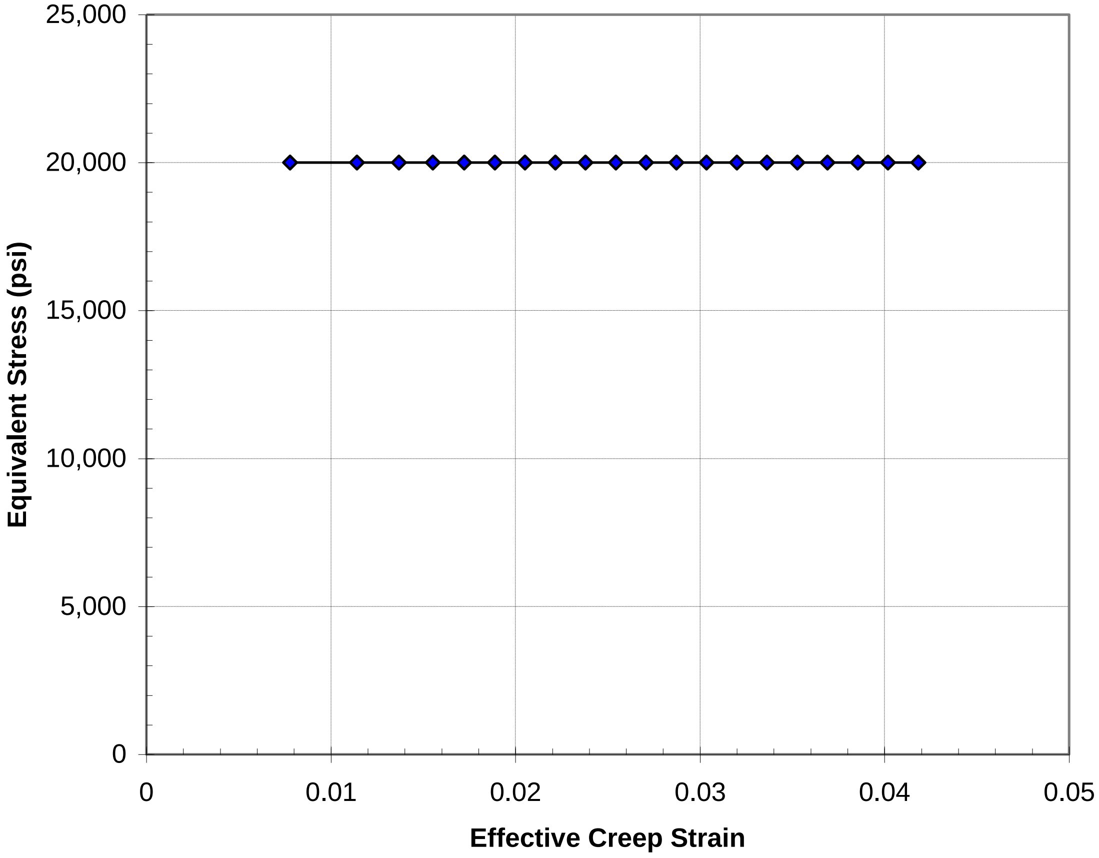
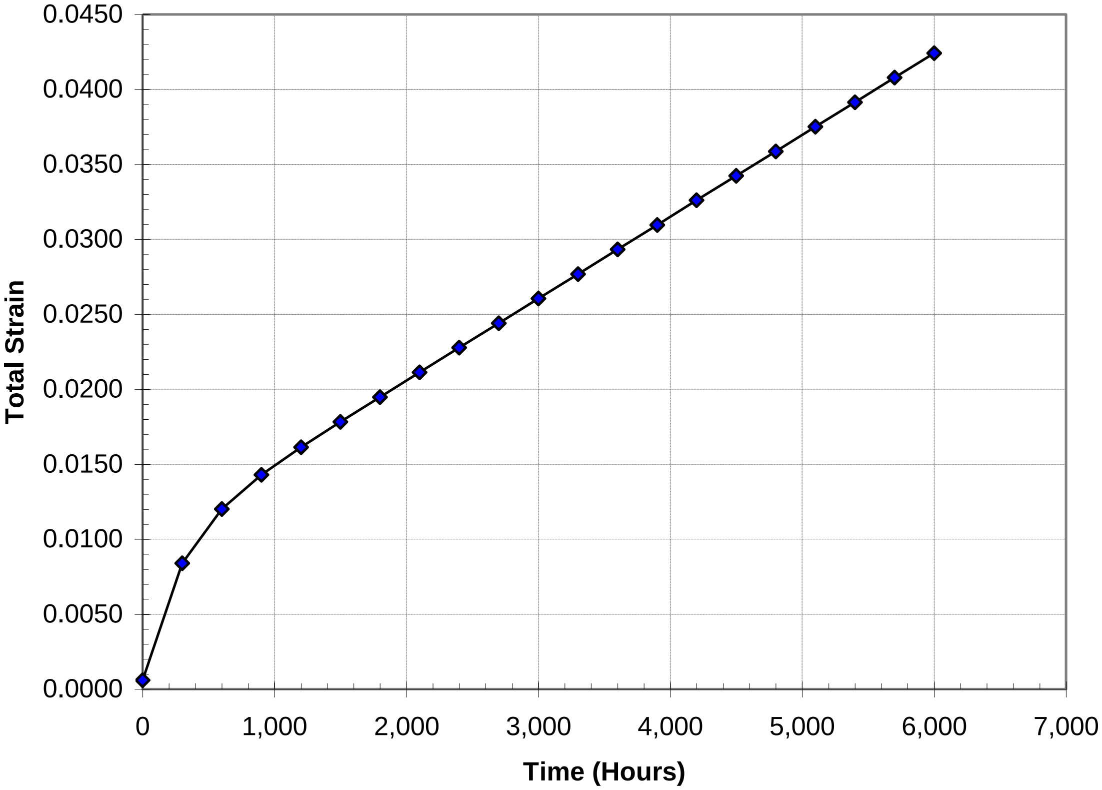


Figure 15-44. Bar Element Total Strain vs. Time.



15.3.12 Arc-length Methods

It is often necessary to study the post buckling behavior of a structure whose response is unstable during part of its loading history. Arc-length methods allow solutions in this unstable regime by modulating the applied loads in order to produce displacement increments of manageable size for a given load step. In order to modulate the applied load, an additional variable and a constraint equation are introduced.

The residual load vector $\{R\}$ is given by

$$R(u, \mu) = P(u, \mu) - F(u)$$

where F represents the internal forces and the total external load P is given by

$$P(u, \mu) = P_0 + \mu \Delta P$$

where P_0 denotes the applied load at the end of the preceding subcase, ΔP represents the load increment in the current subcase and μ is the load factor varying from 0 to 1 within the subcase. Note that the load factor is not limited to the range 0 to 1 and may become negative during load modulation.

There are three different arc-length method available in Autodesk Inventor Nastran: Crisfield, Riks, and Modified Riks. The primary difference is the constraint equation that is used to solve for μ.

For Crisfield's method (default) the constraint equation is given by

$$\left\{u_n^i - u_n^0\right\}\left\{u_n^i - u_n^0\right\} + w^2\left(\mu^i - \mu^0\right)^2 = \Delta l_n^2$$

For Riks' method the constraint equation is given by

$$\left\{u_n^i - u_n^{i-1}\right\}^T\left\{u_n^1 - u_n^0\right\} + w^2 \Delta\mu^i = 0$$

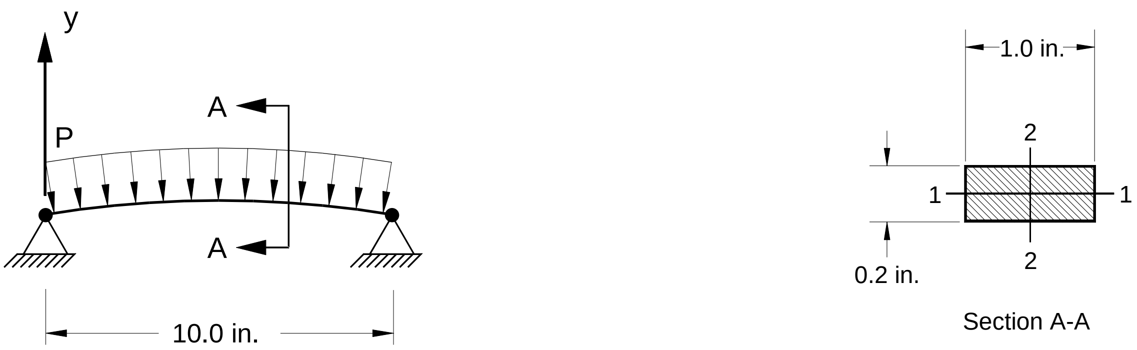
And for the Modified Riks' method the constraint equation is given by

$$\left\{u_n^i - u_n^{i-1}\right\}^T\left\{u_n^{i-1} - u_n^0\right\} + w^2 \Delta\mu^i \left(\mu^{i-1} - \mu^0\right) = 0$$

where w is the user-specified scaling factor used to handle dimension disparities
μ is the load factor
Δl is the arc-length

The constraint equation is defined using the `NLPCI` Bulk Data entry. Generally the Crisfield method performs best and is the default.

As an example consider the shallow circular arch shown in Figure 15-45. The arch is simply supported at its ends and is loaded uniformly. The objective is to find the structural response for snap-through. The material is linear elastic. Listing 15-25 contains the Model Input File.



E = 10.0 x10^{6} psi
υ = 0.33
A = 0.2 in.^{2}
I1 = 6.667 x10^{-4} in.^{4}
I2 = 1.667 x10^{-2} in.^{4}
J = 2.328 x10^{-3} in.^{4}
ρ = 0.100 lb/in.^{3}

Figure 15-45. 2-D Circular Arch Snap-Through Example Problem.

Listing 15-25. Model Input File for 2-D Circular Arch Snap-Through.

```
$
$ REQUEST X-Y PLOT OUTPUT TO A COMMA SEPERATED VARIABLE FILE.
$
NASTRAN XYPLOTCSVOUT=ON
$
$ NONLINEAR STATIC SOLUTION.
$
SOL NONLINEAR STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = 2-D CIRCULAR ARCH SNAP-THROUGH
$
DISPLACEMENT = ALL
STRESS = ALL
FORCE = ALL
SUBCASE 1
 LABEL = 200 POUND/INCH DISTRIBUTED LOAD
 LOAD = 1
 NLPARM = 1
 SPC = 1
$
$ REQUEST X-Y PLOT OUTPUT AT MODEL CENTER POINT IN Y-DIRECTION.
$
XYDATA, 6, 2
$
BEGIN BULK
$
$ TURN ON LARGE DISPLACEMENT EFFECTS.
$
PARAM, LGDISP, ON
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  5,  ,  ,  ,  ,  ,  YES
$
$ SPECIFY ARC-LENGTH METHOD.
$
NLPCI,  1
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,   1,  0,  0.,       0.,       0.,  0
GRID,   2,  0,  0.99619,  0.15688,  0.,  0
GRID,   3,  0,  1.99239,  0.27890,  0.,  0
GRID,   4,  0,  2.98858,  0.36606,  0.,  0
GRID,   5,  0,  3.98478,  0.41835,  0.,  0
GRID,   6,  0,  4.98097,  0.43578,  0.,  0
GRID,   7,  0,  5.97717,  0.41835,  0.,  0
GRID,   8,  0,  6.97336,  0.36606,  0.,  0
GRID,   9,  0,  7.96956,  0.27890,  0.,  0
GRID,  10,  0,  8.96575,  0.15688,  0.,  0
GRID,  11,  0,  9.96195,  0.,       0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
$
$ BAR ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  1.667E-2,  6.667E-4,  2.328E-3,
,  -0.5,  0.1,  0.5,  0.1,  -0.5,  -0.1,  0.5,  -0.1
$
$ LINEAR ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
```

Listing 15-25. Model Input File for 2-D Circular Arch Snap-Through. (Continued)

```
$
$ PINNED AT BOTH ENDS, MOVEMENT CONSTRAINED TO X-Y PLANE ONLY.
$
SPC1,  1,  12345,  1,  11
SPC1,  1,    345,  2,  THRU,  10
$
$ DISTRIBUTED LOAD (NEGATIVE Y-DIRECTION IN POUNDS/INCH).
$
PLOAD1,  1,  1,  FZE,  FR,  0.,  200.,  1.,  200.
PLOAD1,  1,  2,  FZE,  FR,  0.,  200.,  1.,  200.
PLOAD1,  1,  3,  FZE,  FR,  0.,  200.,  1.,  200.
PLOAD1,  1,  4,  FZE,  FR,  0.,  200.,  1.,  200.
PLOAD1,  1,  5,  FZE,  FR,  0.,  200.,  1.,  200.
PLOAD1,  1,  6,  FZE,  FR,  0.,  200.,  1.,  200.
PLOAD1,  1,  7,  FZE,  FR,  0.,  200.,  1.,  200.
PLOAD1,  1,  8,  FZE,  FR,  0.,  200.,  1.,  200.
PLOAD1,  1,  9,  FZE,  FR,  0.,  200.,  1.,  200.
PLOAD1,  1, 10,  FZE,  FR,  0.,  200.,  1.,  200.
ENDDATA
```

The maximum displacements from the Model Results Output File are shown in tabular form in Listing 15-26. The center point y-displacement is plotted against load factor in Figure 15-46 and the deflected shapes are plotted in Figure 15-47. The critical load is reached at 64% and the load must be decrease and eventually reverse direction even though continued increased displacement occurs. At -29% the load direction reverses again and the arch begins to load until the full load is reached.

Listing 15-26. Load Increment Maximum Displacements.

```
                                   M A X I M U M   D I S P L A C E M E N T S

     LOAD          LOAD             T1            T2            T3            R1            R2            R3
  INCREMENT       FACTOR
      1      2.000000E-01     1.064369E-03  2.734367E-02  0.000000E+00  0.000000E+00  0.000000E+00  9.172597E-03
      2      3.984177E-01     1.944157E-03  5.288588E-02  0.000000E+00  0.000000E+00  0.000000E+00  1.717400E-02
      3      5.484409E-01     2.713847E-03  7.913130E-02  0.000000E+00  0.000000E+00  0.000000E+00  2.451466E-02
      4      6.028619E-01     3.292669E-03  1.065737E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.042178E-02
      5      4.240425E-01     3.600876E-03  1.360917E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.486059E-02
      6      5.427737E-01     3.826832E-03  1.675944E-01  0.000000E+00  0.000000E+00  0.000000E+00  4.336880E-02
      7      6.035162E-01     3.952592E-03  1.996172E-01  0.000000E+00  0.000000E+00  0.000000E+00  5.590519E-02
      8      6.087544E-01     4.127067E-03  2.316369E-01  0.000000E+00  0.000000E+00  0.000000E+00  6.948030E-02
      9      5.958818E-01     4.401700E-03  2.634977E-01  0.000000E+00  0.000000E+00  0.000000E+00  8.231548E-02
     10      5.255065E-01     4.767492E-03  2.949992E-01  0.000000E+00  0.000000E+00  0.000000E+00  9.420700E-02
     11      4.809419E-01     5.202060E-03  3.259921E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.051385E-01
     12      4.290141E-01     5.681659E-03  3.563879E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.151531E-01
     13      3.814839E-01     6.185126E-03  3.861782E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.243602E-01
     14      3.303640E-01     6.696317E-03  4.152996E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.327593E-01
     15      2.812991E-01     7.294053E-03  4.437942E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.404487E-01
     16      2.319508E-01     8.132173E-03  4.716095E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.474345E-01
     17      1.830144E-01     8.962259E-03  4.987683E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.537691E-01
     18      1.346517E-01     9.766097E-03  5.252444E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.594637E-01
     19      8.622582E-02     1.052849E-02  5.511030E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.645481E-01
     20      3.989072E-02     1.123141E-02  5.762565E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.690182E-01
     21      5.888430E-03     1.185919E-02  6.006764E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.728701E-01
     22      5.140279E-02     1.239708E-02  6.243199E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.761000E-01
     23      9.296791E-02     1.283125E-02  6.471669E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.787020E-01
     24      1.337138E-01     1.314759E-02  6.691645E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.806610E-01
     25      1.720680E-01     1.333263E-02  6.902560E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.819603E-01
     26      2.073506E-01     1.337366E-02  7.103773E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.862311E-01
     27      2.388643E-01     1.325922E-02  7.294628E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.908725E-01
     28      2.655994E-01     1.297712E-02  7.476317E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.952105E-01
     29      2.862614E-01     1.251942E-02  7.647548E-01  0.000000E+00  0.000000E+00  0.000000E+00  1.992289E-01
     30      3.049074E-01     1.188075E-02  7.808460E-01  0.000000E+00  0.000000E+00  0.000000E+00  2.029030E-01
     31      3.040150E-01     1.106461E-02  7.960439E-01  0.000000E+00  0.000000E+00  0.000000E+00  2.062640E-01
     32      2.938784E-01     1.008549E-02  8.106727E-01  0.000000E+00  0.000000E+00  0.000000E+00  2.154939E-01
     33      3.267057E-01     8.053404E-03  8.362994E-01  0.000000E+00  0.000000E+00  0.000000E+00  2.387010E-01
     34      2.251854E-01     5.997927E-03  8.647077E-01  0.000000E+00  0.000000E+00  0.000000E+00  2.641085E-01
     35      6.510297E-02     4.314709E-03  8.972744E-01  0.000000E+00  0.000000E+00  0.000000E+00  2.877671E-01
     36      4.567181E-01     2.543297E-03  9.332464E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.097129E-01
     37      9.420496E-01     2.232125E-03  9.636789E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.263458E-01
```

Figure 15-46. Y-Displacement vs. Load Factor.

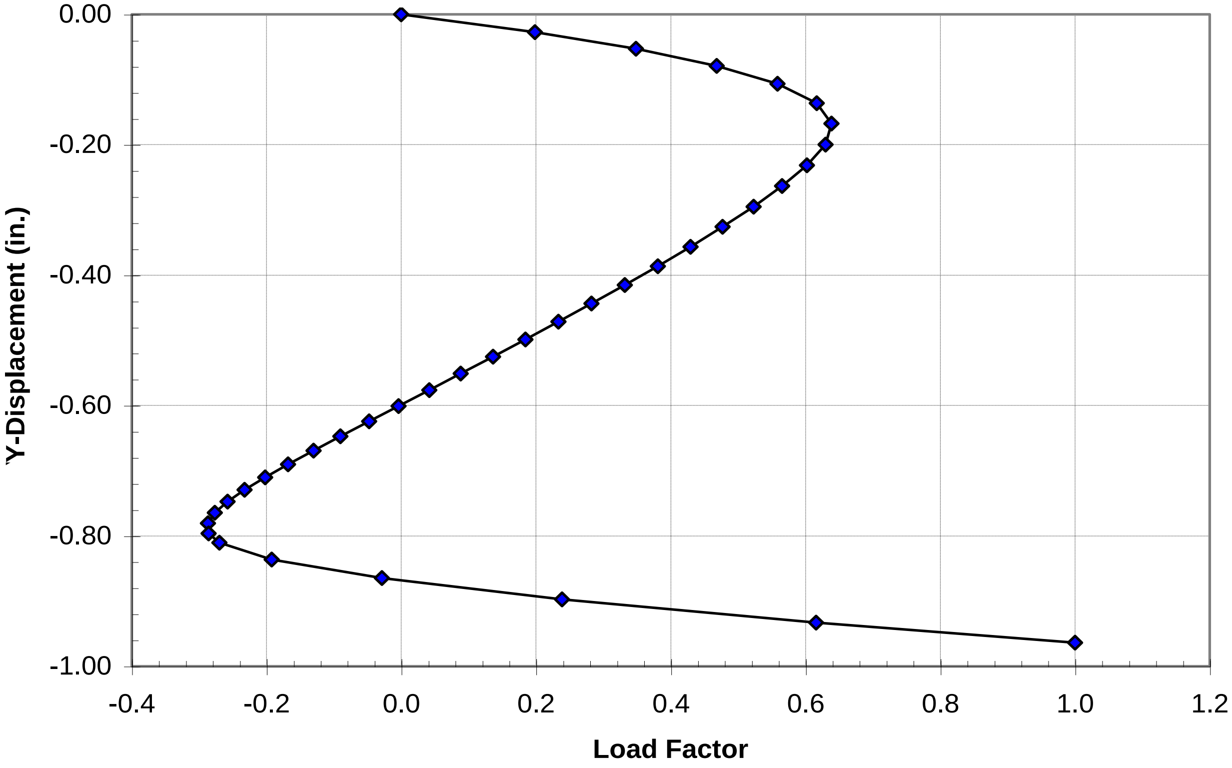
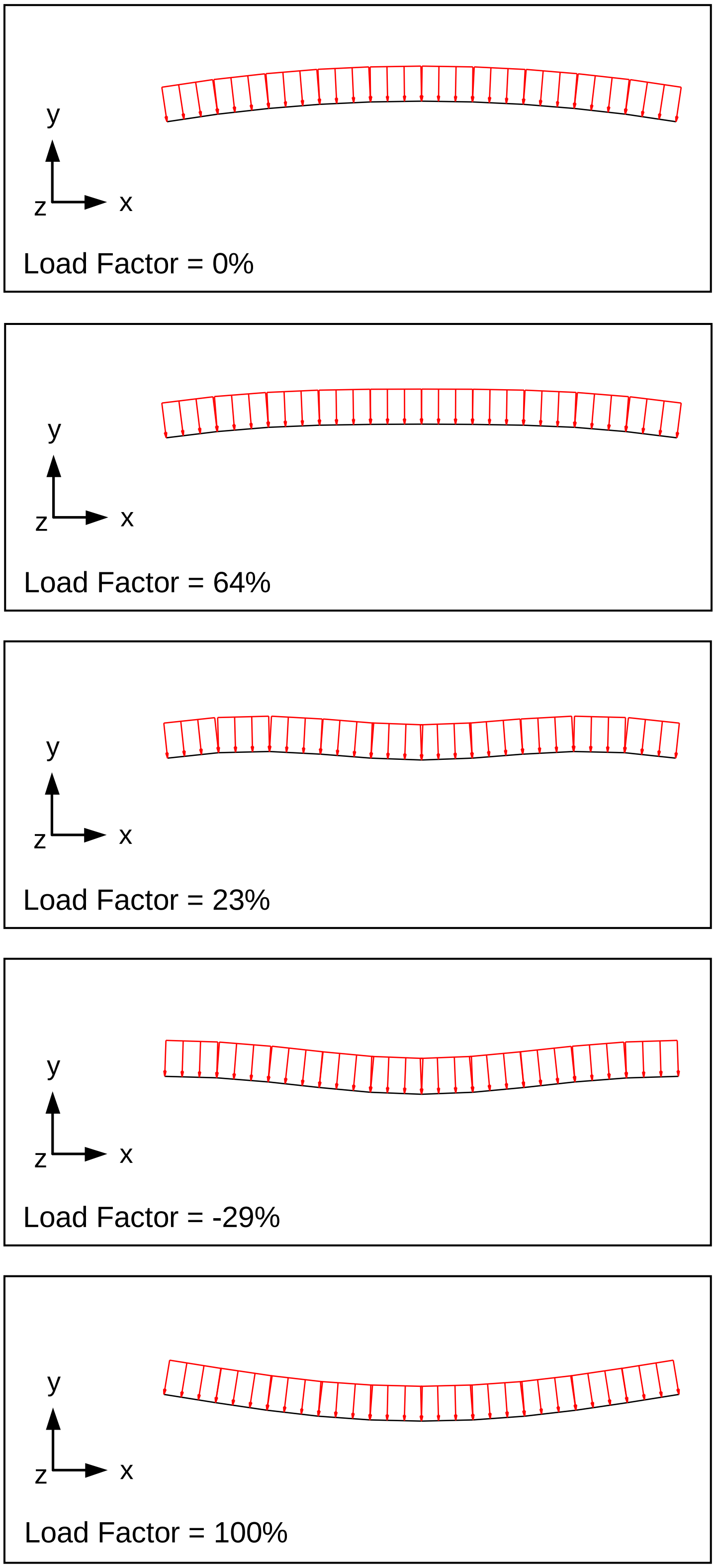


Figure 15-47. Deformed Shapes of a Cantilever Beam with an Axial End Force.



16. NONLINEAR TRANSIENT RESPONSE ANALYSIS

16.1 Introduction

In the previous section we dealt with static analysis where loads and boundary conditions did not vary with time and inertia effects were ignored. If the effects of inertia, damping, and transient loading are to be included in the nonlinear analysis, the nonlinear transient response is analyzed using a direct approach. For this approach we use a step-by-step integration of the general equation of equilibrium system in motion:

$$[M]\{\ddot{u}(t)\}+[B]\{\ddot{u}(t)\}+[K]\{u(t)\}=\{R(t)\}$$

where,

$[K]$	is the global stiffness matrix
$[M]$	is the global mass matrix
$[B]$	is the global damping matrix
$\{R\}$	is the global load vector
$\{\ddot{u}\}$	is the global acceleration vector
$\{\dot{u}\}$	is the global velocity vector
$\{u\}$	is the global displacement vector

By direct numerical integration, equilibrium is satisfied at discrete time steps with an interval of Δt . The equilibrium is obtained through nonlinear iterations until convergence is reached. Newmark’s method of direct integration is used with adaptive time stepping and bisection to improve solution efficiency. Just as in linear transient response loads can be time dependent. However unlike linear transient response nonlinear effects like large displacements and rotation, material nonlinearity, and contact can be included. Convergence is achieved at each time step using the same Newton-Raphson iteration method used in nonlinear static analysis.

16.2 How to Setup a Model Input File for Nonlinear Transient Response Analysis

In Autodesk Inventor Nastran you can solve a nonlinear statics problem by setting `SOLUTION = NONLINEAR TRANSIENT RESPONSE` in the Model Initialization File or by specifying `SOL 129` or `SOL NONLINEAR TRANSIENT RESPONSE` above the Case Control Section in the Model Input File, and following the guidelines listed below:

1. Most nonlinear transient response problems can be setup the same as for linear transient response (geometry, boundary conditions, loading, etc.). As a minimum, all subcases must reference a `TSTEPNL` Bulk Data entry via the `TSTEPNL` Case Control Command. The `TSTEPNL` entry is a combination of the `TSTEP` entry used in linear transient response and the `NLPARM` entry used in nonlinear statics. It controls both the direct time integration (number of time steps, time increment, output interval, etc.), and the nonlinear iteration parameters (maximum iterations permitted, convergence method and tolerances, etc.). Since the solution to a particular load involves a nonlinear search procedure, the solution is not guaranteed. Care must be used when selecting the search procedures on the `TSTEPNL` Bulk Data entry. You may override nearly all iteration control restrictions.
2. All loads, boundary conditions, elements (except `CSHEAR`), element properties (except `PCOMP` with material nonlinearity), and material properties that are supported in linear transient response analysis are supported in nonlinear transient response.
3. For contact solutions, gap (`CGAP)` or slide line (`BCONP`) elements must be specified. Contact elements can be used with all loads, boundary conditions, elements and types of nonlinearity supported. Note that for gap elements, contact planes do not rotate as a function of displacement. The user-specified stiffnesses (`KA`, `KB`, and `KT` on the `PGAP` Bulk Data entry) must be carefully selected when the non-adaptive form is used (`TMAX` ≤ 0.0 on the `PGAP` Bulk Data entry). An optimal selection of values is usually a compromise between accuracy and numerical performance. Slide line and surface contact elements do rotate as a function of displacement, if large displacement effects are turned on (`PARAM`, `LGDISP`, `ON`), and allow elements to slide past each other.
4. Follower forces (forces that follow the deformed geometry) are generated automatically when using element pressures (`PLOAD1`, `PLOAD2`, and `PLOAD4`), element temperatures (`TEMP`, `TEMPD`, `TEMPP1`, and `TEMPRB`), acceleration loads (`GRAV` and `RFORCE`), and grid point forces and moments (`FORCE1` and `MOMENT1`). Follower force effects are controlled using the `LGDISP` parameter.
5. Constraints apply only to the nonrotated displacements at a grid point. In particular, multipoint constraints and rigid elements may cause problems if the connected grid points undergo large motions. However, also note that replacement of the constraints with overly stiff elements may result in convergence problems.
6. Large deformations of elements may cause nonequilibrium loading effects. All elements are assumed to have constant length, area, and volume. Large displacement effects are controlled using the `LGDISP` parameter.
7. In large displacement analysis there are two different approaches for the angular motions: gimbal angle and rotation vector. In the gimbal angle approach, angular motions are treated as three ordered rotations about the x, y, and z-axes. The gimbal angle approach is requested by specifying `PARAM`, `LANGLE`, `1` (default) in the Model Input File. In the rotation vector approach, the three angular motions are treated as a vector. The rotation is about the rotation axis and the magnitude of rotation is equal to the amplitude of the rotation vector. The rotation vector approach is requested by specifying `PARAM`, `LANGLE`, `2` in the Model Input File.

8. Material nonlinear solutions require a `MATS1` Bulk Data entry be specified for elements that have nonlinear material properties. Both linear and nonlinear materials may be specified in the same solution. Material nonlinear properties can be used with all loads, boundary conditions, elements and types of nonlinearity supported. Beam, bar, and rod elements support material nonlinearity only in the axial direction. Better performance may be achieved when using quad elements and elastic-plastic materials if `PARAM`, `QUADINODE` is set to `OFF` and `PARAM`, `QUADRNODE` is set to `ON`.
9. The use of `CQUADR` and `CTRIAR` elements are preferred over the use of `PARAM`, `K6ROT` when large displacements effects are turned on (`PARAM`, `LGDISP`, `ON`). If `PARAM`, `K6ROT` is set to a value greater than zero, only the work convergence criteria (`W`) on the `NLPARM` Bulk Data entry should be used.
10. Unlike other solutions, subcase loads and results are additive. This allows different loads and boundary conditions to be applied in a specific sequence to the structure. Additionally, different time integration and nonlinear iteration parameters (`TSTEPNL`) may be specified for each subcase allowing further control.
11. Models should be simple and relatively small initially to gain insight into behavior and verify the approach taken. A linear static or transient solution should be run first to verify boundary conditions and loading. Large displacement and follower force effects can be turned off by setting `PARAM`, `LGDISP` to `OFF`.

16.3 Interpreting Results

In this section we will present several examples demonstrating the features and capabilities of nonlinear transient response analysis. We will look at three types of nonlinearity: geometric (large displacement and rotation), material (nonlinear elastic and elastic-plastic), and contact.

16.3.1 Impact Analysis

The first problem is an example of large displacement and contact that are involved in impact analysis. A block at rest is dropped from initial height of 5 inches and is accelerated by gravity until impacting the tip of a cantilever beam as shown in Figure 16-1. The block then bounces back and the beam resonates. Gap elements are used to represent the contact between the beam and block. Structural damping is applied based on the resonant frequency of the beam (63.2 Hz). Listing 16-1 contains the Model Input File.

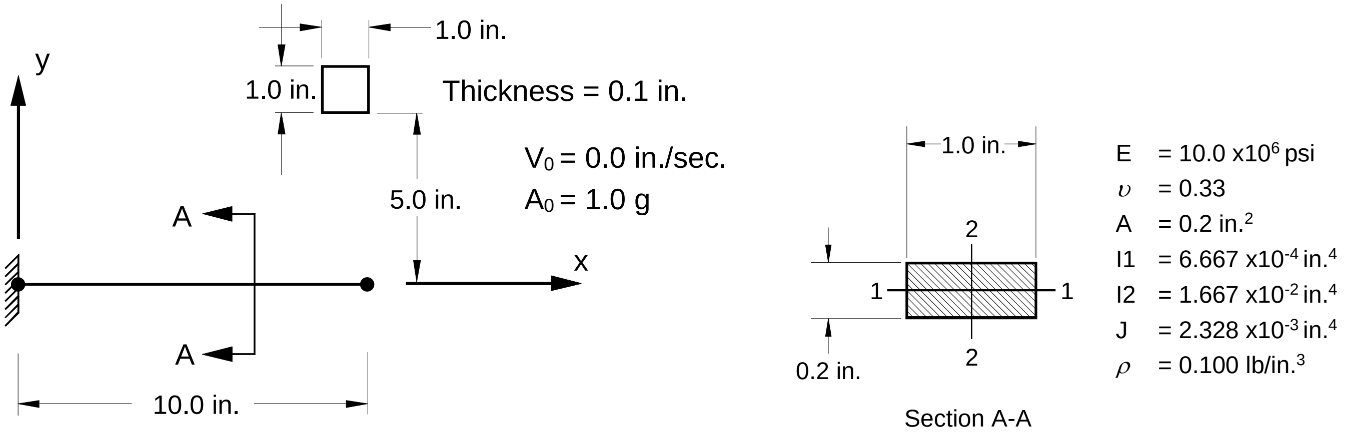


Figure 16-1. 2-D Cantilever Beam Example Problem with Impact.

Listing 16-1. Model Input File for the Cantilever Beam Problem with Impact.

```
$
$ NONLINEAR TRANSIENT RESPONSE SOLUTION.
$
SOL NONLINEAR TRANSIENT RESPONSE
CEND
$
DISPLACEMENT = ALL
FORCE = ALL
STRESS = ALL
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = NONLINEAR DYNAMIC RESPONSE - IMPACT
$
LOADSET = 10
$
SUBCASE 1
 LABEL = 1 G ACCELERATION IN NEGATIVE Y-DIRECTION
 DLOAD = 1
 TSTEPNL = 1
 SPC = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST COUPLED MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, ON
$
$ DEFINE STRUCTURAL DAMPING FREQUENCY OF INTEREST (63.2Hz).
$
PARAM, W4, 397.1
$
$ TURN ON LARGE DISPLACEMENT EFFECTS.
$
PARAM, LGDISP, ON
$
$ DEFINE NONLINEAR TRANSIENT SOLUTION PARAMETERS.
$
TSTEPNL,  1, 2500,  0.0001
$
$ DEFINE LOADING.
$
DLOAD,  1,  1.,  1.,  11
$
$ DEFINE TIME-DEPENDENT LOADING -CONSTANT.
$
TLOAD1,  11,  100,  ,  ,  10
TABLED1, 10,
  ,  0.,  1.,  1.,  1.,  ENDT
$
$ 1 G ACCELERATION IN NEGATIVE Y-DIRECTION
$
GRAV,  1,  ,  1.,  0.,  -1.,  0.
LSEQ,  10,  100,  1
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
```

Listing 16-1. Model Input File for the Cantilever Beam Problem with Impact. (Continued)

```
$
$ GEOMETRY DEFINITION (1" X 1" BLOCK WITH A 1 X 1 MESH).
$
GRID, 13,  0,   9.,  6.,  0.,  0
GRID, 14,  0,   9.,  5.,  0.,  0
GRID, 15,  0,  10.,  6.,  0.,  0
GRID, 16,  0,  10.,  5.,  0.,  0$
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
$
$ GAP ELEMENTS.
$
CGAP, 11,  20,  10,  14,  1.,  0.,  0.
CGAP, 12,  20,  11,  16,  1.,  0.,  0.
$
$ BLOCK MODELED WITH A SHELL ELEMENT.
$
CQUADR, 13,  30,  15,  16,  14,  13
$
$ ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  1.667E-2,  6.667E-4,  2.328E-3,
,  -0.5,  0.1,  0.5,  0.1,  -0.5,  -0.1,  0.5,  -0.1
$
$ GAP ELEMENT PROPERTIES.
$
PGAP,  20,  -1.,  0.,  1.E+7,  0.,  0.
$
$ ELEMENT MATERIAL AND THICKNESS (1.0").
$
PSHELL,  30,  200,  1.,  200,  ,  200
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM) WITH 4% CRITICAL DAMPING
$ (2*C/C0).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1,  ,  ,  0.08
$
$ ELEMENT MATERIAL PROPERTIES (STEEL).
$
MAT1,  200,  3.E+7,  ,  0.33,  0.3
$
$ FIXED AT ONE END, MOVEMENT CONSTRAINED TO X-Y PLANE ONLY, ONE EDGE
$ OF BLOCK CONSTRAINED FROM ROTATION.
$
SPC1,  1,  123456,  1
SPC1,  1,     345,  1,  THRU,  14
SPC1,  1,   13456,  15,  16
ENDDATA
```

The `TSTEPNL` entry controls the direct time integration and nonlinear iteration parameters. The specified number of time steps and time increment are initial values that define the duration of the analysis. The default integration method is an adaptive approach which automatically adjusts the time increment based on changes in the model's dominant frequency.

Figure 16-2 compares the beam tip and block displacement and velocity versus time. The deflected shapes are plotted in Figure 16-3.

Figure 16-2a. Beam Tip and Block Displacement Versus Time.

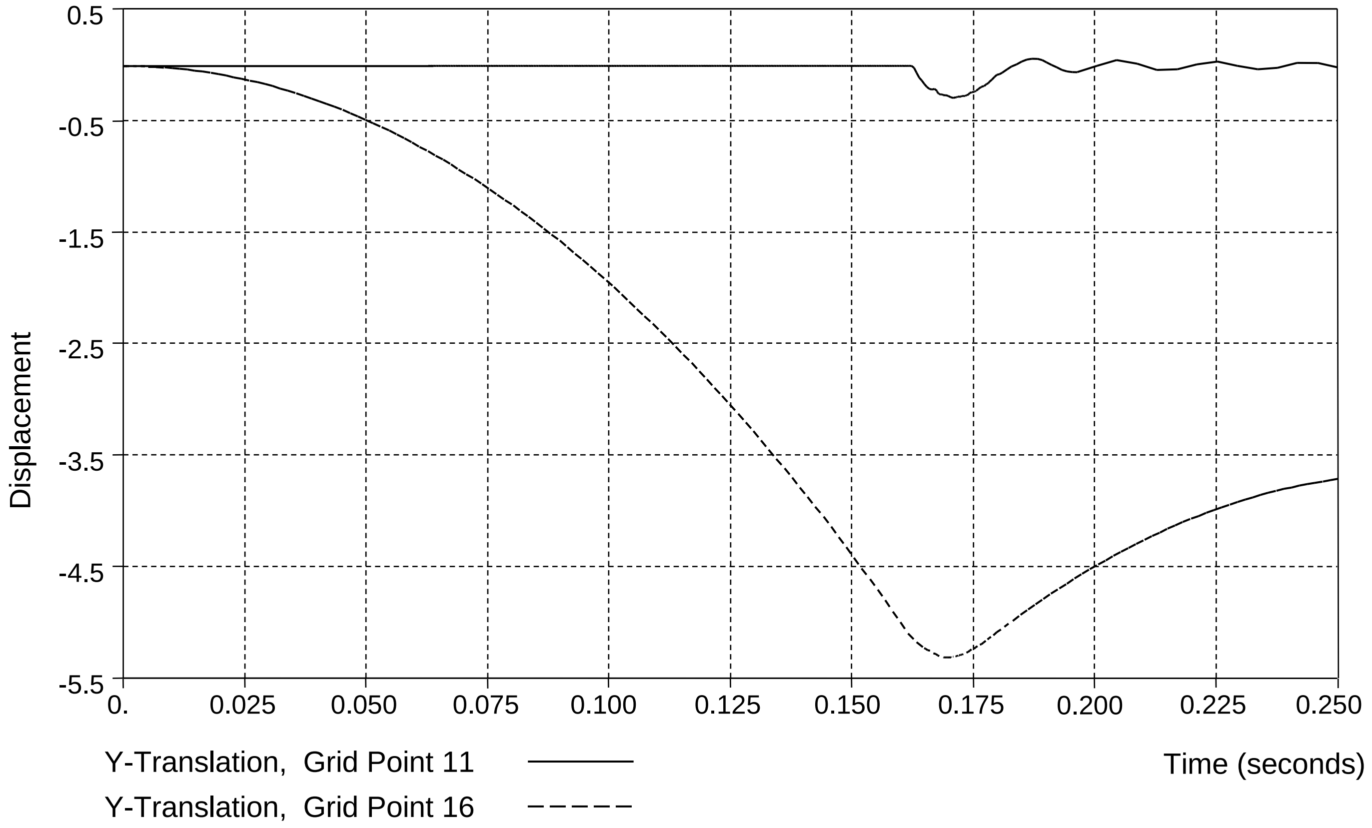


Figure 16-2b. Beam Tip and Block Velocity Versus Time.

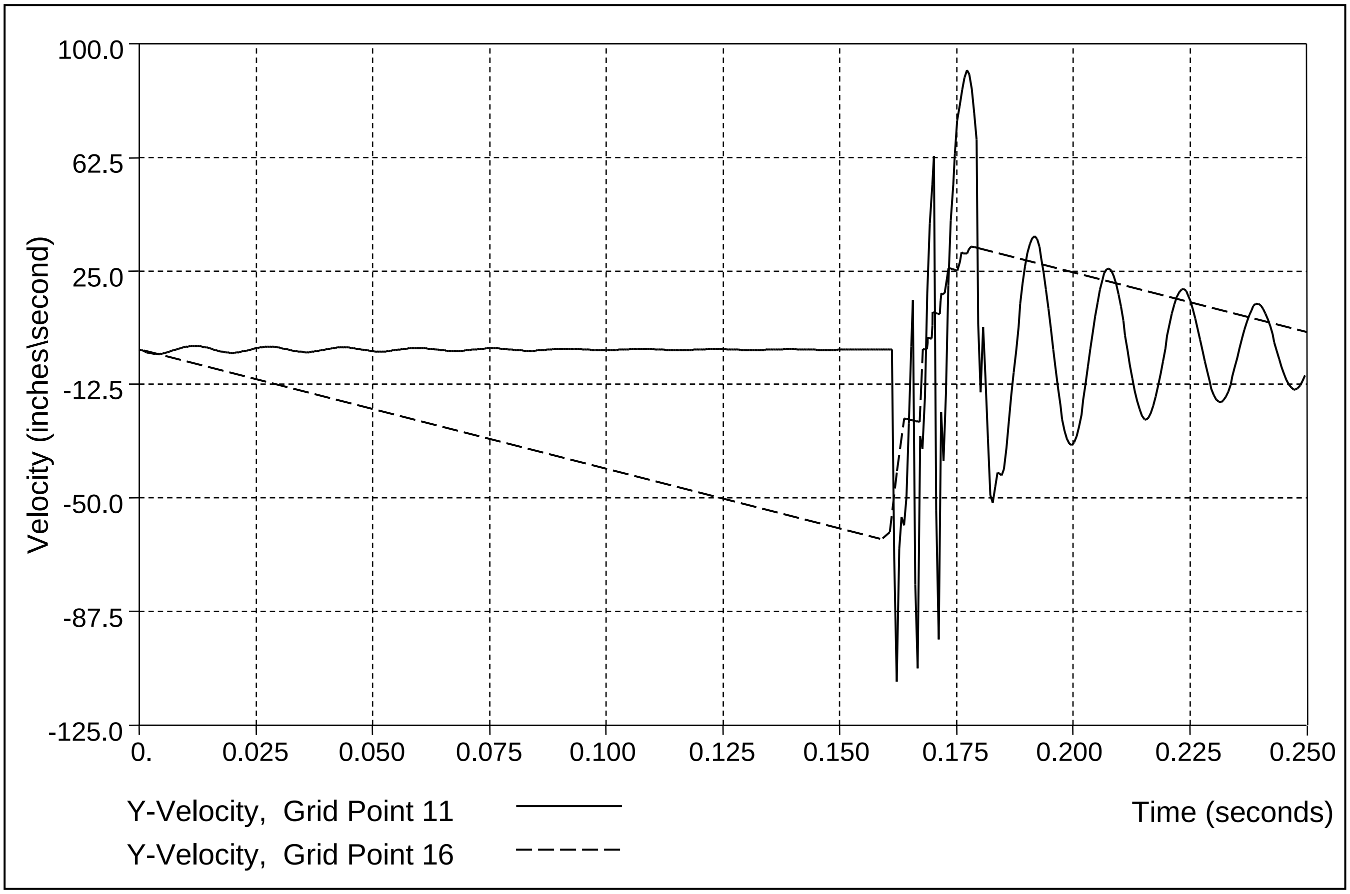


Figure 16-3. Deformed Shapes of a Cantilever Beam with Impact.

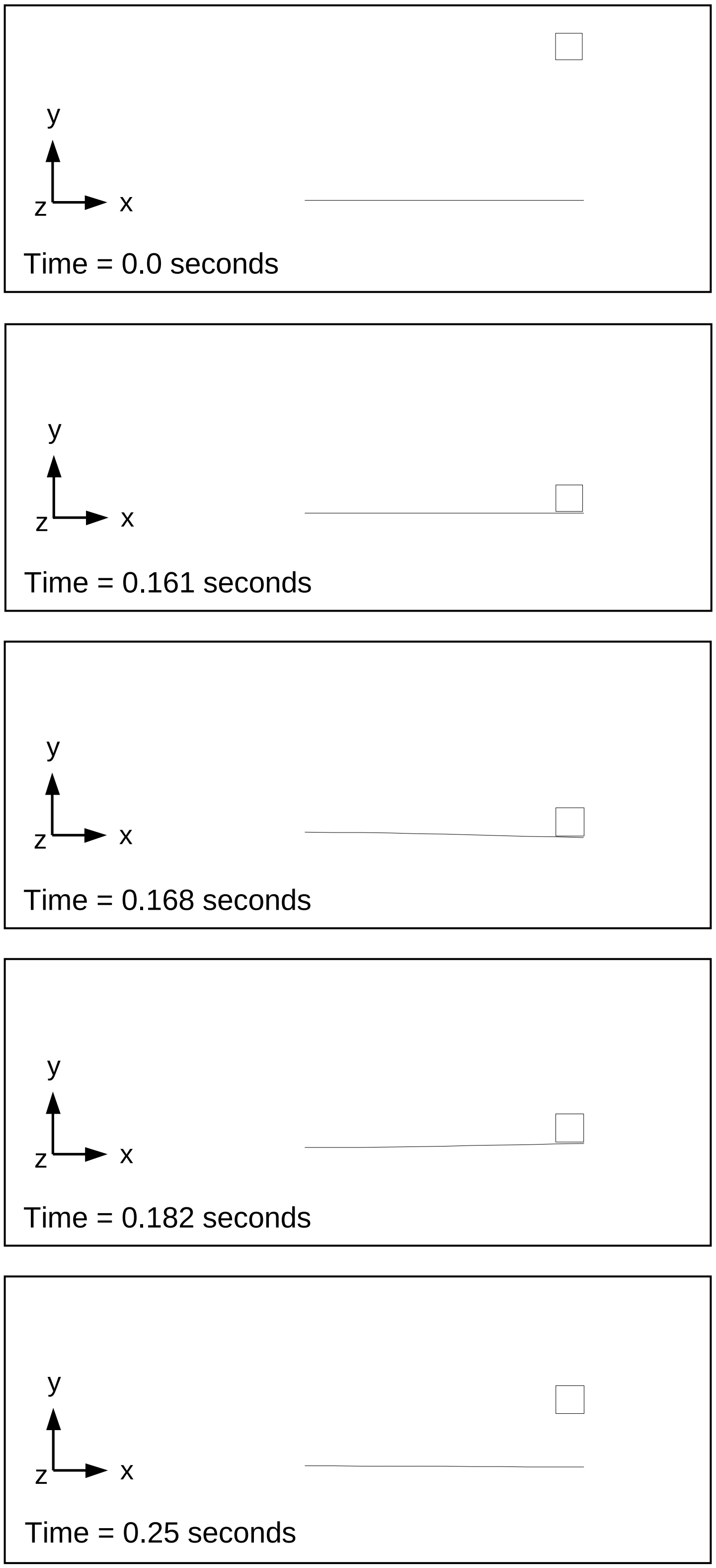


Table 16-1 shows a comparison between Autodesk Inventor Nastran and the theoretical time to impact and the velocity at impact. The formulas for a body starting at rest are:

$$t_{\text{impact}} = \sqrt{\frac{2d}{g}}$$

$$V_{\text{impact}} = 2gd$$

where,

d	is the distance transversed
g	is the gravitational acceleration

Table 16-1a. Comparison of Theoretical Versus Predicted Time to Impact.

Theoretical (sec)	Autodesk Inventor Nastran (sec)	Difference (%)
0.1608	0.1607	0.0

Table 16-1b. Comparison of Theoretical Versus Predicted Velocity at Impact.

Theoretical (sec)	Autodesk Inventor Nastran (sec)	Difference (%)
-62.16	-61.32	1.3

16.3.2 Elastic-Plastic Material with Rupture

The next problem is an example of material nonlinearity in a transient response solution. Figure 16-4 shows a cantilever beam with a large mass at its free end. The beam is subjected to a step load of varying magnitudes: $0.5F_y$, $0.66F_y$, $0.88F_y$, and F_y where F_y is the axial force corresponding to the initial yield of the rod. As a result, the mass responds with different patterns. Listing 16-2 contains the Model Input File for the first loading condition.

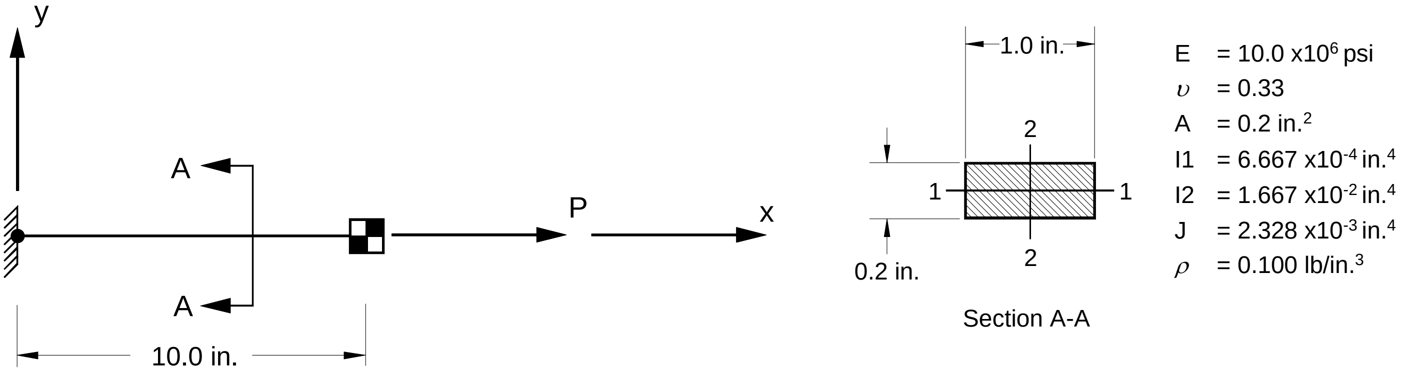


Figure 16-4. 2-D Cantilever Beam Example Problem an Elastic-Plastic Material.

Listing 16-2. Model Input File for the Cantilever Beam Problem with an Elastic-Plastic Material.

```
$
$ NONLINEAR TRANSIENT RESPONSE SOLUTION.
$
SOL NONLINEAR TRANSIENT RESPONSE
CEND
$
DISPLACEMENT = ALL
STRESS = ALL
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = NONLINEAR DYNAMIC RESPONSE WITH ELASTIC-PLASTIC MATERIAL
$
LOADSET = 10
$
SUBCASE 1
 LABEL = 25,000 LB MASS AT TIP, AXIAL LOAD AT 0.5*Fy
 DLOAD = 1
 TSTEPNL = 1
 SPC = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ DEFINE STRUCTURAL DAMPING FREQUENCY OF INTEREST (63.2Hz).
$
PARAM, W4, 397.1
$
$ TURN ON LARGE DISPLACEMENT EFFECTS.
$
PARAM, LGDISP, ON
$
$ DEFINE NONLINEAR TRANSIENT SOLUTION PARAMETERS.
$
TSTEPNL,  1, 50,  0.005
$
$ DEFINE LOADING (0.5*Fy).
$
DLOAD,  1,  0.5,  1.,  11
$
$ DEFINE TIME-DEPENDENT LOADING -CONSTANT.
$
TLOAD1,  11,  100,  ,  ,  10
TABLED1, 10,
  ,  0.,  1.,  1.,  1.,  ENDT
$
$ TENSILE LOAD (X-DIRECTION).
$
FORCE,  1,  11,  0,  6.E+3,  1.,  0.,  0.
LSEQ,  10,  100,  1
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
```

Listing 16-2. Model Input File for the Cantilever Beam Problem with an Elastic-Plastic Material. (Continued)

```
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
$
$ 25,000 LB MASS.
$
CONM2, 11,  11,  ,  2.5E+4
$
$ BAR ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  1.667E-2,  6.667E-4,  2.328E-3,
,  -0.5,  0.1,  0.5,  0.1,  -0.5,  -0.1,  0.5,  -0.1
$
$ LINEAR ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ ELASTIC-PLASTIC ELEMENT MATERIAL PROPERTIES.
$
MATS1,  100,  ,  PLASTIC,  0.,  ,  ,  3.0E+4
$
$ FIXED AT ONE END, FREE TO TRANSLATE IN X-DIR AT OTHER END.
$
SPC1,  1,  123456,   1
SPC1,  1,   23456,   2,  THRU,  11
ENDDATA
```

The theoretical solution can be considered in three different regimes: elastic, plastic, and unloading. In the elastic regime the solution can be obtained by

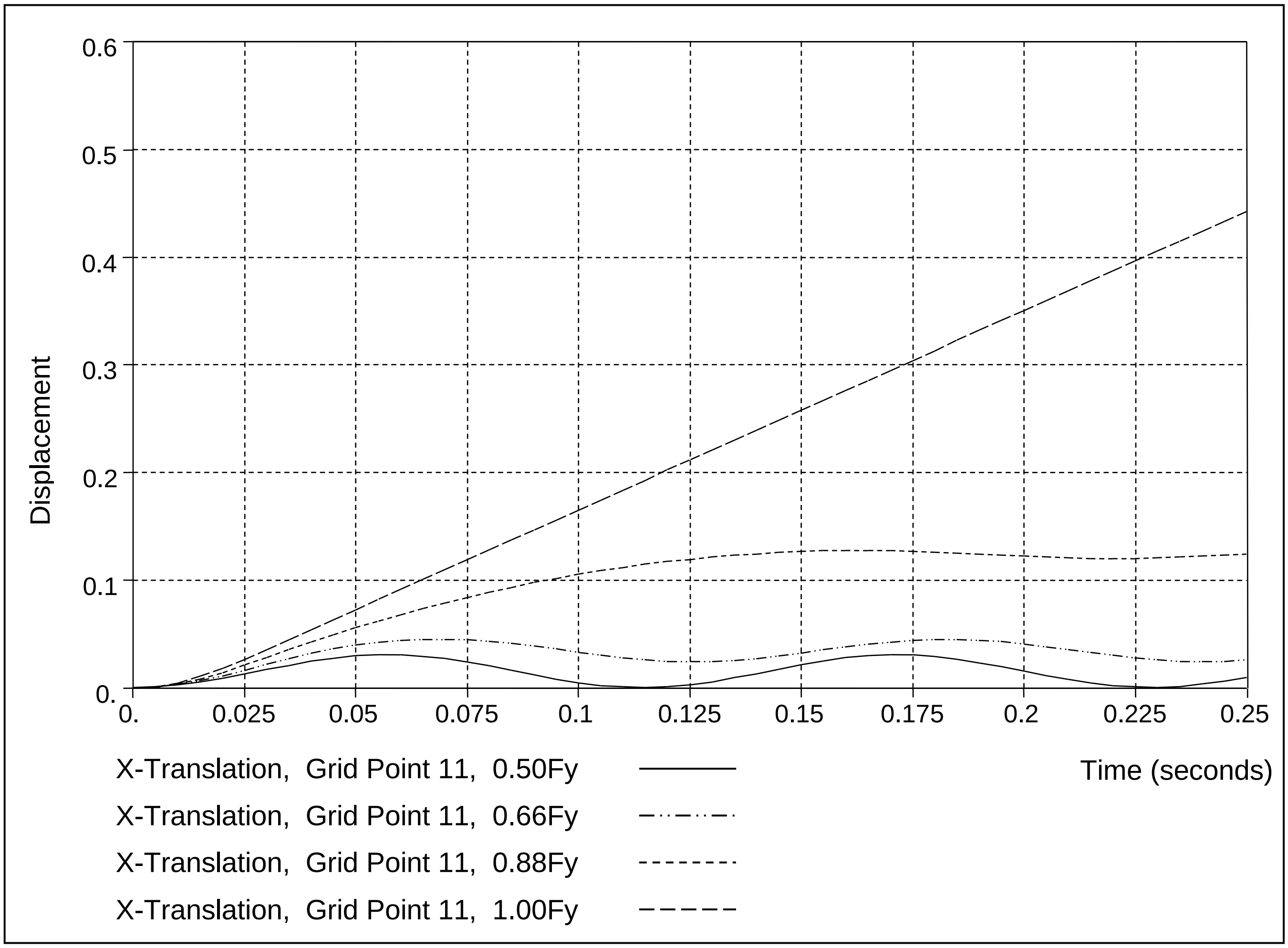
$$u = \frac{P}{k}(1 - \cos\omega t)$$

which is valid for $P \le 0.5F_y$. For $0.5F_y < P < F_y$, there is a point in time where the stress in the rod reaches the yield point. During plastic deformation, the solution may be obtained from

$$M\ddot{u} + F_y = P$$

with the correct initial conditions. When the displacement reaches the maximum value, the system is governed by an elastic equation for rebound. If $P > F_y$, no rebound occurs and the mass will separate from the beam. Figure 16-5 gives the beam tip displacement versus time for the four different loading conditions run.

Figure 16-5. Beam Tip Displacement Versus Time.



17. NONLINEAR STEADY STATE HEAT TRANSFER ANALYSIS

17.1 Introduction

Of principal interest in heat transfer analysis is the temperature distribution within a solid body. A byproduct of temperature calculation is information about the magnitude and direction of heat flow in the body. Temperature and temperature gradients are an important cause of stress in structures.

Heat is transferred to or from a body by convection and radiation (Figure 17-1). Heat flow across a boundary is analogous to a surface load (pressure) in structural analysis. Additionally, there may be internal heat generation, produced by electric current, dielectric heating, or other sources. A distributed internal heat source is analogous to a body force (gravity) in stress analysis. Points may have prescribed temperatures either on the boundary or within. Prescribed temperatures are analogous to prescribed displacements (single point constraints). Heat moves within the body through conduction. For a linear steady state (time independent) problem:

$$[K]\{T\}=\{R\}$$

where,

$[K]$ is the global conductivity matrix

$\{T\}$ is the global temperature vector

$\{R\}$ is the global thermal load vector

The global conductivity matrix depends on the conductivity of the material and is usually a function of temperature. If present, convection and radiation boundary conditions contribute terms to both the conductivity matrix and the thermal load vector. The solution yields a grid point temperature distribution within the solid and is analogous to the solution of displacements in a structural analysis. However, one of the major differences between heat transfer and structural analysis is that temperature is a scalar, whereas displacement is a vector which Autodesk Inventor Nastran assumes may have as many as six components. An important feature of Autodesk Inventor Nastran is that the same model used for heat transfer analysis can be used for thermal stress analysis, where the grid point thermal loading is generated directly from the heat transfer solution (Section 17.5).

As mentioned above, thermal conductivity and other properties, may depend on temperature strongly enough that $[K]$ must be regarded as a function of temperature rather than a matrix of constants. The nonlinear properties permitted with Autodesk Inventor Nastran are temperature dependent material conductivity, temperature dependent free convection heat transfer coefficient, and temperature dependent volume heat addition. For these types of problems, a nonlinear solution is recommended.

Nonlinear static analysis is implemented in Autodesk Inventor Nastran as an iterative process using the Newton-Raphson method where the path dependent problem is broken down into several linear steps. The equilibrium equations in incremental form can be written as:

$$[K_t]\{\Delta T\}=\{\Delta R\}$$

where,

$[K_t]$ is the global tangent conductivity matrix

$\{\Delta T\}$ is the global incremental temperature vector

$\{\Delta R\}$ is the global incremental thermal load vector

The global tangent conductivity matrix $\left[K_t\right]$ is a function of the global temperatures $\{T\}$ because the material conductivity and free convection heat transfer coefficient are temperature dependent. The current global temperature vector is the sum of the preceding $\{\Delta T\}$'s.

The iterative process allows Autodesk Inventor Nastran to solve many nonlinear heat transfer problems. Several examples are given in this section which demonstrate how to setup, run, and interpret results for these types of problems.

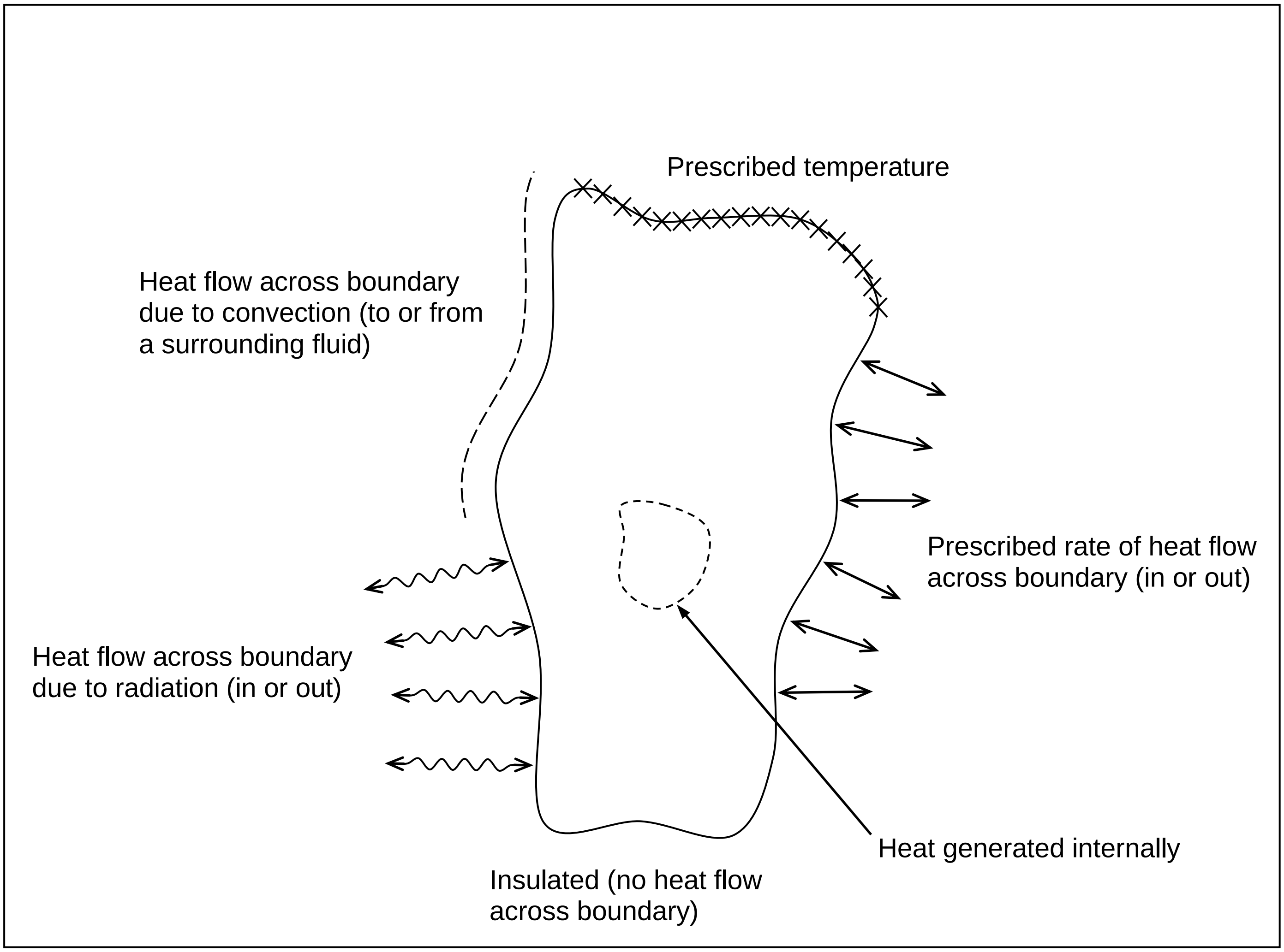


Figure 17-1. Thermal Loads and Boundary Conditions into an Arbitrary Solid.

17.2 How to Setup a Model Input File for Nonlinear Heat Transfer Analysis

In Autodesk Inventor Nastran you can solve a nonlinear heat transfer problem by setting `SOLUTION = NONLINEAR STEADY STATE HEAT TRANSFER` in the Model Initialization File or by specifying `SOL 153` or `SOL NONLINEAR STEADY STATE HEAT TRANSFER` above the Case Control Section and `ANALYSIS = HEAT` in the Case Control Section of the Model Input File. The following the guidelines listed below:

1. Most nonlinear heat transfer problems can be setup the same as for linear heat transfer (geometry, boundary conditions, loading, etc.). As a minimum, all subcases must reference an `NLPARM` Bulk Data entry via the `NLPARM` Case Control Command. The `NLPARM` entry controls the nonlinear iteration parameters (increment size, number of increments, output control, etc.). A load increment of one (default) is recommended. Since the solution to a particular load involves a nonlinear search procedure, the solution is not guaranteed. Care must be used when selecting the search procedures on the `NLPARM` Bulk Data entry. You may override nearly all iteration control restrictions.
2. All loads and material properties that are supported in linear heat transfer analysis are supported in nonlinear heat transfer analysis.
3. All grid points must have an initial temperature defined. The `TEMPD` Bulk Data entry can be used for this purpose.
4. The iterative solution may be sensitive to the initial temperature for highly nonlinear problems. It is recommended to set the initial temperature vector high for radiation dominated problems.
5. Unlike other solutions, subcase loads and results are additive. This allows different loads and boundary conditions to be applied in a specific sequence to the structure. Additionally, different nonlinear iteration parameters (`NLPARM`) may be specified for each subcase allowing further control.
6. Incremental loading reduces the imbalance of the equilibrium equation caused by applied loads. If the solution takes more iterations than the default value for the maximum number of iterations allowed for convergence, the increment size should be reduced.
7. Models should be simple and relatively small initially to gain insight into behavior and verify the approach taken. A linear heat transfer solution should be run first to verify boundary conditions and loading.

17.3 How to Setup a Model Input File for Linear Steady State Heat Transfer Analysis

In Autodesk Inventor Nastran you can solve a linear heat transfer problem by setting `SOLUTION = LINEAR STEADY STATE HEAT TRANSFER` in the Model Initialization File or by specifying `SOL 101` or `SOL LINEAR STEADY STATE HEAT TRANSFER` above the Case Control Section and `ANALYSIS = HEAT` in the Case Control Section of the Model Input File. Linear steady state heat transfer is only recommended if material property temperature dependence is mild. The initial temperature distribution, specified using the `TEMPERATURE(INITIAL)` Case Control command, is used to define a constant temperature for material property generation.

17.4 Interpreting Results

In this section we will present several examples demonstrating the features and capabilities of nonlinear steady state heat transfer analysis. We will look at several types of thermal loading and boundary conditions.

17.4.1 Nonlinear Conduction

The first problem is an example of nonlinear conduction. The circular bar in Figure 17-2 has prescribed temperatures at each end and is completely insulated over the rest of its surface area. Listing 17-1 contains the Model Input File.

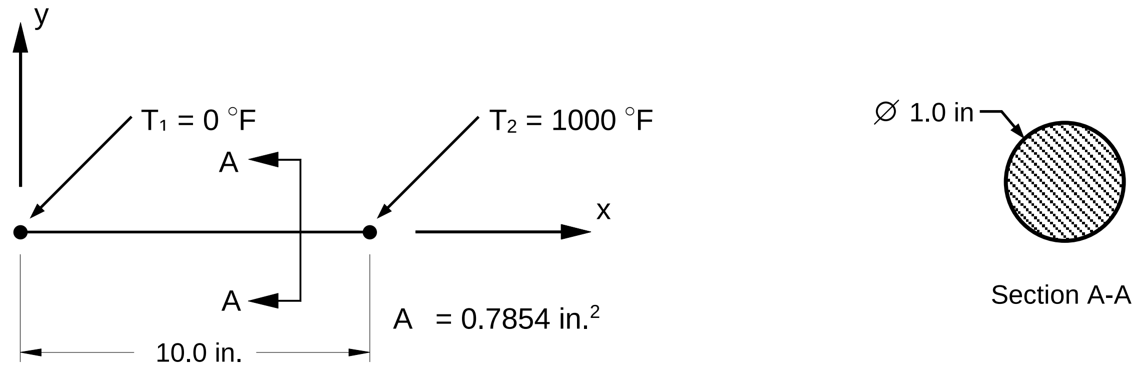


Figure 17-2. 1-Dimensional Bar Example Problem with Constrained End Temperatures.

Listing 17-1. Model Input File for Bar Model with Constrained End Temperatures.

```
$
$ NONLINEAR STEADY STATE HEAT TRANSFER SOLUTION.
$
SOL NONLINEAR STEADY STATE HEAT TRANSFER
$
ANALYSIS = HEAT
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = BAR WITH NONLINEAR TEMPERATURE DEPENDENT MATERIAL
$
THERMAL = ALL
SPCFORCE = ALL
FLUX = ALL
$
TEMPERATURE(INITIAL) = 1
SUBCASE 1
 LABEL = CONSTRAINED END TEMPERATURES
 NLPARM = 1
 SPC = 1
$
BEGIN BULK
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  1
$
$ GEOMETRY DEFINITION (10" BAR DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  ,   0.,  0.,  0.,
GRID,  2,  ,   1.,  0.,  0.,
GRID,  3,  ,   2.,  0.,  0.,
GRID,  4,  ,   3.,  0.,  0.,
GRID,  5,  ,   4.,  0.,  0.,
GRID,  6,  ,   5.,  0.,  0.,
GRID,  7,  ,   6.,  0.,  0.,
GRID,  8,  ,   7.,  0.,  0.,
GRID,  9,  ,   8.,  0.,  0.,
GRID, 10,  ,   9.,  0.,  0.,
GRID, 11,  ,  10.,  0.,  0.,
$
$ CIRCULAR BAR MODELED WITH ROD ELEMENTS.
$
CROD,  1,  10,   1,   2
CROD,  2,  10,   2,   3
CROD,  3,  10,   3,   4
CROD,  4,  10,   4,   5
CROD,  5,  10,   5,   6
CROD,  6,  10,   6,   7
CROD,  7,  10,   7,   8
CROD,  8,  10,   8,   9
CROD,  9,  10,   9,  10
CROD, 10,  10,  10,  11
$
$ ROD ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" DIAMETER).
$
PROD,  10,  100,  0.7854
```

Listing 17-1. Model Input File for Bar Model with Constrained End Temperatures. (Continued)

```
$
$ ELEMENT CONDUCTIVITY (MA956).
$
MAT4,  100,  1.
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
MATT4,  100,  10
$
$ TEMPERATURE DEPENDENT CONDUCTIVITY DATA.
$
TABLEM2,  10,
,    70.,  1.466E-4,   200.,  1.620E-4,   400.,  1.852E-4,   600.,  2.083E-4,
,   800.,  2.315E-4,  1000.,  2.508E-4,  1200.,  2.739E-4,  1400.,  2.951E-4,
,  1600.,  3.164E-4,  1800.,  3.376E-4,  ENDT
$
$ CONSTRAIN BAR END TEMPERATURES.
$
SPC,  1,   1,  1,  0.
SPC,  1,  11,  1,  1000.
$
$ INITIAL TEMPERATURE DISTRIBUTION.
$
TEMP,  1,   1,  0.
TEMP,  1,  11,  1000.
TEMPD,  1,  1000.
ENDDATA
```

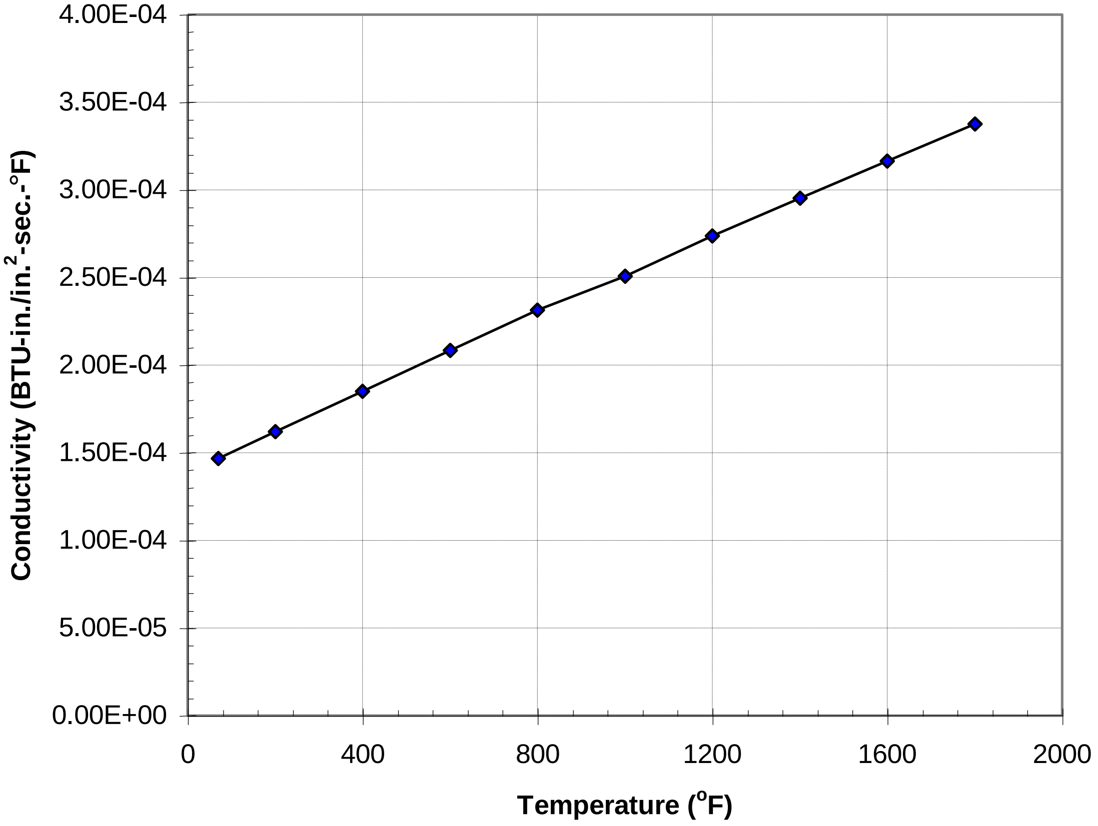
Heat is conducted through the bar according to the Fourier heat conduction equation:

$$f_x = -k\frac{\partial T}{\partial x}$$

where f_x is heat flux per unit area and k is the thermal conductivity. The negative sign means that heat flows in a direction opposite to the direction of temperature increase.

In this example, conductivity varies with temperature as shown in Figure 17-3. The temperature dependence is input using `MATT4` and `TABLEM2` Bulk Data entries. The `MATT4` Bulk Data entry must reference an isotropic material. For anisotropic materials, the `MATT5` and `MAT5` Bulk Data entries may be used.

Figure 17-3. `TABLEM2` **Bulk Data Entry Conductivity vs. Temperature Input Data.**

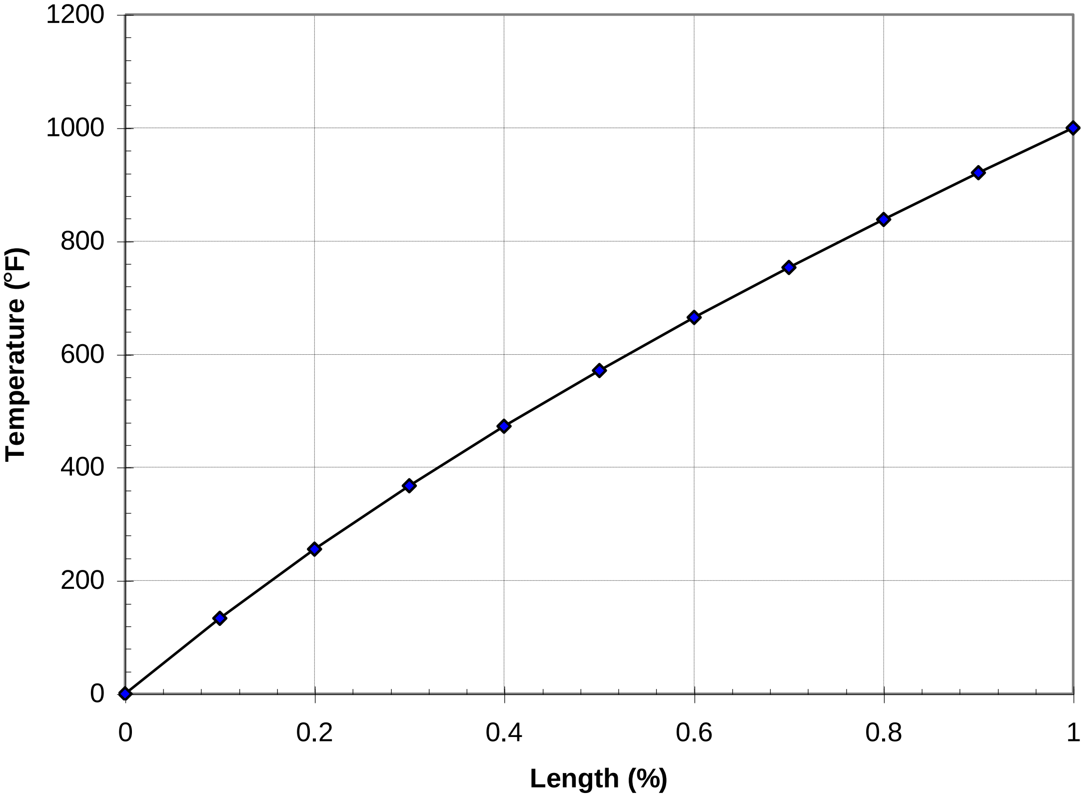


The grid point temperatures from the Model Results Output File are shown in tabular form in Listing 17-2 and plotted against percent of bar length in Figure 17-4. Rod element thermal gradients and heat fluxes are shown in tabular listing form in Listing 17-3. Thermal gradients are plotted against percent of bar length in Figure 17-5.

Listing 17-2. Temperature Vector for Bar Model with Constrained End Temperatures.

```
                                        T E M P E R A T U R E   V E C T O R

     GRID      TEMPERATURE
      ID
       2       1.342274E+02
       3       2.558578E+02
       4       3.680668E+02
       5       4.727612E+02
       6       5.712747E+02
       7       6.645875E+02
       8       7.534450E+02
       9       8.384298E+02
      10       9.205141E+02
      11       1.000000E+03
```

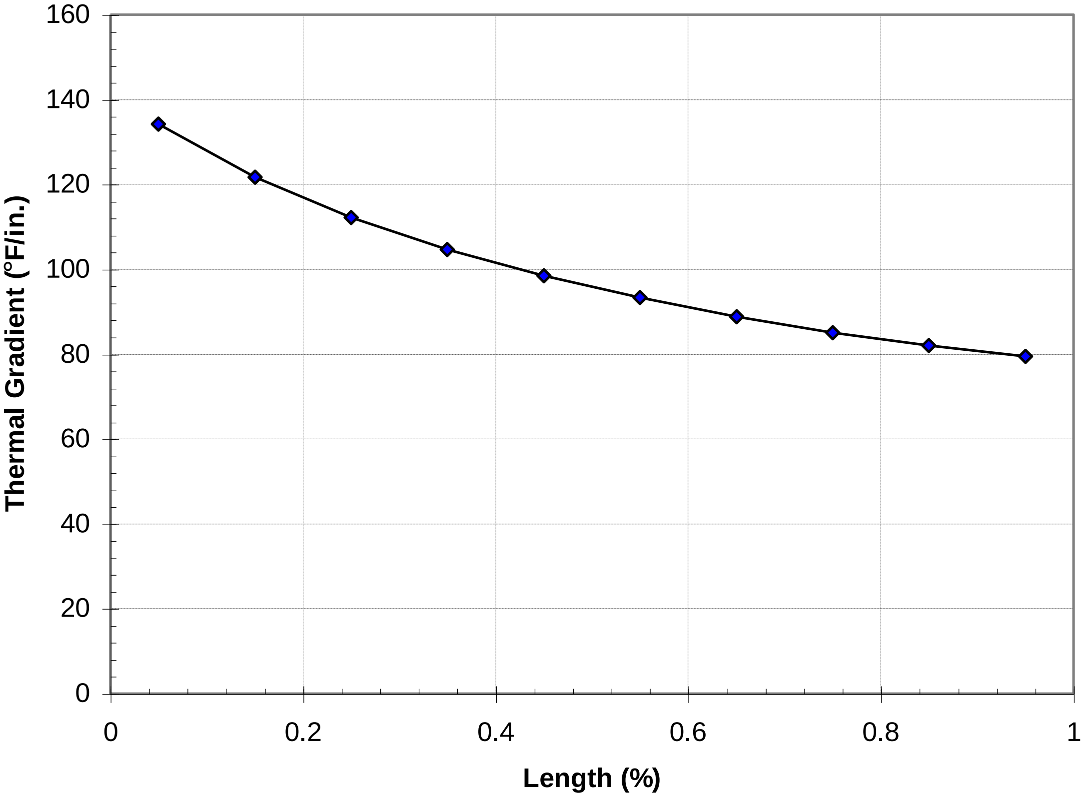
Figure 17-4. Temperature vs. Bar Model Normalized Length.



Listing 17-3. Element Thermal Gradients and Heat Fluxes for Bar Model with Constrained End Temperatures.

```
                 T H E R M A L   G R A D I E N T S   A N D   H E A T   F L U X E S   I N   R O D   E L E M E N T S

     ELEMENT        GRADIENT            FLUX
       ID
         1       1.342274E+02     -1.963187E-04
         2       1.216303E+02     -1.963187E-04
         3       1.122090E+02     -1.963207E-04
         4       1.046944E+02     -1.963230E-04
         5       9.851153E+01     -1.963249E-04
         6       9.331281E+01     -1.963257E-04
         7       8.885748E+01     -1.963254E-04
         8       8.498476E+01     -1.963241E-04
         9       8.208429E+01     -1.963216E-04
        10       7.948594E+01     -1.963201E-04
```

Figure 17-5. Thermal Gradient vs. Bar Model Normalized Length.



The next problem is another example of nonlinear conduction. The circular bar in Figure 17-6 has a prescribed temperature at one end, an applied heat flux at the other end, and is completely insulated over the rest of its surface area. Listing 17-4 contains the Model Input File.

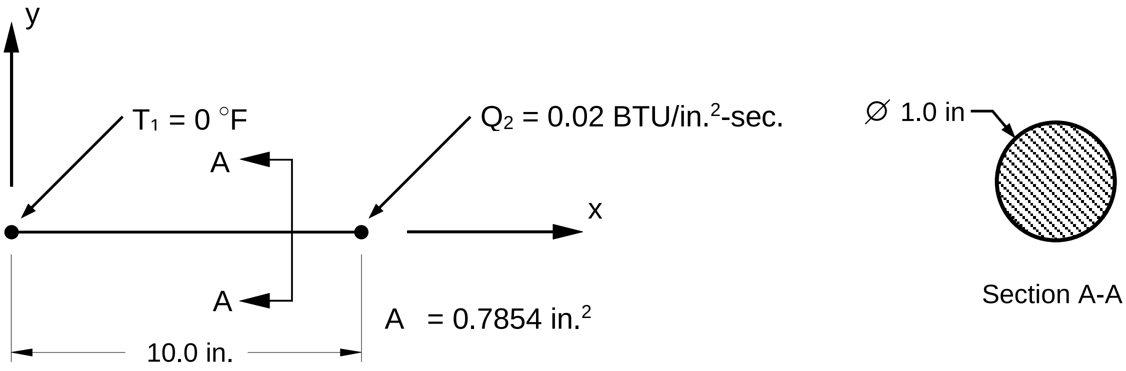


Figure 17-6. 1-Dimensional Bar Example Problem with Constrained End Temperature and Applied Heat Flux.

The grid point temperatures from the Model Results Output File are shown in tabular form in Listing 17-5 and plotted against percent of bar length in Figure 17-7. Rod element thermal gradients and heat fluxes are shown in tabular listing form in Listing 17-6. Thermal gradients are plotted against percent of bar length in Figure 17-8.

Listing 17-4. Model Input File for Bar Model with Constrained End Temperature and Applied Grid Point Heat Flux.

```
$
$ NONLINEAR STEADY STATE HEAT TRANSFER SOLUTION.
$
SOL NONLINEAR STEADY STATE HEAT TRANSFER
$
ANALYSIS = HEAT
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = BAR WITH NONLINEAR TEMPERATURE DEPENDENT MATERIAL
$
THERMAL = ALL
SPCFORCE = ALL
FLUX = ALL
$
TEMPERATURE(INITIAL) = 1
SUBCASE 1
 LABEL = CONSTRAINED END TEMPERATURE AND APPLIED END HEAT FLUX
 LOAD = 1
 NLPARM = 1
 SPC = 1
$
BEGIN BULK
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  1
$
$ GEOMETRY DEFINITION (10" BAR DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  ,   0.,  0.,  0.,
GRID,  2,  ,   1.,  0.,  0.,
GRID,  3,  ,   2.,  0.,  0.,
GRID,  4,  ,   3.,  0.,  0.,
GRID,  5,  ,   4.,  0.,  0.,
GRID,  6,  ,   5.,  0.,  0.,
GRID,  7,  ,   6.,  0.,  0.,
GRID,  8,  ,   7.,  0.,  0.,
GRID,  9,  ,   8.,  0.,  0.,
GRID, 10,  ,   9.,  0.,  0.,
GRID, 11,  ,  10.,  0.,  0.,
$
$ CIRCULAR BAR MODELED WITH ROD ELEMENTS.
$
CROD,  1,  10,   1,   2
CROD,  2,  10,   2,   3
CROD,  3,  10,   3,   4
CROD,  4,  10,   4,   5
CROD,  5,  10,   5,   6
CROD,  6,  10,   6,   7
CROD,  7,  10,   7,   8
CROD,  8,  10,   8,   9
CROD,  9,  10,   9,  10
CROD, 10,  10,  10,  11
$
$ ROD ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" DIAMETER).
$
PROD,  10,  100,  0.7854
```

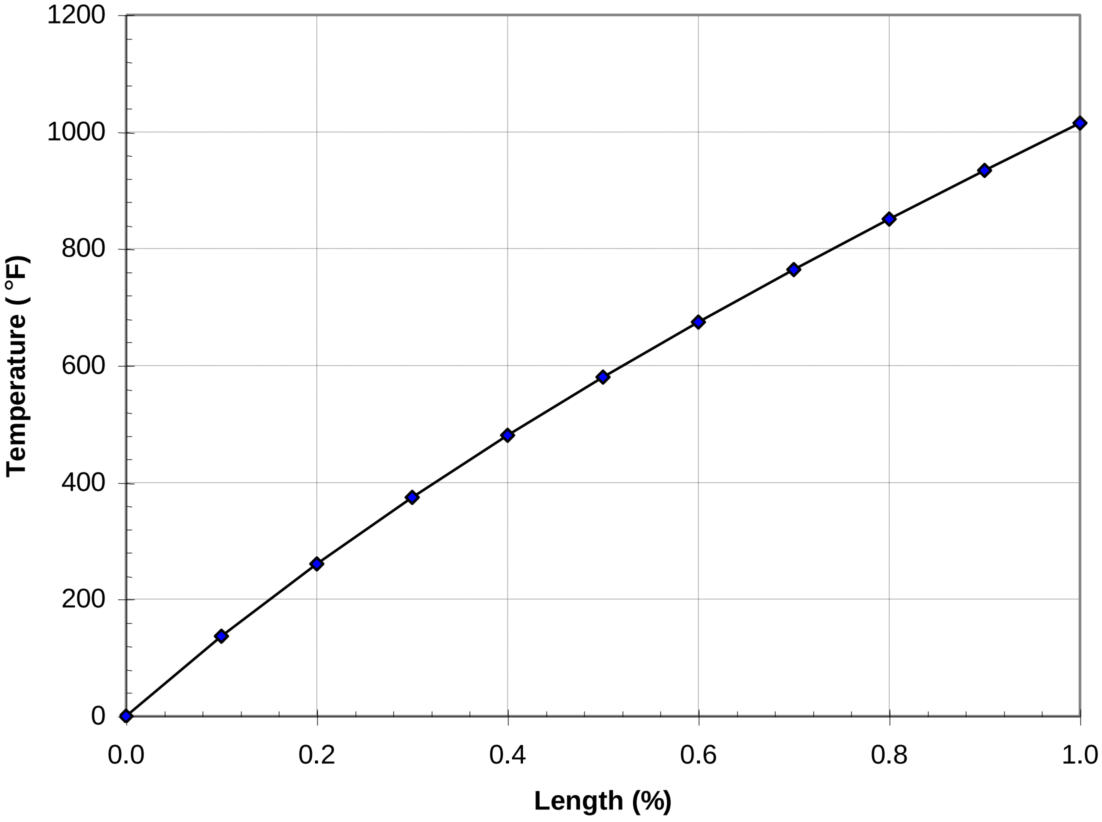
Listing 17-4. Model Input File for Bar Model with Constrained End Temperature and Applied Grid Point Heat Flux. (Continued)

```
$
$ ELEMENT CONDUCTIVITY (MA956).
$
MAT4,  100,  1.
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
MATT4,  100,  10
$
$ TEMPERATURE DEPENDENT CONDUCTIVITY DATA.
$
TABLEM2,  10,
,    70.,  1.466E-4,   200.,  1.620E-4,   400.,  1.852E-4,   600.,  2.083E-4,
,   800.,  2.315E-4,  1000.,  2.508E-4,  1200.,  2.739E-4,  1400.,  2.951E-4,
,  1600.,  3.164E-4,  1800.,  3.376E-4,  ENDT
$
$ GRID POINT HEAT FLUX.
$
QHBDY,  1,  POINT,  0.02,  0.7854,  11
$
$ CONSTRAIN BAR END TEMPERATURE.
$
SPC,  1,  1,  1,  0.
$
$ INITIAL TEMPERATURE DISTRIBUTION.
$
TEMP,  1,  1,  0.
TEMPD,  1,  0.
ENDDATA
```

Listing 17-5. Temperature Vector for Bar Model with Constrained End Temperature and Applied Grid Point Heat Flux.

```
                                          T E M P E R A T U R E   V E C T O R

     GRID      TEMPERATURE
      ID
        2      1.366126E+02
        3      2.602129E+02
        4      3.741140E+02
        5      4.803034E+02
        6      5.801821E+02
        7      6.747510E+02
        8      7.647634E+02
        9      8.508763E+02
       10      9.340643E+02
       11      1.014610E+03
```

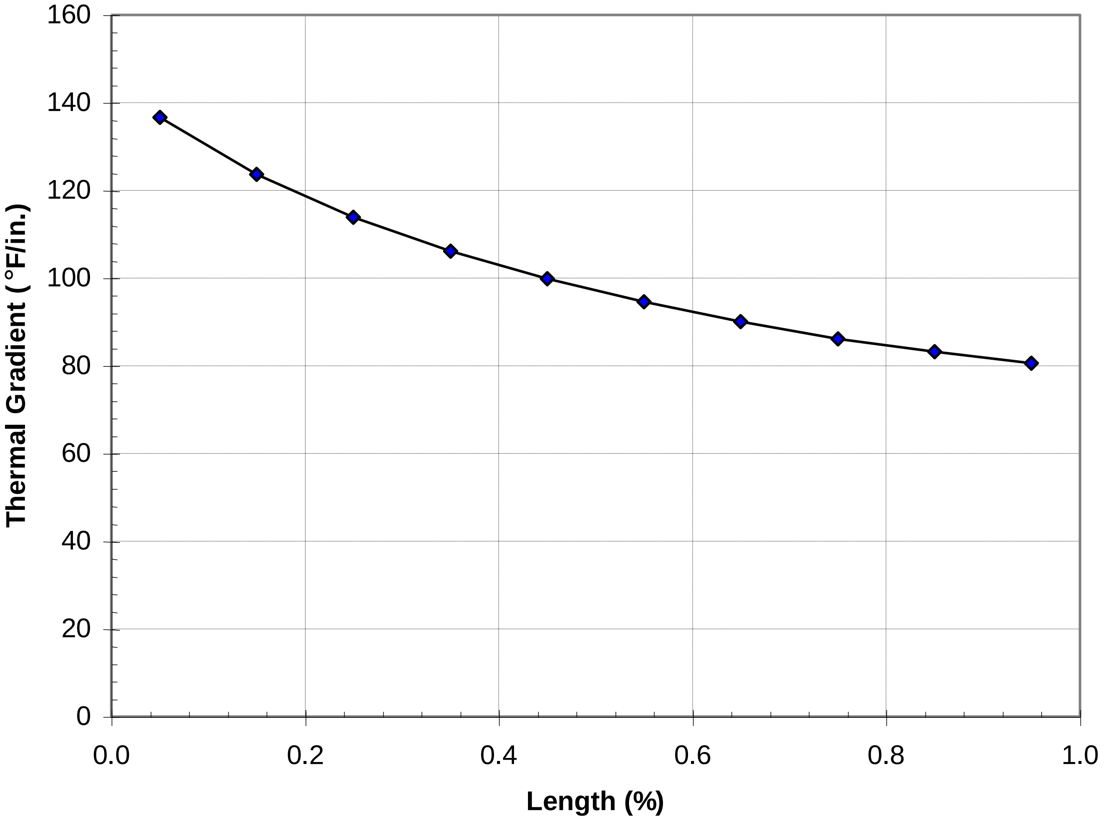
Figure 17-7. Temperature vs. Bar Model Normalized Length.



Listing 17-6. Element Thermal Gradients and Heat Fluxes for Bar Model with Constrained End Temperature and Applied Grid Point Heat Flux.

```
                 T H E R M A L   G R A D I E N T S   A N D   H E A T   F L U X E S   I N   R O D   E L E M E N T S

     ELEMENT         GRADIENT            FLUX
       ID
        1        1.366126E+02     -2.000000E-02
        2        1.236003E+02     -2.000000E-02
        3        1.139011E+02     -2.000000E-02
        4        1.061895E+02     -2.000000E-02
        5        9.987864E+01     -2.000000E-02
        6        9.456890E+01     -2.000001E-02
        7        9.001246E+01     -2.000003E-02
        8        8.611284E+01     -2.000010E-02
        9        8.318802E+01     -2.000034E-02
       10        8.054531E+01     -2.000129E-02
```

Figure 17-8. Thermal Gradient vs. Bar Model Normalized Length.



Heat flux loads can be either applied directly into a set of grid points or onto a `CHBDYG` or `CHBDYP` surface element. Listing 17-4 was an example of heat flux loads applied directly into a set of grid points using the `QHBDY` Bulk Data entry. Listing 17-7 is an example of heat flux loads applied onto a `CHBDYP` element.

The grid point temperatures from the Model Results Output File are shown in tabular form in Listing 17-8 and plotted against percent of bar length in Figure 17-9. Heat flow into the hbdy element is shown in Listing 17-9. Both examples yield equivalent results.

Listing 17-7. Model Input File for Bar Model with Constrained End Temperature and Applied Element Heat Flux.

```
$
$ NONLINEAR STEADY STATE HEAT TRANSFER SOLUTION.
$
SOL NONLINEAR STEADY STATE HEAT TRANSFER
$
ANALYSIS = HEAT
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = BAR WITH NONLINEAR TEMPERATURE DEPENDENT MATERIAL
$
THERMAL = ALL
SPCFORCE = ALL
FLUX = ALL
$
TEMPERATURE(INITIAL) = 1
SUBCASE 1
 LABEL = CONSTRAINED END TEMPERATURE AND APPLIED END HEAT FLUX
 LOAD = 1
 NLPARM = 1
 SPC = 1
$
BEGIN BULK
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  1
$
$ GEOMETRY DEFINITION (10" BAR DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  ,   0.,  0.,  0.,
GRID,  2,  ,   1.,  0.,  0.,
GRID,  3,  ,   2.,  0.,  0.,
GRID,  4,  ,   3.,  0.,  0.,
GRID,  5,  ,   4.,  0.,  0.,
GRID,  6,  ,   5.,  0.,  0.,
GRID,  7,  ,   6.,  0.,  0.,
GRID,  8,  ,   7.,  0.,  0.,
GRID,  9,  ,   8.,  0.,  0.,
GRID, 10,  ,   9.,  0.,  0.,
GRID, 11,  ,  10.,  0.,  0.,
$
$ CIRCULAR BAR MODELED WITH ROD ELEMENTS.
$
CROD,  1,  10,   1,   2
CROD,  2,  10,   2,   3
CROD,  3,  10,   3,   4
CROD,  4,  10,   4,   5
CROD,  5,  10,   5,   6
CROD,  6,  10,   6,   7
CROD,  7,  10,   7,   8
CROD,  8,  10,   8,   9
CROD,  9,  10,   9,  10
CROD, 10,  10,  10,  11
$
$ ROD ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" DIAMETER).
$
PROD,  10,  100,  0.7854
```

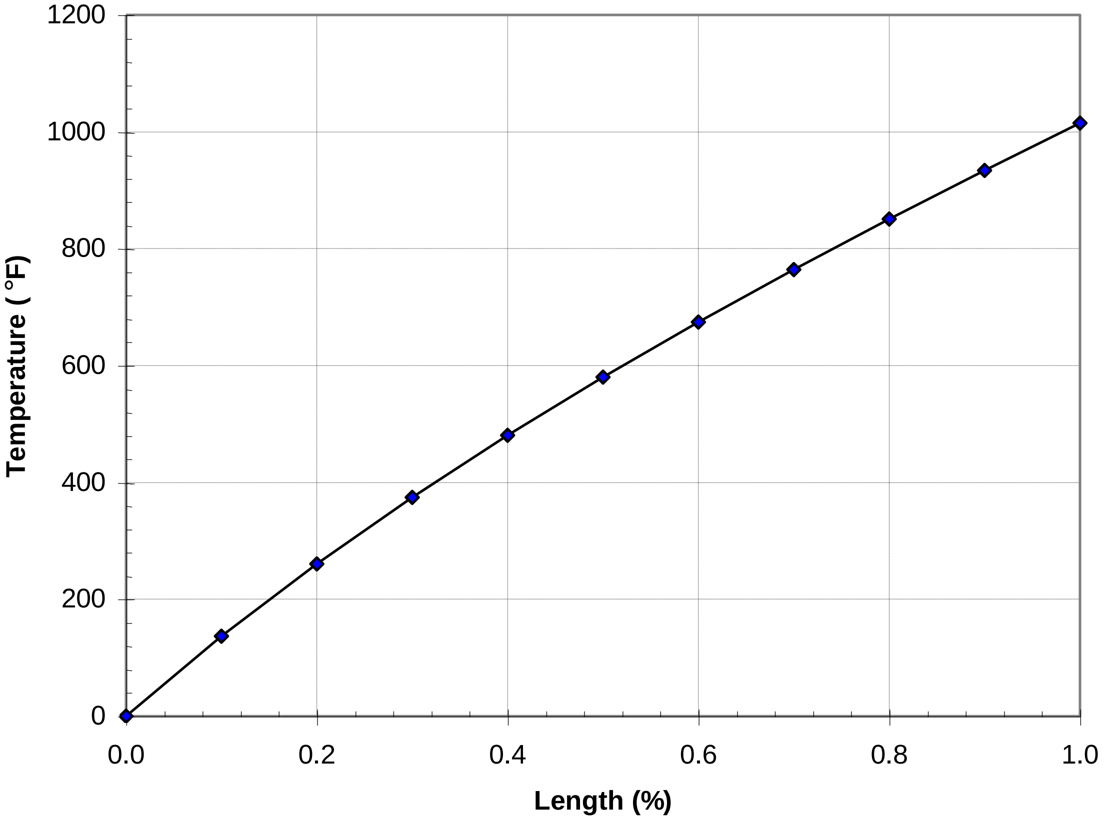
Listing 17-7. Model Input File for Bar Model with Constrained End Temperature and Applied Element Heat Flux. (Continued)

```
$
$ ELEMENT CONDUCTIVITY (MA956).
$
MAT4,  100,  1.
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
MATT4,  100,  10
$
$ TEMPERATURE DEPENDENT CONDUCTIVITY DATA.
$
TABLEM2,  10,
,    70.,  1.466E-4,   200.,  1.620E-4,   400.,  1.852E-4,   600.,  2.083E-4,
,   800.,  2.315E-4,  1000.,  2.508E-4,  1200.,  2.739E-4,  1400.,  2.951E-4,
,  1600.,  3.164E-4,  1800.,  3.376E-4,  ENDT
$
$ DEFINE BOUNDARY HEAT FLUX.
$
CHBDYP,  11,  20,  POINT,  ,  ,  11,
  ,  ,  ,  ,  ,  1.,  0.,  0.
$
$ AREA FACTOR TO DEFINE ROD END SURFACE AREA
$ AREA = (PI/4)*DIAMETER**2 = 0.7854
$
PHBDY,  20,  0.7854
$
$ HEAT FLUX.
$
QBDY1,  1,  0.02,  11
$
$ CONSTRAIN BAR END TEMPERATURE.
$
SPC,  1,  1,  1,  0.
$
$ INITIAL TEMPERATURE DISTRIBUTION.
$
TEMP,  1,  1,  0.
TEMPD,  1,  0.
ENDDATA
```

Listing 17-8. Temperature Vector for Bar Model with Constrained End Temperature and Applied Element Heat Flux.

```
                                   T E M P E R A T U R E   V E C T O R

     GRID      TEMPERATURE
      ID
        2      1.366126E+02
        3      2.602129E+02
        4      3.741140E+02
        5      4.803034E+02
        6      5.801821E+02
        7      6.747510E+02
        8      7.647634E+02
        9      8.508763E+02
       10      9.340643E+02
       11      1.014610E+03
```

Figure 17-9. Temperature vs. Bar Model Normalized Length.



Listing 17-9. Heat Flow Into Hbdy Element for Bar Model with Constrained End Temperature and Applied Element Heat Flux.

```
                                   H E A T   F L O W   I N T O   H B D Y   E L E M E N T S

      ELEMENT        APPLIED        CONVECTION        TOTAL
        ID
         11       1.570800E-02   0.000000E+00   1.570800E-02
```

17.4.2 Volume Heat Addition

The next problem is example of volume heat addition. The circular bar in Figure 17-10 has a prescribed temperature at one end, temperature dependent volumetric heat addition in each element, and is completely insulated over the rest of its surface area. Listing 17-10 contains the Model Input File.

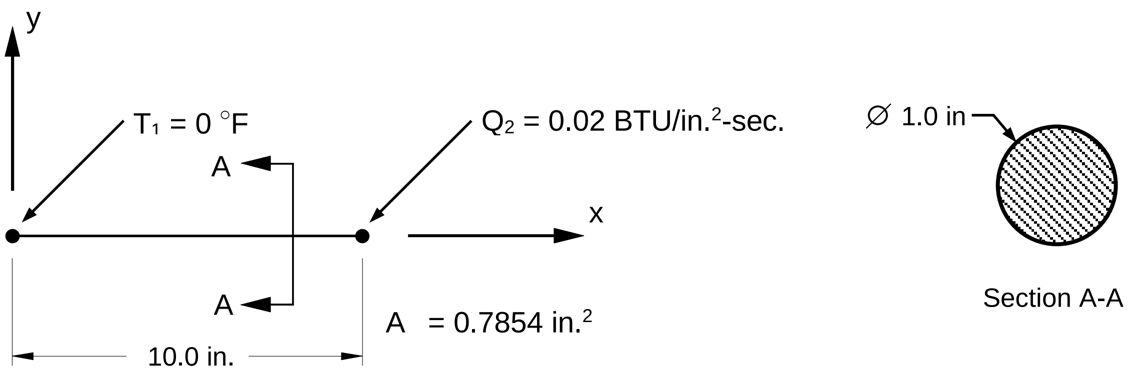


Figure 17-10. 1-Dimensional Bar Example Problem with Constrained End Temperature and Volumetric Heat Addition.

Volumetric heat addition results in elemental power input given by the equation:

$$P_{in} = Volume * \text{HGEN} * \text{QVOL}$$

Where QVOL is the power density given on the `QVOL` Bulk Data entry and HGEN is a temperature dependent scale factor given on the `MAT4` or `MAT5` Bulk Data entry.

The grid point temperatures from the Model Results Output File are shown in tabular form in Listing 17-11 and plotted against percent of bar length in Figure 17-11. Rod element thermal gradients and heat fluxes are shown in tabular listing form in Listing 17-12. Thermal gradients are plotted against percent of bar length in Figure 17-12.

Listing 17-10. Model Input File for Bar Model with Constrained End Temperature and Volumetric Heat Addition.

```
$
$ NONLINEAR STEADY STATE HEAT TRANSFER SOLUTION.
$
SOL NONLINEAR STEADY STATE HEAT TRANSFER
$
ANALYSIS = HEAT
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = BAR WITH NONLINEAR TEMPERATURE DEPENDENT MATERIAL
$
THERMAL = ALL
SPCFORCE = ALL
FLUX = ALL
$
TEMPERATURE(INITIAL) = 1
SUBCASE 1
 LABEL = CONSTRAINED END TEMPERATURE AND VOLUME HEAT ADDITION
 LOAD = 1
 NLPARM = 1
 SPC = 1
$
BEGIN BULK
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  1
$
$ GEOMETRY DEFINITION (10" BAR DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  ,   0.,  0.,  0.,
GRID,  2,  ,   1.,  0.,  0.,
GRID,  3,  ,   2.,  0.,  0.,
GRID,  4,  ,   3.,  0.,  0.,
GRID,  5,  ,   4.,  0.,  0.,
GRID,  6,  ,   5.,  0.,  0.,
GRID,  7,  ,   6.,  0.,  0.,
GRID,  8,  ,   7.,  0.,  0.,
GRID,  9,  ,   8.,  0.,  0.,
GRID, 10,  ,   9.,  0.,  0.,
GRID, 11,  ,  10.,  0.,  0.,
$
$ CIRCULAR BAR MODELED WITH ROD ELEMENTS.
$
CROD,  1,  10,   1,   2
CROD,  2,  10,   2,   3
CROD,  3,  10,   3,   4
CROD,  4,  10,   4,   5
CROD,  5,  10,   5,   6
CROD,  6,  10,   6,   7
CROD,  7,  10,   7,   8
CROD,  8,  10,   8,   9
CROD,  9,  10,   9,  10
CROD, 10,  10,  10,  11
$
$ ROD ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" DIAMETER).
$
PROD,  10,  100,  0.7854
```

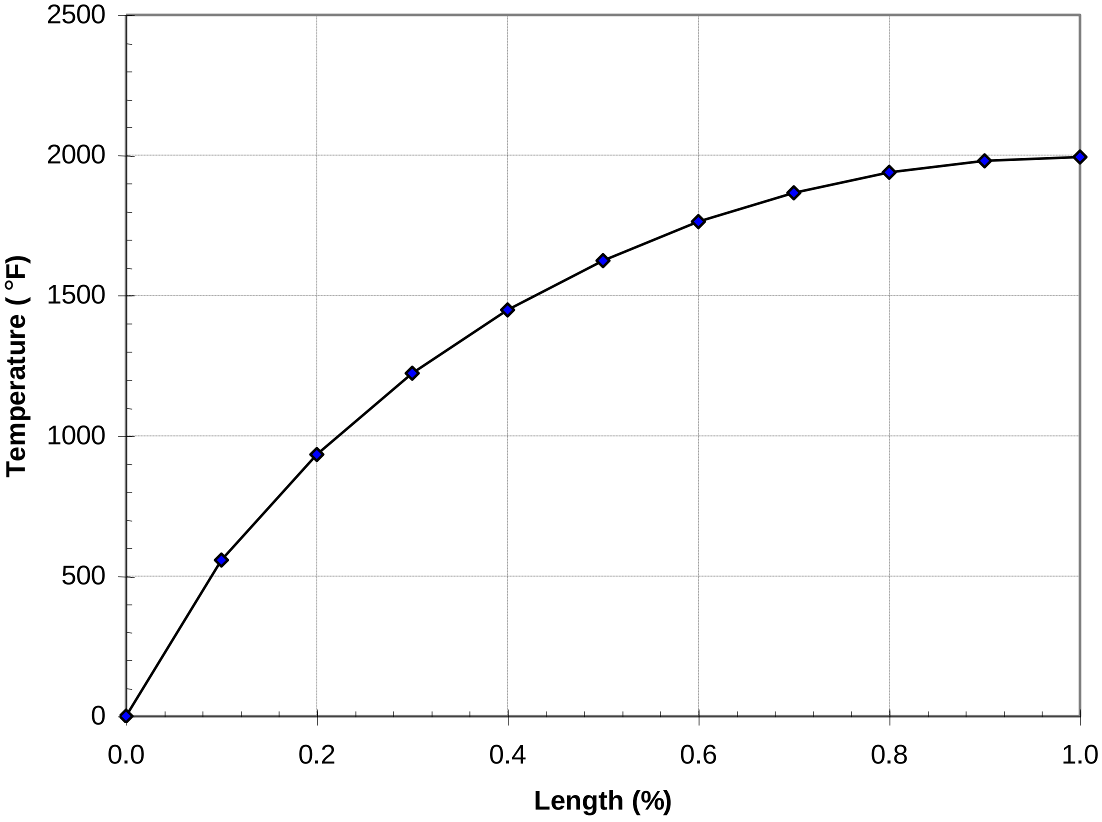
Listing 17-10. Model Input File for Bar Model with Constrained End Temperature and Volumetric Heat Addition. (Continued)

```
$
$ ELEMENT CONDUCTIVITY (MA956).
$
MAT4,  100,  1.
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
MATT4,  100,  10
$
$ TEMPERATURE DEPENDENT CONDUCTIVITY DATA.
$
TABLEM2,  10,
,    70.,  1.466E-4,   200.,  1.620E-4,   400.,  1.852E-4,   600.,  2.083E-4,
,   800.,  2.315E-4,  1000.,  2.508E-4,  1200.,  2.739E-4,  1400.,  2.951E-4,
,  1600.,  3.164E-4,  1800.,  3.376E-4,  ENDT
$
$ VOLUME HEAT ADDITION.
$
QVOL,  1,  0.01,  ,  1,  THRU,  10
$
$ ELEMENT POWER DENSITY.
$
MAT4,  200,  ,  ,  ,  ,  ,  1.
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
MATT4,  200,  ,  ,  ,  ,  ,  20
$
$ TEMPERATURE DEPENDENT HEAT GENERATION COEFFICIENT.
$
TABLEM2,  20,
,    70.,  0.993,   200.,  0.980,   400.,  0.960,   600.,  0.940,
,   800.,  0.920,  1000.,  0.900,  1200.,  0.880,  1400.,  0.860,
,  1600.,  0.840,  1800.,  0.820,  ENDT
$
$ CONSTRAIN BAR END TEMPERATURE.
$
SPC,  1,  1,  1,  0.
$
$ INITIAL TEMPERATURE DISTRIBUTION.
$
TEMP,  1,  1,  0.
TEMPD,  1,  0.
ENDDATA
```

Listing 17-11. Temperature Vector for Bar Model with Constrained End Temperature and Volumetric Heat Addition.

```
                                       T E M P E R A T U R E   V E C T O R

      GRID       TEMPERATURE
       ID
        2       5.555005E+02
        3       9.332160E+02
        4       1.221964E+03
        5       1.447538E+03
        6       1.625198E+03
        7       1.763098E+03
        8       1.866315E+03
        9       1.938081E+03
       10       1.980407E+03
       11       1.994398E+03
```

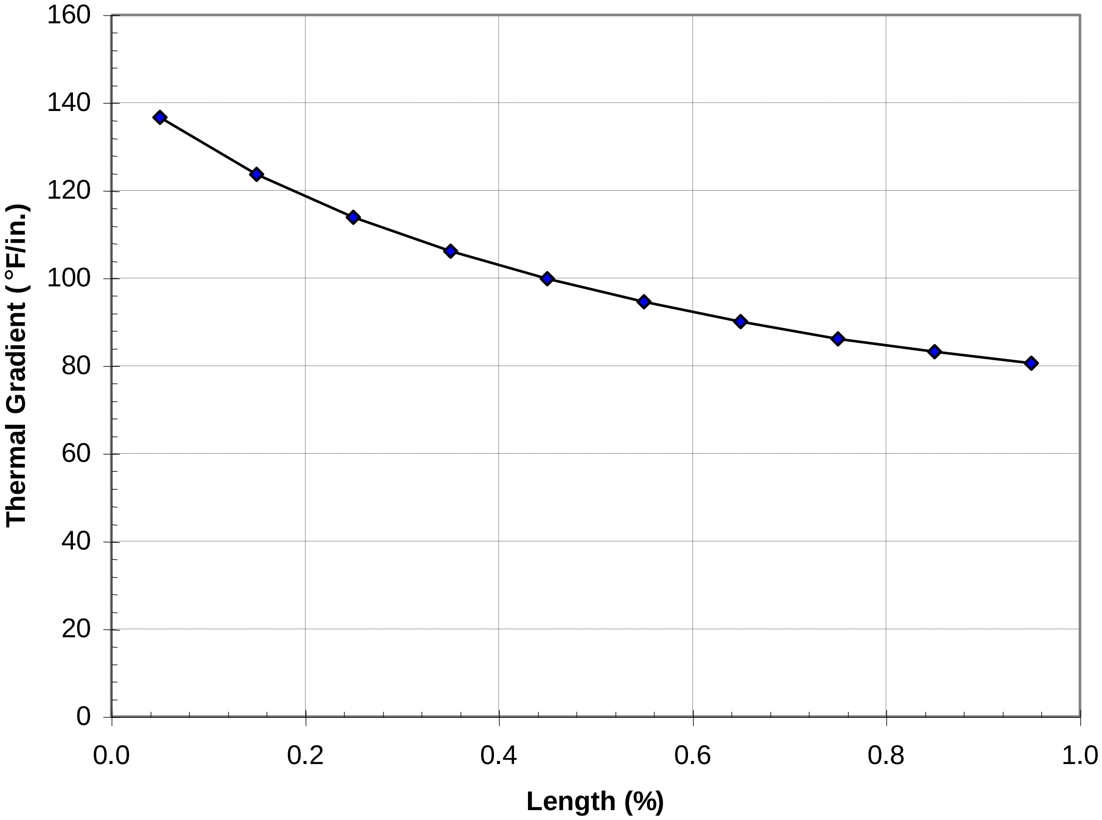
Figure 17-11. Temperature vs. Bar Model Normalized Length.



Listing 17-12. Element Thermal Gradients and Heat Fluxes for Bar Model with Constrained End Temperature and Volumetric Heat Addition.

```
                  T H E R M A L   G R A D I E N T S   A N D   H E A T   F L U X E S   I N   R O D   E L E M E N T S

      ELEMENT         GRADIENT            FLUX
        ID
          1        5.555005E+02     -9.500115E-02
          2        3.777156E+02     -8.500321E-02
          3        2.887477E+02     -7.500556E-02
          4        2.255742E+02     -6.500677E-02
          5        1.776602E+02     -5.500771E-02
          6        1.379001E+02     -4.500778E-02
          7        1.032173E+02     -3.500706E-02
          8        7.176593E+01     -2.500562E-02
          9        4.232571E+01     -1.500361E-02
         10        1.399088E+01     -5.001245E-03
```

Figure 17-12. Thermal Gradient vs. Bar Model Normalized Length.



17.4.3 Nonlinear Convection

The next problem is an example of nonlinear convection. The circular bar in Figure 17-13 has prescribed temperature at one end and a convection boundary condition over the rest of its surface area. Listing 17-11 contains the Model Input File.

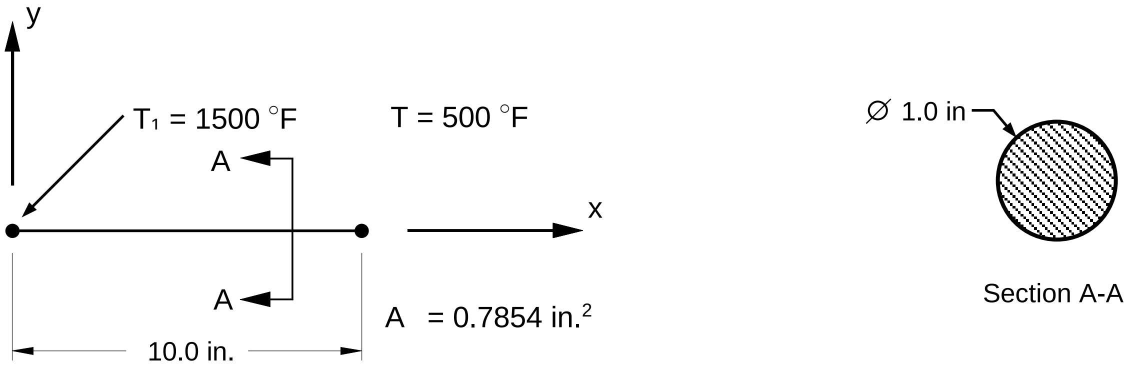


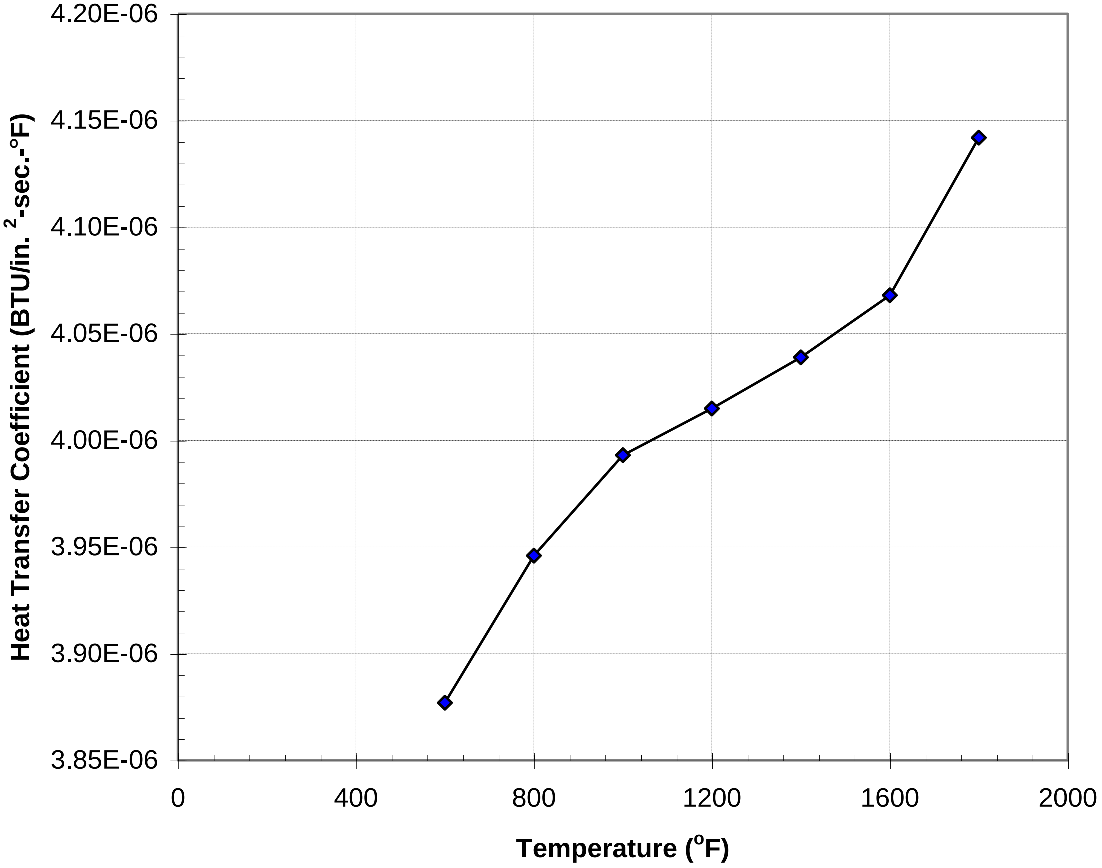
Figure 17-13. 1-Dimensional Bar Example Problem with Constrained End Temperature and a Convection Boundary Condition.

The `CONV` Bulk Data entry defines the convection properties and the ambient grid points. The basic exchange relationship defines the rate of heat transfer as:

$$q = H*(T - TAMB)$$

where H is the temperature dependent free convection heat transfer coefficient given on the `MAT4` Bulk Data entry and T and TAMB are the fluid film and ambient temperatures, respectively. H is given in Reference 7 for various shapes and conditions. For our example, H is plotted as a function of temperature in Figure 17-14.

Figure 17-14. `TABLEM2` **Bulk Data Entry Free Convection Heat Transfer Coefficient vs. Temperature Input Data.**



The grid point temperatures from the Model Results Output File are shown in tabular form in Listing 17-14 and plotted against percent of bar length in Figure 17-15. Rod element thermal gradients and heat fluxes are shown in tabular listing form in Listing 17-15. Thermal gradients are plotted against percent of bar length in Figure 17-16. Heat flows into the hbdy elements are shown in tabular form in Listing 17-16 and plotted against percent of bar length in Figure 17-17.

Listing 17-13. Model Input File for Bar Model with Constrained End Temperature and a Convection Boundary Condition.

```
$
$ NONLINEAR STEADY STATE HEAT TRANSFER SOLUTION.
$
SOL NONLINEAR STEADY STATE HEAT TRANSFER
$
ANALYSIS = HEAT
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = BAR WITH NONLINEAR TEMPERATURE DEPENDENT MATERIAL
$
THERMAL = ALL
SPCFORCE = ALL
FLUX = ALL
$
TEMPERATURE(INITIAL) = 1
SUBCASE 1
 LABEL = CONSTRAINED END TEMPERATURE AND FREE CONVECTION BOUNDARY
 SPC = 1
 NLPARM = 1
$
BEGIN BULK
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  1
$
$ GEOMETRY DEFINITION (10" BAR DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  ,   0.,  0.,  0.,
GRID,  2,  ,   1.,  0.,  0.,
GRID,  3,  ,   2.,  0.,  0.,
GRID,  4,  ,   3.,  0.,  0.,
GRID,  5,  ,   4.,  0.,  0.,
GRID,  6,  ,   5.,  0.,  0.,
GRID,  7,  ,   6.,  0.,  0.,
GRID,  8,  ,   7.,  0.,  0.,
GRID,  9,  ,   8.,  0.,  0.,
GRID, 10,  ,   9.,  0.,  0.,
GRID, 11,  ,  10.,  0.,  0.,
GRID, 12,  ,   0.,  0.,  0.,
$
$ CIRCULAR BAR MODELED WITH ROD ELEMENTS.
$
CROD,  1,  10,   1,   2
CROD,  2,  10,   2,   3
CROD,  3,  10,   3,   4
CROD,  4,  10,   4,   5
CROD,  5,  10,   5,   6
CROD,  6,  10,   6,   7
CROD,  7,  10,   7,   8
CROD,  8,  10,   8,   9
CROD,  9,  10,   9,  10
CROD, 10,  10,  10,  11
$
$ ROD ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" DIAMETER).
$
PROD,  10,  100,  0.7854
$
$ ELEMENT CONDUCTIVITY (MA956).
$
MAT4,  100,  1.
```

Listing 17-13. Model Input File for Bar Model with Constrained End Temperature and a Convection Boundary Condition. (Continued)

```
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
MATT4,  100,  10
$
$ TEMPERATURE DEPENDENT CONDUCTIVITY DATA.
$
TABLEM2,  10,
,    70.,  1.466E-4,   200.,  1.620E-4,   400.,  1.852E-4,   600.,  2.083E-4,
,   800.,  2.315E-4,  1000.,  2.508E-4,  1200.,  2.739E-4,  1400.,  2.951E-4,
,  1600.,  3.164E-4,  1800.,  3.376E-4,  ENDT
$
$ CONVECTION BOUNDARY DEFINITION.
$
CHBDYP,  11,  20,  LINE,  ,  ,   1,   2,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  12,  20,  LINE,  ,  ,   2,   3,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  13,  20,  LINE,  ,  ,   3,   4,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  14,  20,  LINE,  ,  ,   4,   5,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  15,  20,  LINE,  ,  ,   5,   6,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  16,  20,  LINE,  ,  ,   6,   7,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  17,  20,  LINE,  ,  ,   7,   8,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  18,  20,  LINE,  ,  ,   8,   9,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  19,  20,  LINE,  ,  ,   9,  10,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  20,  20,  LINE,  ,  ,  10,  11,
  ,  ,  ,  ,  ,  0.,  1.,  0.
$
$ AREA FACTOR TO DEFINE ROD SURFACE AREA
$ AF = PI*DIAMETER = 3.142, AREA = AF*LENGTH
$
PHBDY,  20,  3.142
$
$ CONVECTION LOAD.
$
CONV,  11,  30,  ,  ,  12,  12
CONV,  12,  30,  ,  ,  12,  12
CONV,  13,  30,  ,  ,  12,  12
CONV,  14,  30,  ,  ,  12,  12
CONV,  15,  30,  ,  ,  12,  12
CONV,  16,  30,  ,  ,  12,  12
CONV,  17,  30,  ,  ,  12,  12
CONV,  18,  30,  ,  ,  12,  12
CONV,  19,  30,  ,  ,  12,  12
CONV,  20,  30,  ,  ,  12,  12
$
$ CONVECTION PROPERTY REFERENCE.
$
PCONV,  30,  200
$
$ FREE CONVECTION HEAT TRANSFER COEFFICIENT (AIR).
$
MAT4,  200,  ,  ,  ,  1.
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
MATT4,  200,  ,  ,  ,  20
```

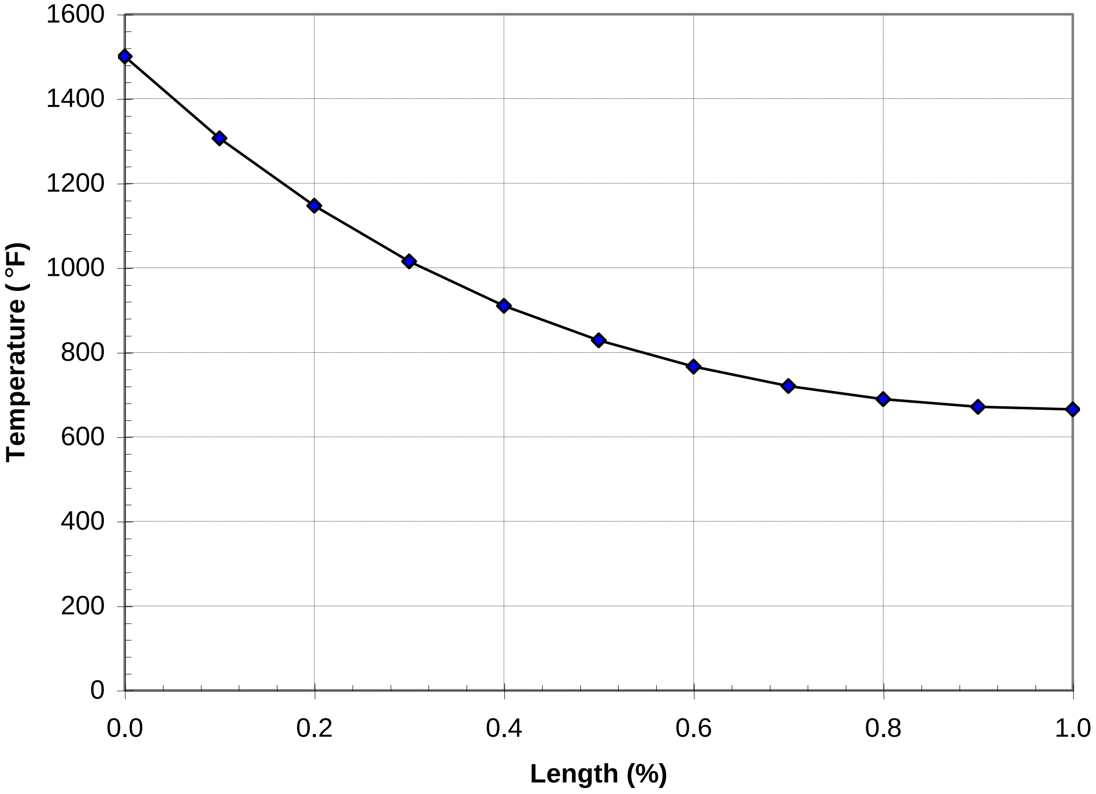
Listing 17-13. Model Input File for Bar Model with Constrained End Temperature and a Convection Boundary Condition. (Continued)

```
$
$ TEMPERATURE DEPENDENT FREE CONVECTION HEAT TRANSFER COEFFICIENT.
$
TABLEM2,  20,
,   600.,  3.877E-6,   800.,  3.946E-6,  1000.,  3.993E-6,  1200.,  4.015E-6,
,  1400.,  4.039E-6,  1600.,  4.068E-6,  1800.,  4.142E-6,  2000.,  4.261E-6,
,  2200.,  4.380E-6,  2400.,  4.501E-6,  ENDT
$
$ AMBIENT TEMPERATURE DEFINITION.
$
SPC,  1,  12,  1,  500.
$
$ CONSTRAIN BAR END TEMPERATURE.
$
SPC,  1,  1,  1,  1500.
$
$ INITIAL TEMPERATURE DISTRIBUTION.
$
TEMP,  1,  12,  500.
TEMPD,  1,  1500.
ENDDATA
```

Listing 17-14. Temperature Vector for Bar Model with Constrained End Temperature and a Convection Boundary Condition.

```
                                      T E M P E R A T U R E   V E C T O R

     GRID       TEMPERATURE
      ID
        1       1.500000E+03
        2       1.324682E+03
        3       1.161522E+03
        4       1.027907E+03
        5       9.205718E+02
        6       8.366836E+02
        7       7.728891E+02
        8       7.261412E+02
        9       6.943046E+02
       10       6.758315E+02
       11       6.697793E+02
       12       5.000000E+02
```

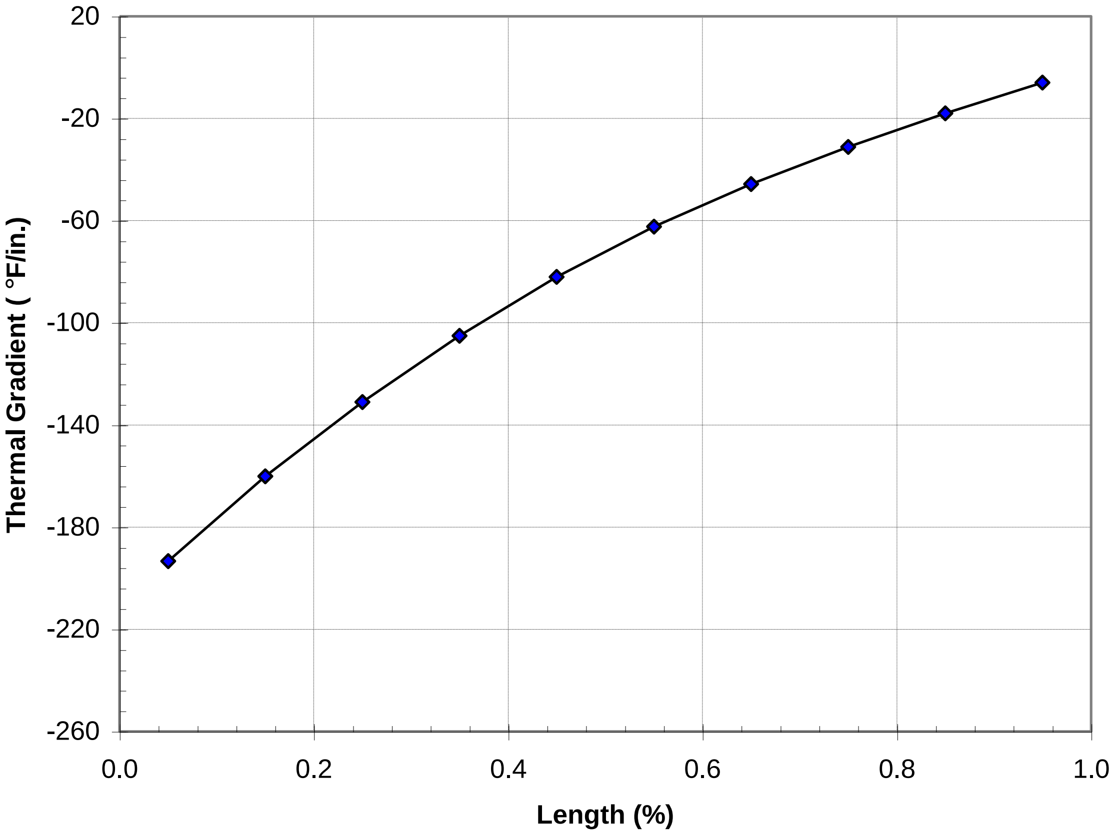
Figure 17-15. Temperature vs. Bar Model Normalized Length.



Listing 17-15. Element Thermal Gradients and Heat Fluxes for Bar Model with Constrained End Temperature and a Convection Boundary Condition.

```
               T H E R M A L   G R A D I E N T S   A N D   H E A T   F L U X E S   I N   R O D   E L E M E N T S

  ELEMENT        GRADIENT            FLUX
    ID
      1       -1.933375E+02      5.712249E-02
      2       -1.601330E+02      4.431186E-02
      3       -1.310460E+02      3.409244E-02
      4       -1.050667E+02      2.597509E-02
      5       -8.204297E+01      1.954236E-02
      6       -6.235922E+01      1.441586E-02
      7       -4.568761E+01      1.027550E-02
      8       -3.109997E+01      6.856118E-03
      9       -1.803999E+01      3.925574E-03
     10       -5.909447E+00      1.277711E-03
```

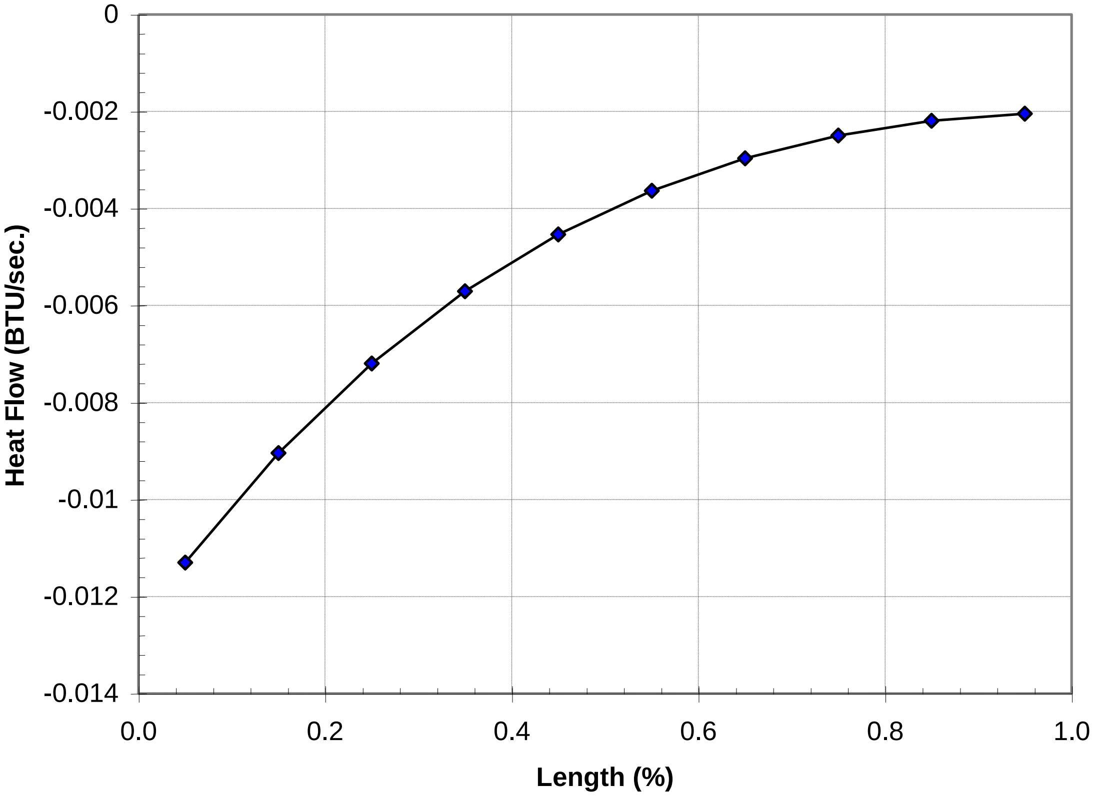
Figure 17-16. Thermal Gradient vs. Bar Model Normalized Length.



Listing 17-16. Heat Flow Into Hbdy Elements for Bar Model with Constrained End Temperature and a Convection Boundary Condition.

```
                                          H E A T   F L O W   I N T O   H B D Y   E L E M E N T S

     ELEMENT       APPLIED        CONVECTION     RADIATION        TOTAL
       ID
        11       0.000000E+00  -1.130096E-02   0.000000E+00  -1.130096E-02
        12       0.000000E+00  -9.042538E-03   0.000000E+00  -9.042538E-03
        13       0.000000E+00  -7.197530E-03   0.000000E+00  -7.197530E-03
        14       0.000000E+00  -5.705421E-03   0.000000E+00  -5.705421E-03
        15       0.000000E+00  -4.533716E-03   0.000000E+00  -4.533716E-03
        16       0.000000E+00  -3.635937E-03   0.000000E+00  -3.635937E-03
        17       0.000000E+00  -2.967883E-03   0.000000E+00  -2.967883E-03
        18       0.000000E+00  -2.495028E-03   0.000000E+00  -2.495028E-03
        19       0.000000E+00  -2.193265E-03   0.000000E+00  -2.193265E-03
        20       0.000000E+00  -2.046431E-03   0.000000E+00  -2.046431E-03
```

Figure 17-17. Surface Heat Flow vs. Bar Model Normalized Length.



17.4.4 Nonlinear Radiation

The next problem is an example of nonlinear radiation. The circular bar in Figure 17-18 has prescribed temperature at one end and a radiation boundary condition over the rest of its surface area. Listing 17-17 contains the Model Input File.

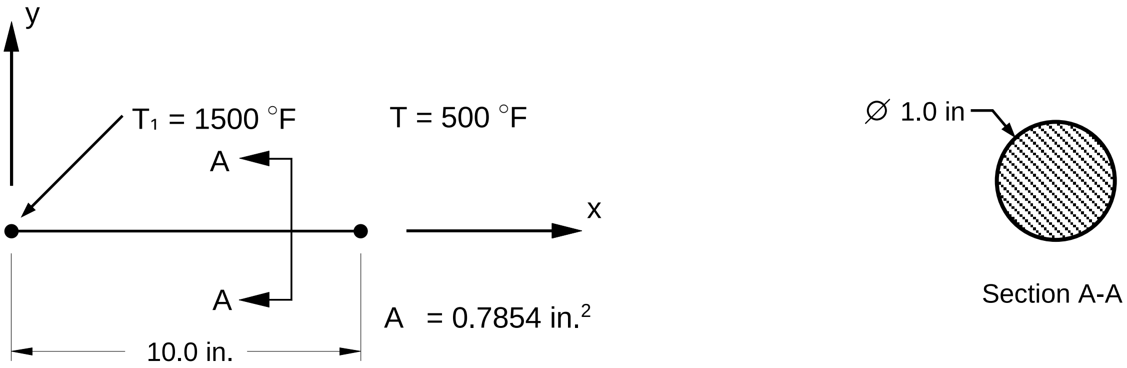


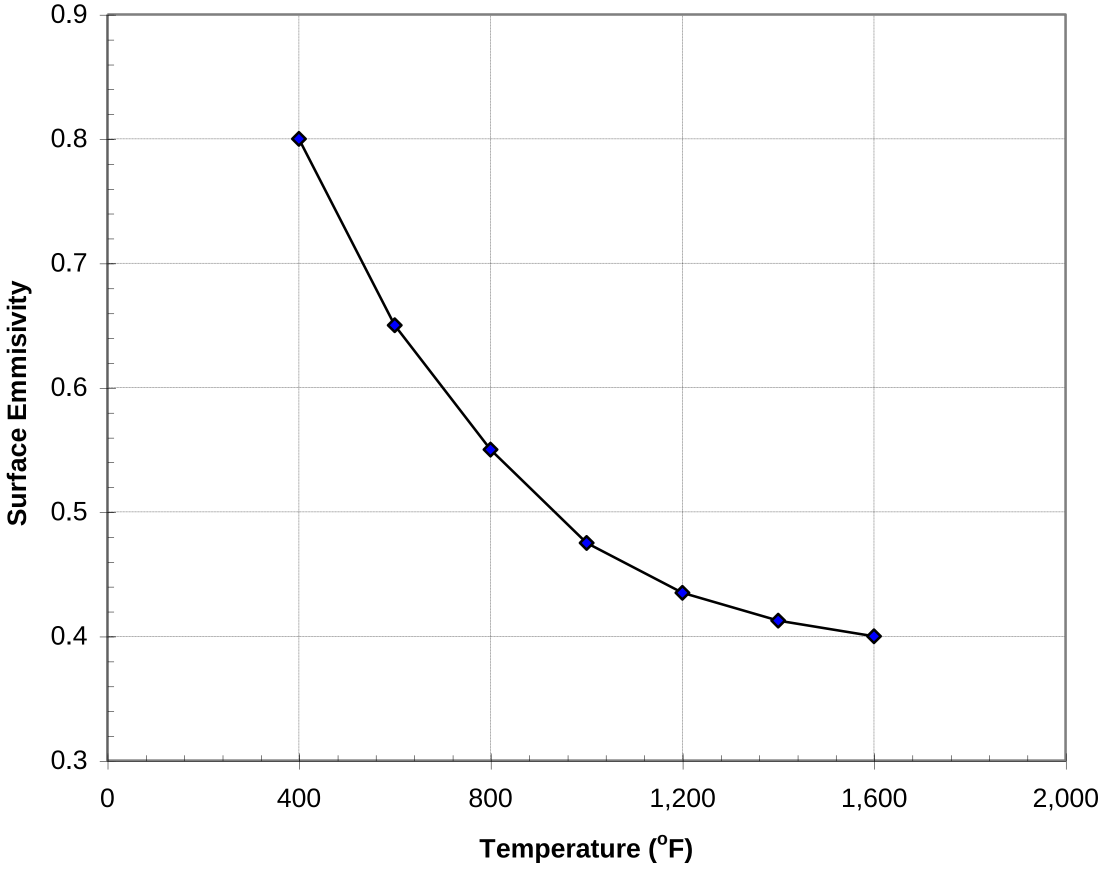
Figure 17-18. 1-Dimensional Bar Example Problem with Constrained End Temperature and a Radiation Boundary Condition.

The `RADBC` Bulk Data entry defines the radiation properties and the ambient grid point. The basic exchange relationship defines the rate of heat transfer as:

$$q = \sigma * \mathrm{FAMB} * (\epsilon T^4 - \alpha \mathrm{TAMB}^4)$$

where σ is the Stefan-Boltzmann constant, FAMB is the radiation view factor between the surface and the ambient point, ϵ and α are the temperature dependent surface emissivity and absorptivity given on the `RADM` Bulk Data entry, and T and TAMB are the surface and ambient temperatures, respectively. For our example, emissivity and absorptivity are equal with the emissivity plotted as a function of temperature in Figure 17-19.

Figure 17-19. `TABLEM2` Bulk Data Entry Emissivity vs. Temperature Input Data.



The grid point temperatures from the Model Results Output File are shown in tabular form in Listing 17-18 and plotted against percent of bar length in Figure 17-20. Rod element thermal gradients and heat fluxes are shown in tabular listing form in Listing 17-19. Thermal gradients are plotted against percent of bar length in Figure 17-21. Heat flows into the hbdy elements are shown in tabular form in Listing 17-20 and plotted against percent of bar length in Figure 17-22.

Listing 17-17. Model Input File for Bar Model with Constrained End Temperature and a Radiation Boundary Condition.

```
$
$ NONLINEAR STEADY STATE HEAT TRANSFER SOLUTION.
$
SOL NONLINEAR STEADY STATE HEAT TRANSFER
$
ANALYSIS = HEAT
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = BAR WITH NONLINEAR TEMPERATURE DEPENDENT MATERIAL
$
THERMAL = ALL
SPCFORCE = ALL
FLUX = ALL
$
TEMPERATURE(INITIAL) = 1
SUBCASE 1
 LABEL = CONSTRAINED END TEMPERATURE AND RADIATION BOUNDARY
 SPC = 1
 NLPARM = 1
$
BEGIN BULK
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  1
$
$ DEFINE STEFAN-BOLTZMANN CONSTANT.
$
PARAM, SIGMA, 3.97E-14
$
$ FAHRENHEIT TO ABSOLUTE TEMPERATURE CONVERSION FACTOR.
$
PARAM, TABS, 459.69
$
$ GEOMETRY DEFINITION (10" BAR DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  ,   0.,  0.,  0.,
GRID,  2,  ,   1.,  0.,  0.,
GRID,  3,  ,   2.,  0.,  0.,
GRID,  4,  ,   3.,  0.,  0.,
GRID,  5,  ,   4.,  0.,  0.,
GRID,  6,  ,   5.,  0.,  0.,
GRID,  7,  ,   6.,  0.,  0.,
GRID,  8,  ,   7.,  0.,  0.,
GRID,  9,  ,   8.,  0.,  0.,
GRID, 10,  ,   9.,  0.,  0.,
GRID, 11,  ,  10.,  0.,  0.,
GRID, 12,  ,   0.,  0.,  0.,
$
$ CIRCULAR BAR MODELED WITH ROD ELEMENTS.
$
CROD,  1,  10,   1,   2
CROD,  2,  10,   2,   3
CROD,  3,  10,   3,   4
CROD,  4,  10,   4,   5
CROD,  5,  10,   5,   6
CROD,  6,  10,   6,   7
CROD,  7,  10,   7,   8
CROD,  8,  10,   8,   9
CROD,  9,  10,   9,  10
CROD, 10,  10,  10,  11
$
$ ROD ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" DIAMETER).
$
PROD,  10,  100,  0.7854
```

Listing 17-17. Model Input File for Bar Model with Constrained End Temperature and a Radiation Boundary Condition. (Continued)

```
$
$ ELEMENT CONDUCTIVITY (MA956).
$
MAT4,  100,  1.
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
MATT4,  100,  10
$
$ TEMPERATURE DEPENDENT CONDUCTIVITY DATA.
$
TABLEM2,  10,
,    70.,  1.466E-4,   200.,  1.620E-4,   400.,  1.852E-4,   600.,  2.083E-4,
,   800.,  2.315E-4,  1000.,  2.508E-4,  1200.,  2.739E-4,  1400.,  2.951E-4,
,  1600.,  3.164E-4,  1800.,  3.376E-4,  ENDT
$
$ CONVECTION BOUNDARY DEFINITION.
$
CHBDYP,  11,  20,  LINE,  ,  ,   1,   2,
  ,  200,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  12,  20,  LINE,  ,  ,   2,   3,
  ,  200,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  13,  20,  LINE,  ,  ,   3,   4,
  ,  200,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  14,  20,  LINE,  ,  ,   4,   5,
  ,  200,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  15,  20,  LINE,  ,  ,   5,   6,
  ,  200,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  16,  20,  LINE,  ,  ,   6,   7,
  ,  200,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  17,  20,  LINE,  ,  ,   7,   8,
  ,  200,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  18,  20,  LINE,  ,  ,   8,   9,
  ,  200,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  19,  20,  LINE,  ,  ,   9,  10,
  ,  200,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  20,  20,  LINE,  ,  ,  10,  11,
  ,  200,  ,  ,  ,  0.,  1.,  0.
$
$ AREA FACTOR TO DEFINE ROD END SURFACE AREA
$ AREA = (PI/4)*DIAMETER**2 = 0.7854
$
PHBDY,  20,  0.7854
$
$ RADIATION LOAD.
$
RADBC,  12,  1.,  ,  11,  THRU,  20
$
$ RADIATION BOUNDARY MATERIAL PROPERTIES.
$
RADM,  200,  1.,  1.
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
RADMT,  200,  20,  20
$
$ TEMPERATURE DEPENDENT EMISSIVITY AND ABSORPTIVITY DATA.
$
TABLEM2,  20,
,   400.,  0.800,   600., 0.6500,   800.,  0.550,  1000.,  0.475,
,  1200.,  0.435,  1400., 0.4125,  1600.,  0.400,  ENDT
```

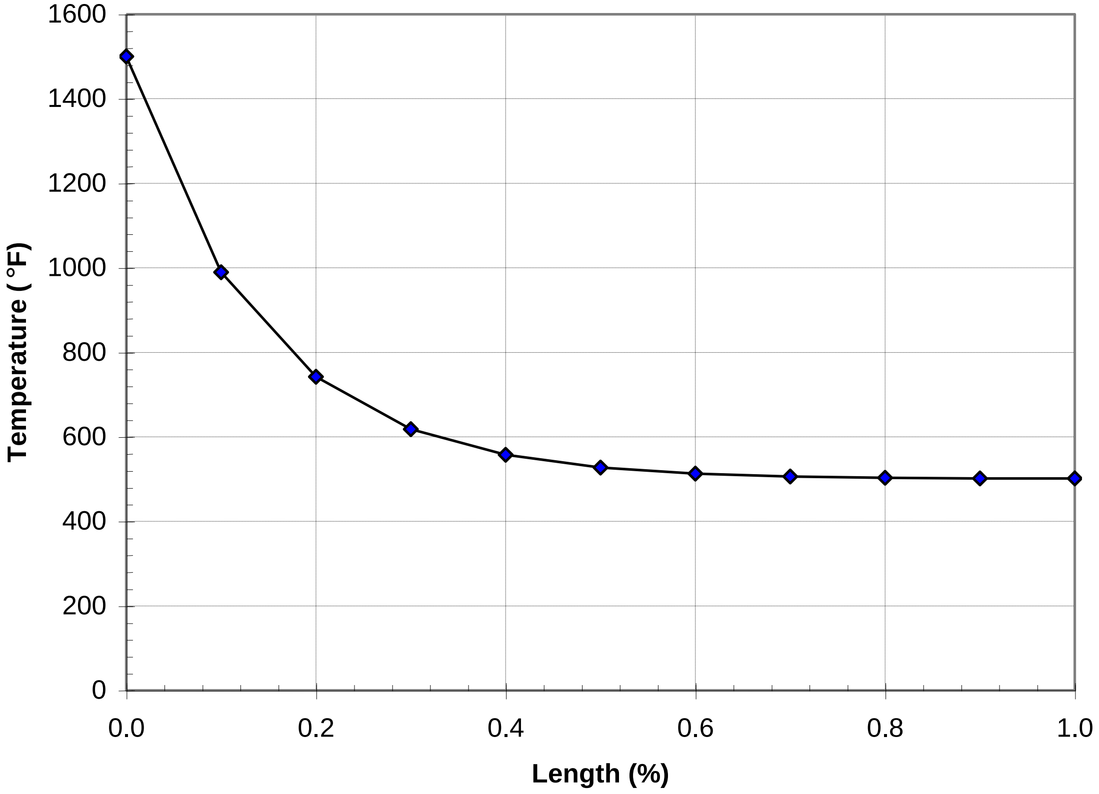
Listing 17-17. Model Input File for Bar Model with Constrained End Temperature and a Radiation Boundary Condition. (Continued)

```
$
$ BOUNDARY TEMPERATURE DEFINITION.
$
SPC,  1,  12,  1,  500.
$
$ CONSTRAIN BAR END TEMPERATURE.
$
SPC,  1,  1,  1,  1500.
$
$ INITIAL TEMPERATURE DISTRIBUTION.
$
TEMP,  1,  12,  500.
TEMPD,  1,  1500.
ENDDATA
```

Listing 17-18. Temperature Vector for Bar Model with Constrained End Temperature and a Radiation Boundary Condition.

```
                                         T E M P E R A T U R E   V E C T O R

     GRID      TEMPERATURE
      ID
        1      1.500000E+03
        2      9.894889E+02
        3      7.417983E+02
        4      6.182798E+02
        5      5.572263E+02
        6      5.274541E+02
        7      5.131228E+02
        8      5.062922E+02
        9      5.030968E+02
       10      5.017062E+02
       11      5.013176E+02
       12      5.000000E+02
```

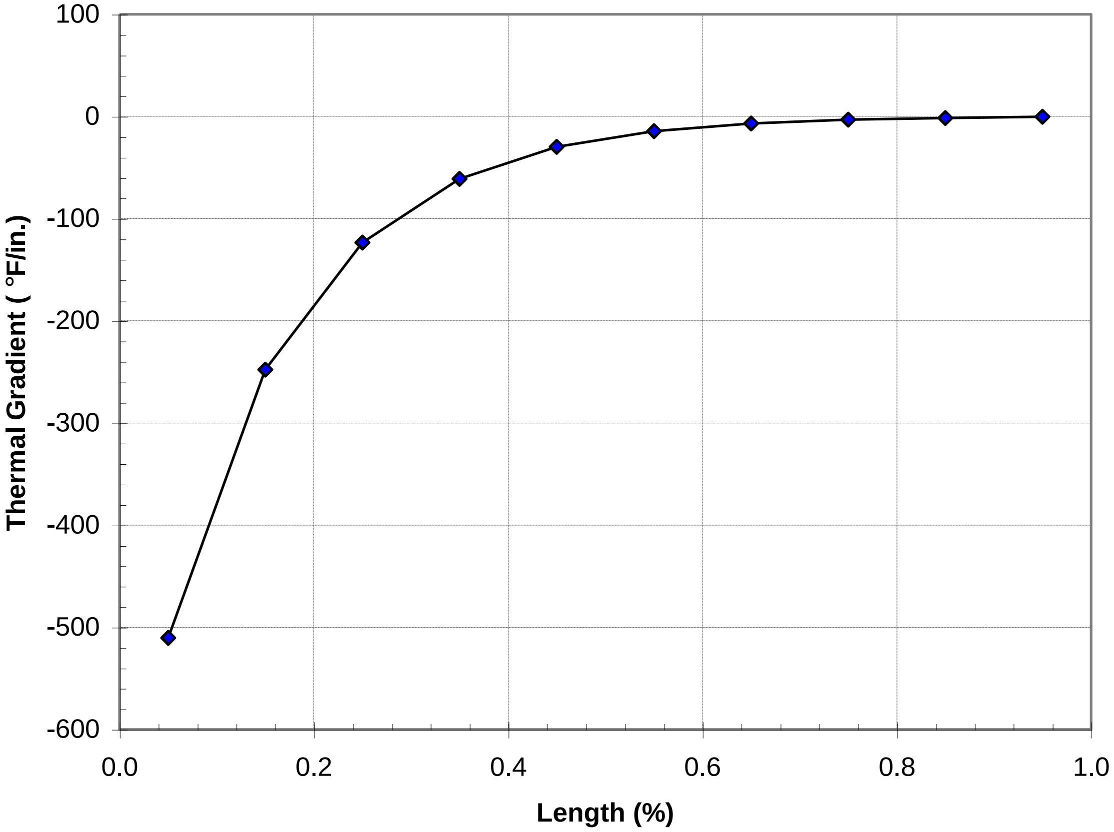
Figure 17-20. Temperature vs. Bar Model Normalized Length.



Listing 17-19. Element Thermal Gradients and Heat Fluxes for Bar Model with Constrained End Temperature and a Radiation Boundary Condition.

```
              T H E R M A L   G R A D I E N T S   A N D   H E A T   F L U X E S   I N   R O D   E L E M E N T S

  ELEMENT         GRADIENT            FLUX
    ID
     1       -5.105111E+02      1.422503E-01
     2       -2.476905E+02      5.890938E-02
     3       -1.235186E+02      2.687573E-02
     4       -6.105351E+01      1.263108E-02
     5       -2.977215E+01      6.003266E-03
     6       -1.433133E+01      2.853272E-03
     7       -6.830613E+00      1.351582E-03
     8       -3.195369E+00      6.304214E-04
     9       -1.390655E+00      2.739972E-04
    10       -3.885286E-01      7.651085E-05
```

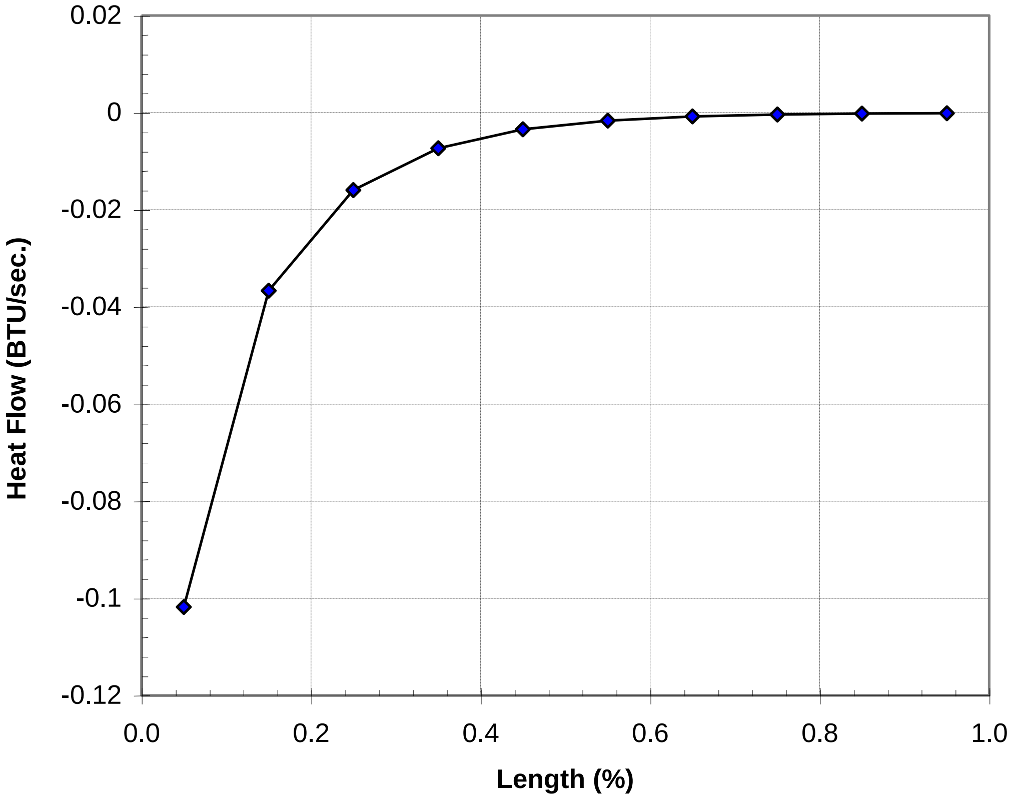
Figure 17-21. Thermal Gradient vs. Bar Model Normalized Length.



Listing 17-20. Heat Flow Into Hbdy Elements for Bar Model with Constrained End Temperature and a Radiation Boundary Condition.

```
                                        H E A T   F L O W   I N T O   H B D Y   E L E M E N T S

     ELEMENT       APPLIED       CONVECTION     RADIATION        TOTAL
       ID
        11       0.000000E+00   0.000000E+00  -1.017733E-01  -1.017733E-01
        12       0.000000E+00   0.000000E+00  -3.664700E-02  -3.664700E-02
        13       0.000000E+00   0.000000E+00  -1.595929E-02  -1.595929E-02
        14       0.000000E+00   0.000000E+00  -7.305986E-03  -7.305986E-03
        15       0.000000E+00   0.000000E+00  -3.456245E-03  -3.456245E-03
        16       0.000000E+00   0.000000E+00  -1.638538E-03  -1.638538E-03
        17       0.000000E+00   0.000000E+00  -7.797943E-04  -7.797943E-04
        18       0.000000E+00   0.000000E+00  -3.761259E-04  -3.761259E-04
        19       0.000000E+00   0.000000E+00  -1.921777E-04  -1.921777E-04
        20       0.000000E+00   0.000000E+00  -1.209322E-04  -1.209322E-04
```

Figure 17-22. Surface Heat Flow vs. Bar Model Normalized Length.



The next problem is another example of nonlinear radiation. In this example a heat flux is applied to the surface of a flat plate with a radiation boundary condition as shown in Figure 17-23. Listing 17-21 contains the Model Input File. The grid point temperatures from the Model Results Output File are shown in tabular form in Listing 17-22. Heat flows into the hbdy elements are shown in Listing 17-23.

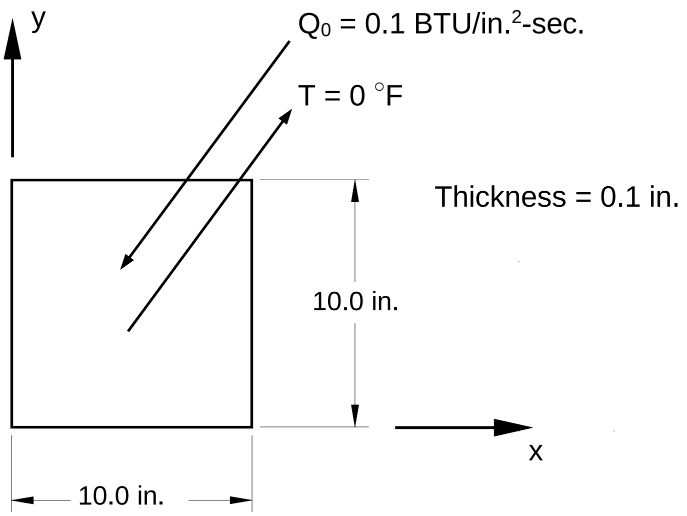


Figure 17-23. 3-D Plate Example Problem with an Applied Heat Flux and a Radiation Boundary Condition.

For this example, the energy balance can be expressed as:

$$Q = \sigma A \varepsilon F (T_e^4 - T_\infty^4)$$

or,

$$(0.1 \text{ BTU/sec.-in.}^2)(100.0 \text{ in.}^2) = (3.97\text{E-14 BTU/sec.-in.}^2\text{-}^\circ\text{R}^4)(100.0 \text{ in.}^2)(1.0)(1.0)(T_e^4 - (459.69)^4)$$

which gives,

$$T_e = 805.7\ ^\circ\text{F}$$

Listing 17-21. Model Input File for 3-D Plate with an Applied Heat Flux and a Radiation Boundary Condition.

```
$
$ NONLINEAR STEADY STATE HEAT TRANSFER SOLUTION.
$
SOL NONLINEAR STEADY STATE HEAT TRANSFER
$
ANALYSIS = HEAT
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = PLATE IN RADIATIVE EQUILIBRIUM
$
THERMAL = ALL
SPCFORCE = ALL
FLUX = ALL
$
TEMPERATURE(INITIAL) = 1
SUBCASE 1
 LABEL = APPLIED HEAT FLUX WITH RADIATION BOUNDARY
 LOAD = 1
 NLPARM = 1
 SPC = 1
$
BEGIN BULK
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  1
$
$ DEFINE STEFAN-BOLTZMANN CONSTANT.
$
PARAM, SIGMA, 3.97E-14
$
$ FAHRENHEIT TO ABSOLUTE TEMPERATURE CONVERSION FACTOR.
$
PARAM, TABS, 459.69
$
$ GEOMETRY DEFINITION (10" X 10" PLATE WITH A 1 X 1 MESH).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,  10.,  0.,  0.,  0
GRID,  3,  0,  10., 10.,  0.,  0
GRID,  4,  0,   0., 10.,  0.,  0
GRID,  5,  0,   0.,  0.,  0.,  0
$
$ BLOCK MODELED WITH A SHELL ELEMENT.
$
CQUAD4,  1,  30,  1,  2,  3,  4
$
$ ELEMENT MATERIAL AND THICKNESS (0.1").
$
PSHELL,  30,  100,  0.1,  100,  ,  100
$
$ ELEMENT CONDUCTIVITY (MA956).
$
MAT4,  100,  1.
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
MATT4,  100,  10
$
$ TEMPERATURE DEPENDENT CONDUCTIVITY DATA.
$
TABLEM2,  10,
,    70.,  1.466E-4,   200.,  1.620E-4,   400.,  1.852E-4,   600.,  2.083E-4,
,   800.,  2.315E-4,  1000.,  2.508E-4,  1200.,  2.739E-4,  1400.,  2.951E-4,
,  1600.,  3.164E-4,  1800.,  3.376E-4,  ENDT
```

Listing 17-21. Model Input File for 3-D Plate with an Applied Heat Flux and a Radiation Boundary Condition. (Continued)

```
$
$ RADIATION BOUNDARY DEFINITION.
$
CHBDYG,  2,  ,  AREA4,  ,  ,  200,  ,  ,
  ,  1,  2,  3,  4
$
$ RADIATION LOAD.
$
RADBC,  5,  1.,  ,  2
$
$ RADIATION BOUNDARY MATERIAL PROPERTIES.
$
RADM,  200,  1.,  1.
$
$ HEAT FLUX.
$
QBDY1,  1,  0.1,  2
$
$ BOUNDARY TEMPERATURE DEFINITION.
$
SPC,  1,  5,  1,  0.
$
$ INITIAL TEMPERATURE DISTRIBUTION.
$
TEMPD,  1,  0.
ENDDATA
```

Listing 17-22. Temperature Vector for 3-D Plate with an Applied Heat Flux and a Radiation Boundary Condition.

```
                                          T E M P E R A T U R E   V E C T O R

     GRID       TEMPERATURE
      ID
       1        8.056589E+02
       2        8.056589E+02
       3        8.056589E+02
       4        8.056589E+02
```

Listing 17-23. Heat Flow Into Hbdy Elements for 3-D Plate with an Applied Heat Flux and a Radiation Boundary Condition.

```
                                  H E A T   F L O W   I N T O   H B D Y   E L E M E N T S

    ELEMENT        APPLIED        CONVECTION      RADIATION         TOTAL
      ID
       2       1.000000E+01   0.000000E+00  -1.000000E+01  -1.520630E-06
```

The next problem is an example of enclosure radiation. In this example two square plates are part of a radiation enclosure with the radiation boundary conditions as shown in Figure 17-24. Listing 17-24 contains the Model Input File. Heat flows into the hbdy elements are shown in Listing 17-25.

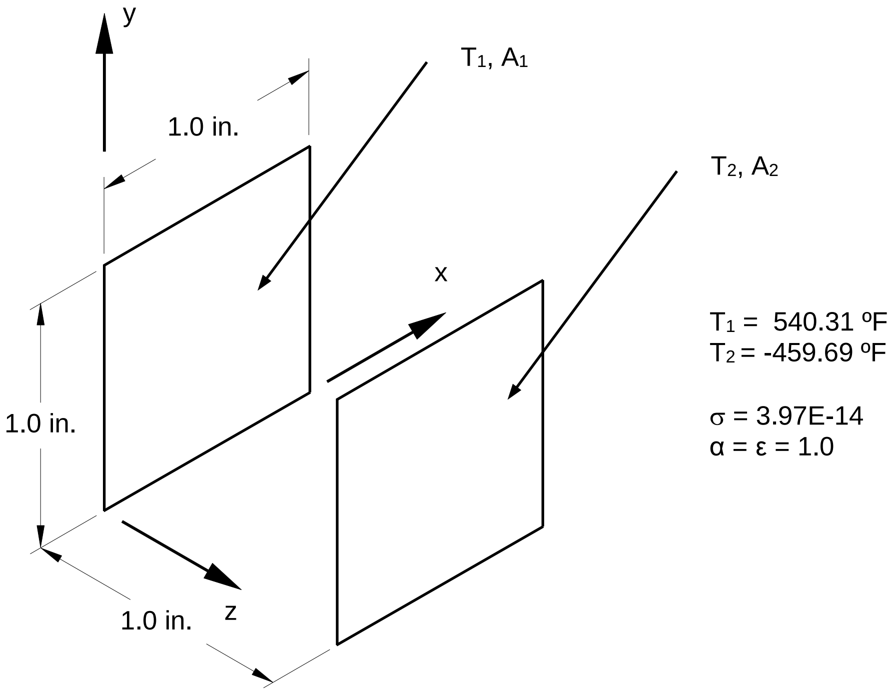


Figure 17-24. Two Plates Radiation Enclosure.

The `RADCAV` Bulk Data entry defines a radiation cavity and uses the parameters defined through `VIEW`and `VIEW3D` Bulk Data entries for view factor calculations. The `RADSET` Bulk Data entry specifies which radiation cavities are to be included for the radiation enclosure analysis. The `RADM` Bulk Data entry defines the absorptivity, α, and the emissivity, ε. If an absorptivity is not defined, it is assumed to be equal emissivity. The Stefan-Boltzmann constant, σ, is defined through the parameter SIGMA. When modeling radiation, if the temperatures in the model are defined in Fahrenheit or Celsius, the parameter TABS is needed to convert to absolute temperatures.

For this example, the energy balance can be expressed as:

$$\{Q\} = -[R]\{T\}^4$$

where,

$\{Q\}$	is the heat flow vector
$[R]$	is the radiation exchange matrix
$\{T\}$	is the vector of absolute temperatures

$$\{q\}^{in} = \sigma\left[(A - F(I - \alpha))^{-1} F\varepsilon\right]\{T\}^4$$

$$\{q\}^{out} = \sigma\left[\varepsilon + (I - \alpha)(A - F(I - \alpha))^{-1} F\varepsilon\right]\{T\}^4$$

$$\{Q\} = -[A]\left(\{q\}^{in} - \{q\}^{out}\right) = -[R]\{\mathrm{T}\}^4$$

where,

$\{q\}^{in}$	is the irradiation vector
$\{q\}^{out}$	is the radiosity vector
σ	is the Stefan-Boltzmann constant
A	is the diagonal matrix element areas
F	is the matrix of exchange coefficients, $A_i f_{ij}$
f_{ij}	is the view factor
α	is the diagonal matrix of surface absorptivities
ε	is the diagonal matrix of surface emissivities

The radiation exchange matrix, $[R]$, is defined as:

$$[R] = \sigma\left[A\varepsilon - A\alpha(A - F(I - \alpha))^{-1} F\varepsilon\right]$$

Assume the plates are black bodies, therefore $\alpha_1 = \alpha_2 = \varepsilon_1 = \varepsilon_2 = 1.0$

$$A\varepsilon = \begin{bmatrix} A_1\varepsilon_1 & 0 \\ 0 & A_2\varepsilon_2 \end{bmatrix} = \begin{bmatrix} A_1 & 0 \\ 0 & A_2 \end{bmatrix}$$

$$A\alpha = \begin{bmatrix} A_1\alpha_1 & 0 \\ 0 & A_2\alpha_2 \end{bmatrix} = \begin{bmatrix} A_1 & 0 \\ 0 & A_2 \end{bmatrix}$$

$$F\varepsilon = \begin{bmatrix} 0 & A_1 f_{12}\varepsilon_2 \\ A_2 f_{21}\varepsilon_1 & 0 \end{bmatrix} = \begin{bmatrix} 0 & A_1 f_{12} \\ A_2 f_{21} & 0 \end{bmatrix}$$

$$(A - F(I - \alpha))^{-1} = \left[\begin{bmatrix} A_1 & 0 \\ 0 & A_2 \end{bmatrix} - \begin{bmatrix} 0 & A_1 f_{12} \\ A_2 f_{21} & 0 \end{bmatrix}\begin{bmatrix} 1-\alpha_1 & 0 \\ 0 & 1-\alpha_2 \end{bmatrix}\right]^{-1}$$

$$(A - F(I - \alpha))^{-1} = \begin{bmatrix} A_1 & -A_1 f_{12}(1-\alpha_2) \\ -A_2 f_{21}(1-\alpha_1) & A_2 \end{bmatrix}^{-1} = \begin{bmatrix} \frac{1}{A_1} & 0 \\ 0 & \frac{1}{A_2} \end{bmatrix}$$

$$[R] = \sigma\left[\begin{bmatrix} A_1 & 0 \\ 0 & A_2 \end{bmatrix} - \begin{bmatrix} A_1 & 0 \\ 0 & A_2 \end{bmatrix}\begin{bmatrix} \dfrac{1}{A_1} & 0 \\ 0 & \dfrac{1}{A_2} \end{bmatrix}\begin{bmatrix} 0 & A_1 f_{12} \\ A_2 f_{21} & 0 \end{bmatrix}\right]$$

$$[R] = \sigma\begin{bmatrix} A_1 & -A_1 f_{12} \\ -A_2 f_{21} & A_2 \end{bmatrix}$$

$$\{Q\} = -[R]\{\mathrm{T}\}^4$$

$$\{Q\} = -\sigma\begin{bmatrix} A_1 & -A_1 f_{12} \\ -A_2 f_{21} & A_2 \end{bmatrix}\begin{Bmatrix} T_1{}^4 \\ T_2{}^4 \end{Bmatrix}$$

$$\begin{Bmatrix} Q_1 \\ Q_2 \end{Bmatrix} = -\sigma\begin{Bmatrix} A_1 T_1{}^4 - A_1 f_{12} T_2{}^4 \\ -A_2 f_{21} T_1{}^4 + A_2 T_2{}^4 \end{Bmatrix}$$

In this model:

$$A_1 = A_2 = 1.0$$

$$f_{12} = f_{21} \approx 0.2$$

which gives:

$$Q_1 = -\sigma\left(T_1{}^4 - (0.2) T_2{}^4\right) = -\sigma\left(\left(1000^4\right) - (0.2)\left(0^4\right)\right) = -3.97x10^{-2}$$

$$Q_2 = -\sigma\left(-A_2 f_{21} T_1{}^4 + A_2 T_2{}^4\right) = -\sigma\left(-(0.2)\left(1000^4\right) + \left(0^4\right)\right) = 7.94x10^{-3}$$

Listing 17-24. Model Input File for Two Plates Radiation Enclosure.

```
$
$ NONLINEAR STEADY STATE HEAT TRANSFER SOLUTION.
$
SOL NONLINEAR STEADY STATE HEAT TRANSFER
$
ANALYSIS = HEAT
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = TWO PLATE RADIATION ENCLOSURE
$
THERMAL = ALL
SPCFORCE = ALL
FLUX = ALL
$
TEMPERATURE(INITIAL) = 1
SUBCASE 1
 LABEL = RADIATION ENCLOSURE
 LOAD = 1
 NLPARM = 1
 SPC = 1
$
BEGIN BULK
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  1,
,  ,  ,  ,  ,  , 6,  , 0.2
$
$ DEFINE STEFAN-BOLTZMANN CONSTANT.
$
PARAM, SIGMA, 3.97E-14
$
$ FAHRENHEIT TO ABSOLUTE TEMPERATURE CONVERSION FACTOR.
$
PARAM, TABS, 459.69
$
$ GEOMETRY DEFINITION (TWO 1" X 1" PLATES WITH 1 X 1 MESH, 1" APART).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   1.,  1.,  0.,  0
GRID,  4,  0    0.,  1.,  0.,  0
GRID,  5,  0,   0.,  0.,  1.,  0
GRID,  6,  0,   1.,  0.,  1.,  0
GRID,  7,  0,   1.,  1.,  1.,  0
GRID,  8,  0,   0.,  1.,  1.,  0
$
$ PLATES MODELED WITH SHELL ELEMENTS.
$
CQUAD4,  1,  30,  1,  2,  3,  4
CQUAD4,  2,  30,  5,  6,  7,  8
```

Listing 17-24. Model Input File for Two Plates Radiation Enclosure. (Continued)

```
$
$ ELEMENT MATERIAL AND THICKNESS (0.1").
$
PSHELL,  30,  100,  0.1,  100,  ,  100
$
$ ELEMENT MATERIAL.
$
MAT4,  100,  1.
$
$ RADIATION BOUNDARY DEFINITION.
$
CHBDYG,  3,  ,  AREA4,  1,  ,  200,  ,  ,
  ,  1,  2,  3,  4
CHBDYG,  4,  ,  AREA4,  1,  ,  200,  ,  ,
  ,  5,  8,  7,  6
$
$ RADIATION BOUNDARY MATERIAL PROPERTIES.
$
RADM,  200,  1.,  1.
$
$ VIEW FACTOR DEFNITION.
$
VIEW,  1,  400,  BOTH
$
$ VIEW FACTOR DEFINITION - GAUSSIAN INTEGRATION METHOD.
$
VIEW3D,  400
$
$ RADIATION CAVITY IDENTIFICATION.
$
RADCAV,  400
$
$ IDENTIFIES A SET OF RADIATION CAVITIES.
$
RADSET,  400
$
$ BOUNDARY TEMPERATURE DEFINITION.
$
SPC,  1,  1,  1,   540.31
SPC,  1,  2,  1,   540.31
SPC,  1,  3,  1,   540.31
SPC,  1,  4,  1,   540.31
SPC,  1,  5,  1,  -459.69
SPC,  1,  6,  1,  -459.69
SPC,  1,  7,  1,  -459.69
SPC,  1,  8,  1,  -459.69
$
$ INITIAL TEMPERATURE DISTRIBUTION.
$
TEMPD,  1,  -459.69
ENDDATA
```

Listing 17-25. Heat Flow Into Hbdy Elements from Two Plates Radiation Enclosure.

```
                                   H E A T   F L O W   I N T O   H B D Y   E L E M E N T S

ELEMENT       APPLIED       CONVECTION     RADIATION        TOTAL
  ID
    3       0.000000E+00   0.000000E+00   3.970000E-02   3.970000E-02
    4       0.000000E+00   0.000000E+00  -7.933051E-03  -7.933051E-03
```

17.5 Grid Point Temperature Generation

Temperatures generated from heat transfer solutions can be used directly in structural analysis in Autodesk Inventor Nastran. This section discusses how to translate output grid point temperature into `TEMP` Bulk Data entries. The `TEMP` Bulk Data entries are used to define temperature distributions within the model, which are further used to define temperature dependent material properties and thermal loading.

At the end of a heat transfer solution, grid point temperatures are translated into `TEMP` Bulk Data entries and output to the Bulk Data Output File (*filename*.BDF). This operation is controlled with the `TRSLMODLDATA`, `TRSLTEMPDATA`, and `OUTTEMPSETID` Model Initialization directives (see *Nastran Solver Reference Guide*, Section 2, *Initialization,* for directive format). When `TRSLTEMPDATA` is set to `ON`, the result heat transfer grid point temperatures are output using the setid specified by `OUTTEMPSETID`. These temperatures can then be imported directly into a modeler or merged into a Model Input File.

As an example, we will use the conduction problem shown in Figure 17-2. Listing 17-26 shows the model input file for a nonlinear static solution.

Listing 17-26. Model Input File for Thermally loaded Bar Model with Constrained Ends.

```
$
$ NONLINEAR STATIC SOLUTION.
$
SOL NONLINEAR STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = CONSTRAINED BAR WITH TEMPERATURE DEPENDENT MATERIAL
$
STRESS = ALL
FORCE = ALL
TEMPERATURE(INITIAL) = 1
SUBCASE 1
 LABEL = THERMAL GRADIENT PRODUCED FROM NONLINEAR HEAT TRANSFER SOLUTION
 TEMPERATURE(BOTH) = 2
 NLPARM = 1
 SPC = 1
$
BEGIN BULK
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  10,  ,  ,  ,  ,  ,  YES
$
$ GEOMETRY DEFINITION (10" BAR DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
```

Listing 17-26. Model Input File for Thermally Loaded Bar Model with Constrained Ends. (Continued)

```
$
$ CIRCULAR BAR MODELED WITH ROD ELEMENTS.
$
CROD,  1,  10,   1,   2
CROD,  2,  10,   2,   3
CROD,  3,  10,   3,   4
CROD,  4,  10,   4,   5
CROD,  5,  10,   5,   6
CROD,  6,  10,   6,   7
CROD,  7,  10,   7,   8
CROD,  8,  10,   8,   9
CROD,  9,  10,   9,  10
CROD, 10,  10,  10,  11
$
$ ROD ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" DIAMETER).
$
PROD,  10,  100,  0.7854
$
$ LINEAR ELEMENT MATERIAL PROPERTIES (MA956).
$
MAT1,  100,  1.,  ,  0.33,  0.3,  1.
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
MATT1,  100,  10,  ,  ,  ,  20
$
$ TEMPERATURE DEPENDENT MODULUS OF ELASTICITY.
$
TABLEM2,  10,
,    70.,  39.0E+6,   200.,  38.4E+6,   400.,  37.4E+6,   600.,  36.5E+6,
,   800.,  35.1E+6,  1000.,  33.9E+6,  1200.,  32.5E+6,  1400.,  30.6E+6,
,  1600.,  29.4E+6,  1800.,  28.0E+6,  ENDT
$
$ TEMPERATURE DEPENDENT COEFFICIENT OF THERMAL EXPANSION.
$
TABLEM2,  20,
,    70.,  6.00E-6,   200.,  6.25E-6,   400.,  6.47E-6,   600.,  6.67E-6,
,   800.,  6.89E-6,  1000.,  7.11E-6,  1200.,  7.33E-6,  1400.,  7.61E-6,
,  1600.,  7.89E-6,  1800.,  8.22E-6,  ENDT
$
$ FIXED AT BOTH ENDS.
$
SPC1,  1,  123456,  1,  11
SPC1,  1,   23456,  2,  THRU,  10
$
$ INITIAL TEMPERATURE DISTRIBUTION.
$
TEMPD,  1,  70.
$
$ TEMPERATURE DISTRIBUTION FROM HEAT TRANSFER ANALYSIS.
$
TEMP,  2,   1,     0.0
TEMP,  2,   2,   134.2
TEMP,  2,   3,   255.8
TEMP,  2,   4,   368.0
TEMP,  2,   5,   472.7
TEMP,  2,   6,   571.2
TEMP,  2,   7,   664.5
TEMP,  2,   8,   753.4
TEMP,  2,   9,   838.4
TEMP,  2,  10,   920.5
TEMP,  2,  11,  1000.0
ENDDATA
```

18. NONLINEAR TRANSIENT HEAT TRANSFER ANALYSIS

18.1 Introduction

In the previous section we dealt with steady state heat transfer where loads and boundary conditions did not vary with time. When steady state conditions do not prevail, temperature change in a unit volume of material is resisted by thermal mass that depends on the mass density of the material and its specific heat. The general heat transfer equation then becomes

$$[K]\{T\}+[C]\{\dot{T}\}=\{R\}$$

where,

$[K]$ is the global conductivity matrix

$[C]$ is the global capacitance matrix

$\{T\}$ is the global temperature vector

$\{\dot{T}\}$ is $\partial T/\partial t$

$\{R\}$ is the global thermal load vector

Nonlinear transient heat transfer analysis is implemented in Autodesk Inventor Nastran using Newmark’s method with adaptive time stepping. Loads can be both temperature and time dependent. Nonlinear effects like temperature dependent materials and radiation can also be included. Convergence is achieved at each time step using the same Newton-Raphson iteration method used in nonlinear steady state heat transfer analysis.

18.2 How to Setup a Model Input File for Nonlinear Transient Heat Transfer Analysis

In Autodesk Inventor Nastran you can solve a nonlinear transient heat transfer problem by setting `SOLUTION = NONLINEAR TRANSIENT HEAT TRANSFER` in the Model Initialization File or by specifying `SOL 159` or `SOL NONLINEAR TRANSIENT HEAT TRANSFER` above the Case Control Section and `ANALYSIS = HEAT` in the Case Control Section of the Model Input File. The following the guidelines listed below:

1. Most nonlinear transient response problems can be setup the same as for linear transient response (geometry, boundary conditions, loading, etc.). One exception is that initial conditions are specified using the `IC` Case Control command which references a starting temperature distribution in the Bulk Data. As a minimum, all subcases must reference a `TSTEPNL` Bulk Data entry via the `TSTEPNL` Case Control command. The `TSTEPNL` entry is a combination of the `TSTEP` entry used in linear transient response and the `NLPARM` entry used in nonlinear statics. It controls both the direct time integration (number of time steps, time increment, output interval, etc.), and the nonlinear iteration parameters (maximum iterations permitted, convergence method and tolerances, etc.). Since the solution to a particular load involves a nonlinear search procedure, the solution is not guaranteed. Care must be used when selecting the search procedures on the `TSTEPNL` Bulk Data entry. You may override nearly all iteration control restrictions.
2. All loads and material properties that are supported in steady state heat transfer analysis are supported in nonlinear transient heat transfer analysis.
3. All grid points must have an initial temperature defined. The `TEMPD` Bulk Data entry can be used for this purpose. The `IC` Case Control command is then used to reference this temperature set.
4. Unlike other solutions, subcase loads and results are additive. This allows different loads and boundary conditions to be applied in a specific sequence to the structure. Additionally, different time integration and nonlinear iteration parameters (`TSTEPNL`) may be specified for each subcase allowing further control.
5. Models should be simple and relatively small initially to gain insight into behavior and verify the approach taken. A linear heat transfer solution should be run first to verify boundary conditions and loading.

18.3 Interpreting Results

In this section we will present several examples demonstrating the features and capabilities of nonlinear transient heat transfer analysis. We will look at several types of thermal loading and boundary conditions.

18.3.1 Volume Heat Addition

The first problem is an example of transient volume heat addition. The circular bar in Figure 18-1 is initially at 0 ^{o}F. The volumetric heat addition load is both transient and temperature dependent and is applied to the first element only. Listing 18-1 contains the Model Input File.

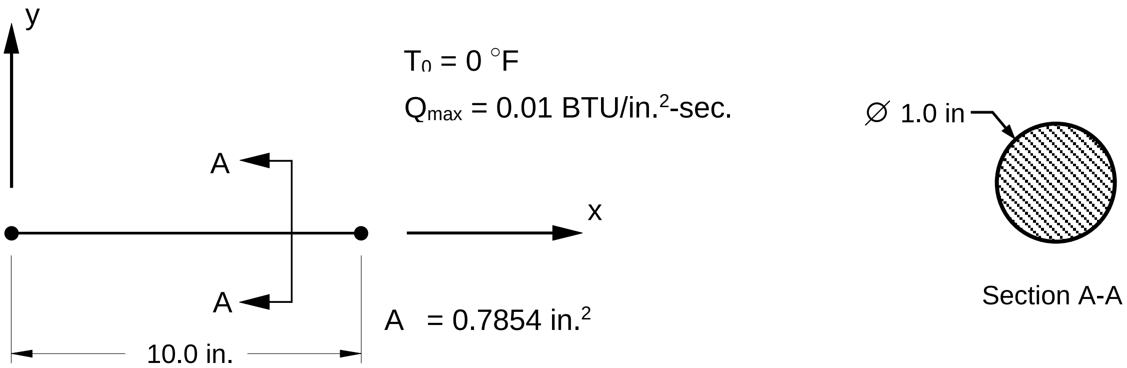


Figure 18-1. 1-Dimensional Bar Example Problem with Transient Volumetric Heat Addition.

Steady state volumetric heat addition results in elemental power input given by the equation:

$$P_{in} = Volume * \text{HGEN} * \text{QVOL}$$

Where QVOL is the power density given on the `QVOL` Bulk Data entry and HGEN is a temperature dependent scale factor given on the `MAT4` or `MAT5` Bulk Data entry. Unlike structural solutions which require a load sequence entry (`LSEQ`) to define a dynamic area (reference link), heat transfer load sets are directly referenced on `TLOADi` Bulk Data entries. The time-dependent dynamic load (`TLOAD1`) then references the area factor defined by the `QVOL` set identification number for spatial definition (area) and a `TABLED1` for temporal definition (time). The `DLOAD` Bulk Data entry is optionally used to combine and scale dynamic loads defined using the `TLOADi` Bulk Data entries. The `DLOAD` and/or `TLOADi` Bulk Data entries are called out in the Case Control Section using the `DLOAD` Case Control command. The resulting load time history is shown graphically in Figure 18-2.

Listing 18-1. Model Input File for Bar Model with Transient Volumetric Heat Addition.

```
$
$ NONLINEAR TRANSIENT HEAT TRANSFER SOLUTION.
$
SOL NONLINEAR TRANSIENT HEAT TRANSFER
$
ANALYSIS = HEAT
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = BAR WITH NONLINEAR TEMPERATURE DEPENDENT MATERIAL
$
THERMAL = ALL
FLUX = ALL
$
IC = 10
SUBCASE 1
 LABEL = TRANSIENT VOLUME HEAT ADDITION
 DLOAD = 1
 TSTEPNL = 1
$
BEGIN BULK
$
$ DEFINE NONLINEAR TRANSIENT SOLUTION PARAMETERS.
$
TSTEPNL,  1,  100,  100.
$
$ DEFINE TIME-DEPENDENT LOADING.
$
TLOAD1,  1,  100,  ,  ,  10
TABLED1, 10,
,    0.,  0.,  1000.,  1.,  2000.,  1.,  3000.,  0.,
, 4000.,  0.,  ENDT
$
$ VOLUME HEAT ADDITION.
$
QVOL,  100,  0.01,  ,  1
$
$ GEOMETRY DEFINITION (10" BAR DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  ,   0.,  0.,  0.,
GRID,  2,  ,   1.,  0.,  0.,
GRID,  3,  ,   2.,  0.,  0.,
GRID,  4,  ,   3.,  0.,  0.,
GRID,  5,  ,   4.,  0.,  0.,
GRID,  6,  ,   5.,  0.,  0.,
GRID,  7,  ,   6.,  0.,  0.,
GRID,  8,  ,   7.,  0.,  0.,
GRID,  9,  ,   8.,  0.,  0.,
GRID, 10,  ,   9.,  0.,  0.,
GRID, 11,  ,  10.,  0.,  0.,
$
$ CIRCULAR BAR MODELED WITH ROD ELEMENTS.
$
CROD,  1,  10,   1,   2
CROD,  2,  10,   2,   3
CROD,  3,  10,   3,   4
CROD,  4,  10,   4,   5
CROD,  5,  10,   5,   6
CROD,  6,  10,   6,   7
CROD,  7,  10,   7,   8
CROD,  8,  10,   8,   9
CROD,  9,  10,   9,  10
CROD, 10,  10,  10,  11
$
$ ROD ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" DIAMETER).
$
PROD,  10,  100,  0.7854
```

Listing 18-1. Model Input File for Bar Model with Transient Volumetric Heat Addition. (Continued)

```
$
$ ELEMENT CONDUCTIVITY, SPECIFIC HEAT, AND DENSITY (MA956).
$
MAT4,  100,  1.,  1.,  0.26
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
MATT4,  100,  20,  30
$
$ TEMPERATURE DEPENDENT CONDUCTIVITY DATA.
$
TABLEM2,  20,
,    70.,  1.466E-4,   200.,  1.620E-4,   400.,  1.852E-4,   600.,  2.083E-4,
,   800.,  2.315E-4,  1000.,  2.500E-4,  1200.,  2.739E-4,  1400.,  2.951E-4,
,  1600.,  3.164E-4,  1800.,  3.376E-4,  ENDT
$
$ TEMPERATURE DEPENDENT SPECIFIC HEAT DATA.
$
TABLEM2,  30,
,    70.,  0.1166,   200.,  0.1214,   400.,  0.1288,   600.,  0.1362,
,   800.,  0.1435,  1000.,  0.1509,  1200.,  0.1583,  1400.,  0.1657,
,  1600.,  0.1731,  1800.,  0.1805,  ENDT
$
$ ELEMENT POWER DENSITY.
$
MAT4,  200,  ,  ,  ,  ,  ,  1.
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
MATT4,  200,  ,  ,  ,  ,  ,  40
$
$ TEMPERATURE DEPENDENT HEAT GENERATION COEFFICIENT.
$
TABLEM2,  40,
,    70.,  0.993,   200.,  0.980,   400.,  0.960,   600.,  0.940,
,   800.,  0.920,  1000.,  0.900,  1200.,  0.880,  1400.,  0.860,
,  1600.,  0.840,  1800.,  0.820,  ENDT
$
$ INITIAL TEMPERATURE DISTRIBUTION.
$
TEMPD,  10,  0.
ENDDATA
```

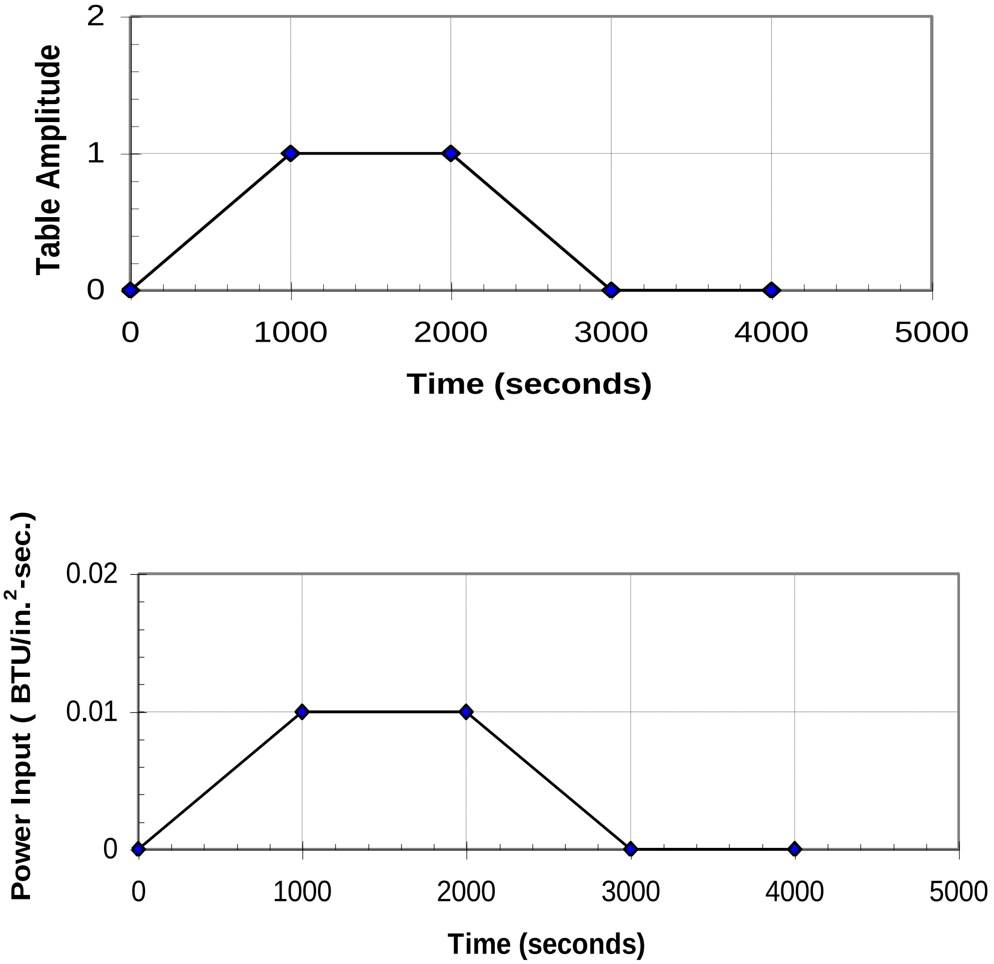
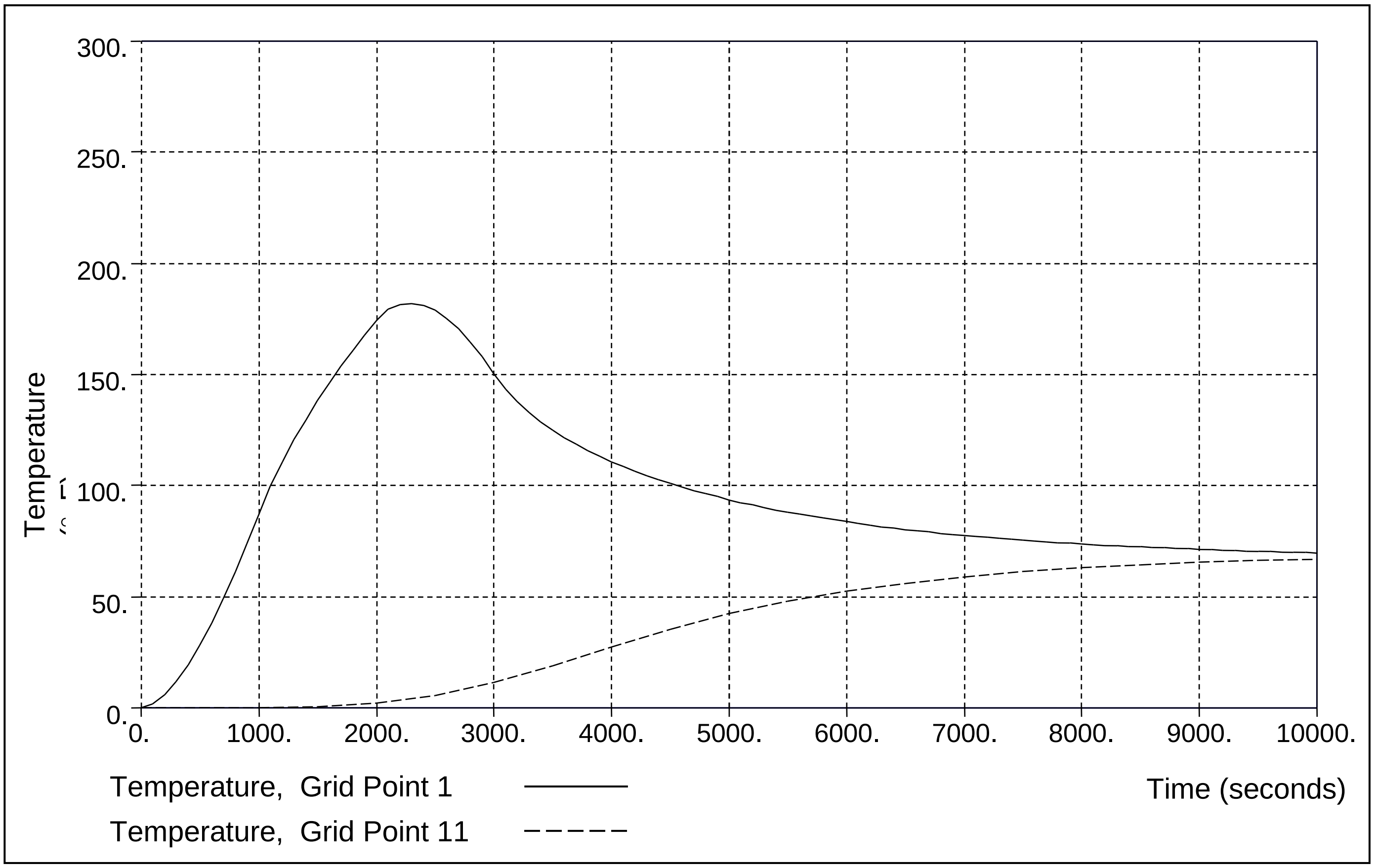


Figure 18-2. Time History from the TABLED1 Entry (top) and Resulting Applied Load (bottom).

Figure 18-3 gives the thermal response at each end of the bar. Since the heat load is applied only to the first element, the unloaded end stays at zero degrees until approximately 1500 seconds, while loaded end continues to increase in temperature reaching 177.3 °F at approximately 2200 seconds. At 3000 seconds the applied heat load returns to zero, but the unloaded end continues to increase in temperature reaching a peak value of 63.6 °F.

Figure 18-3. Bar End Temperatures Versus Time.



18.3.2 Nonlinear Convection

The next problem is an example of transient cool down with a nonlinear convection boundary condition. The circular bar in Figure 18-4 is initially at 500 °F with a constant convection boundary condition over its surface area maintained at 0 °F. Listing 18-2 contains the Model Input File.

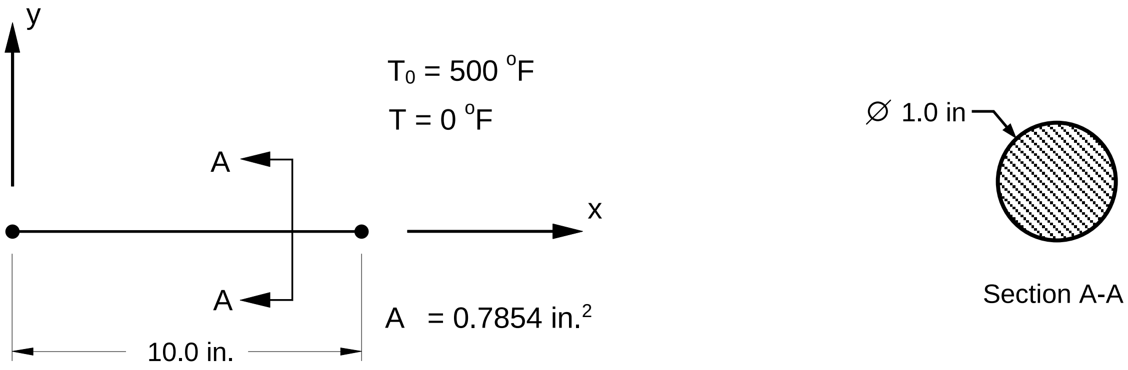


Figure 18-4. 1-Dimensional Bar Example Problem with Transient Cool Down Using a Constant Convection Boundary Condition.

There are several techniques for specifying temperature boundary conditions or ambient grid point temperatures for transient analysis. If the temperature is to remain constant throughout the analysis, a single point constraint (`SPC`) may be used to set the boundary condition just as in steady state analysis. The next section discusses transient boundary conditions where the ambient temperature varies with time.

Listing 18-2. Model Input File for Bar Model with Transient Cool Down Using a Constant Convection Boundary Condition.

```
$
$ NONLINEAR TRANSIENT HEAT TRANSFER SOLUTION.
$
SOL NONLINEAR TRANSIENT HEAT TRANSFER
$
ANALYSIS = HEAT
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = BAR WITH NONLINEAR TEMPERATURE DEPENDENT MATERIAL
$
THERMAL = ALL
SPCFORCE = ALL
FLUX = ALL
$
IC = 10
SUBCASE 1
 LABEL = TRANSIENT COOL DOWN WITH A CONSTANT CONVECTION BOUNDARY
 TSTEPNL = 1
 SPC = 1
$
BEGIN BULK
$
$ DEFINE NONLINEAR TRANSIENT SOLUTION PARAMETERS.
$
TSTEPNL,  1,  100,  100.
$
$ GEOMETRY DEFINITION (10" BAR DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  ,   0.,  0.,  0.,
GRID,  2,  ,   1.,  0.,  0.,
GRID,  3,  ,   2.,  0.,  0.,
GRID,  4,  ,   3.,  0.,  0.,
GRID,  5,  ,   4.,  0.,  0.,
GRID,  6,  ,   5.,  0.,  0.,
GRID,  7,  ,   6.,  0.,  0.,
GRID,  8,  ,   7.,  0.,  0.,
GRID,  9,  ,   8.,  0.,  0.,
GRID, 10,  ,   9.,  0.,  0.,
GRID, 11,  ,  10.,  0.,  0.,
GRID, 12,  ,   0.,  0.,  0.,
$
$ CIRCULAR BAR MODELED WITH ROD ELEMENTS.
$
CROD,  1,  10,   1,   2
CROD,  2,  10,   2,   3
CROD,  3,  10,   3,   4
CROD,  4,  10,   4,   5
CROD,  5,  10,   5,   6
CROD,  6,  10,   6,   7
CROD,  7,  10,   7,   8
CROD,  8,  10,   8,   9
CROD,  9,  10,   9,  10
CROD, 10,  10,  10,  11
$
$ ROD ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" DIAMETER).
$
PROD,  10,  100,  0.7854
$
$ ELEMENT CONDUCTIVITY, SPECIFIC HEAT, AND DENSITY (MA956).
$
MAT4,  100,  1.,  1.,  0.26
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
MATT4,  100,  10,  20
```

Listing 18-2. Model Input File for Bar Model with Transient Cool Down Using a Constant Convection Boundary Condition. (Continued)

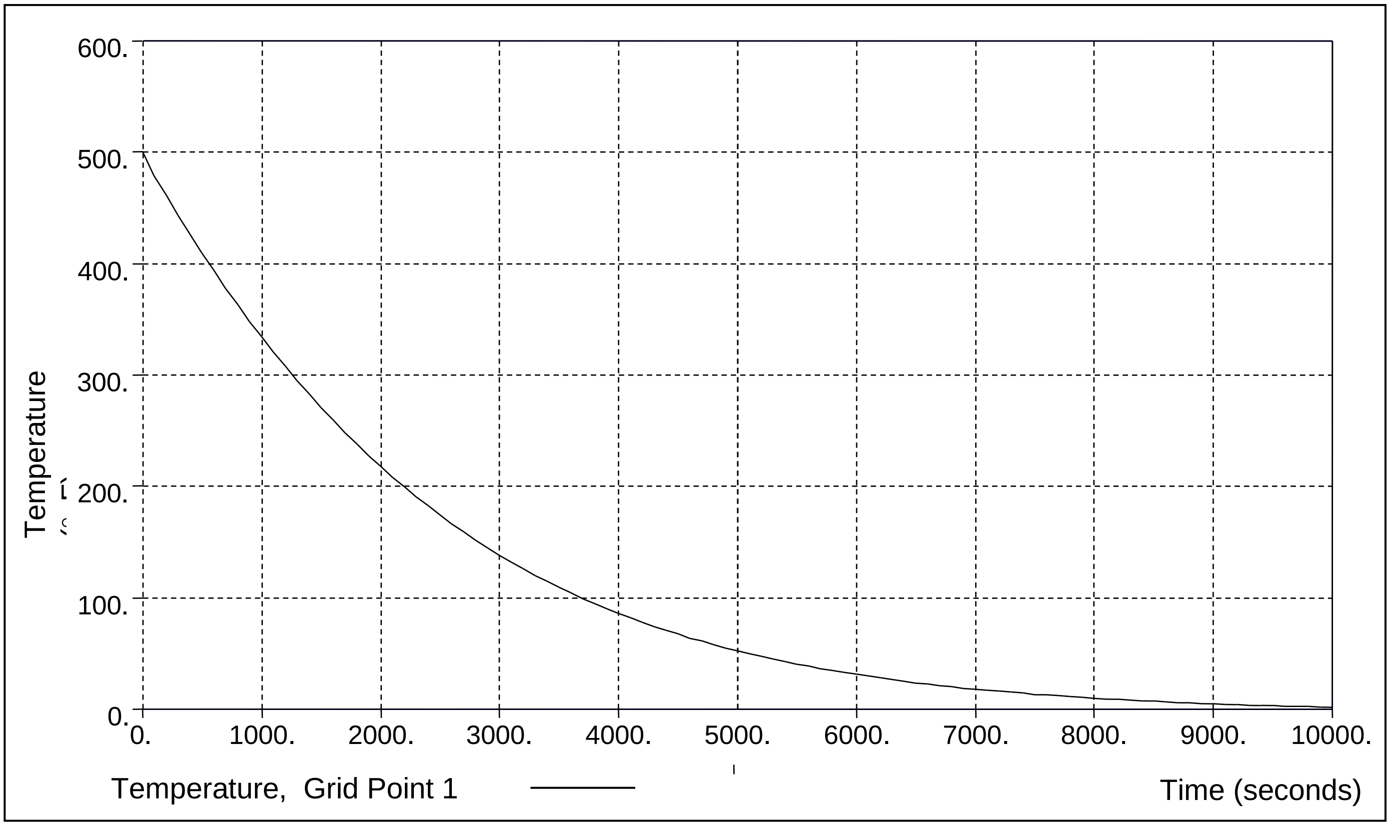
```
$
$ TEMPERATURE DEPENDENT CONDUCTIVITY DATA.
$
TABLEM2,  10,
,    70.,  1.466E-4,   200.,  1.620E-4,   400.,  1.852E-4,   600.,  2.083E-4,
,   800.,  2.315E-4,  1000.,  2.508E-4,  1200.,  2.739E-4,  1400.,  2.951E-4,
,  1600.,  3.164E-4,  1800.,  3.376E-4,  ENDT
$
$ TEMPERATURE DEPENDENT SPECIFIC HEAT DATA.
$
TABLEM2,  20,
,    70.,  0.1166,   200.,  0.1214,   400.,  0.1288,   600.,  0.1362,
,   800.,  0.1435,  1000.,  0.1509,  1200.,  0.1583,  1400.,  0.1657,
,  1600.,  0.1731,  1800.,  0.1805,  ENDT
$
$ CONVECTION BOUNDARY DEFINITION.
$
CHBDYP,  11,  20,  LINE,  ,  ,   1,   2,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  12,  20,  LINE,  ,  ,   2,   3,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  13,  20,  LINE,  ,  ,   3,   4,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  14,  20,  LINE,  ,  ,   4,   5,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  15,  20,  LINE,  ,  ,   5,   6,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  16,  20,  LINE,  ,  ,   6,   7,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  17,  20,  LINE,  ,  ,   7,   8,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  18,  20,  LINE,  ,  ,   8,   9,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  19,  20,  LINE,  ,  ,   9,  10,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  20,  20,  LINE,  ,  ,  10,  11,
  ,  ,  ,  ,  ,  0.,  1.,  0.
$
$ AREA FACTOR TO DEFINE ROD SURFACE AREA
$ AF = PI*DIAMETER = 3.142, AREA = AF*LENGTH
$
PHBDY,  20,  3.142
$
$ CONVECTION LOAD.
$
CONV,  11,  30,  ,  ,  12,  12
CONV,  12,  30,  ,  ,  12,  12
CONV,  13,  30,  ,  ,  12,  12
CONV,  14,  30,  ,  ,  12,  12
CONV,  15,  30,  ,  ,  12,  12
CONV,  16,  30,  ,  ,  12,  12
CONV,  17,  30,  ,  ,  12,  12
CONV,  18,  30,  ,  ,  12,  12
CONV,  19,  30,  ,  ,  12,  12
CONV,  20,  30,  ,  ,  12,  12
$
$ CONVECTION PROPERTY REFERENCE.
$
PCONV,  30,  200
$
$ FREE CONVECTION HEAT TRANSFER COEFFICIENT (AIR).
$
MAT4,  200,  ,  ,  ,  1.
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
MATT4,  200,  ,  ,  ,  30
```

Listing 18-2. Model Input File for Bar Model with Transient Cool Down Using a Constant Convection Boundary Condition. (Continued)

```
$
$ TEMPERATURE DEPENDENT FREE CONVECTION HEAT TRANSFER COEFFICIENT.
$
TABLEM2,  30,
,   600.,  3.877E-6,   800.,  3.946E-6,  1000.,  3.993E-6,  1200.,  4.015E-6,
,  1400.,  4.039E-6,  1600.,  4.068E-6,  1800.,  4.142E-6,  2000.,  4.261E-6,
,  2200.,  4.380E-6,  2400.,  4.501E-6,  ENDT
$
$ AMBIENT TEMPERATURE DEFINITION.
$
SPC,  1,  12,  1,  0.
$
$ INITIAL TEMPERATURE DISTRIBUTION.
$
TEMP,  10,  12,  0.
TEMPD,  10,  500.
ENDDATA
```

Figure 18-5 gives the thermal response for the bar. As expected the surface temperature approaches the ambient temperature of 0 °F.

Figure 18-5. Bar Temperature Versus Time.



The next problem is another example of transient cool down with a nonlinear convection boundary condition. The circular bar in Figure 18-6 is initially at 500 ^{o}F with a time-varying convection boundary condition over its surface area varied as shown in Figure 18-7. Listing 18-3 contains the Model Input File.

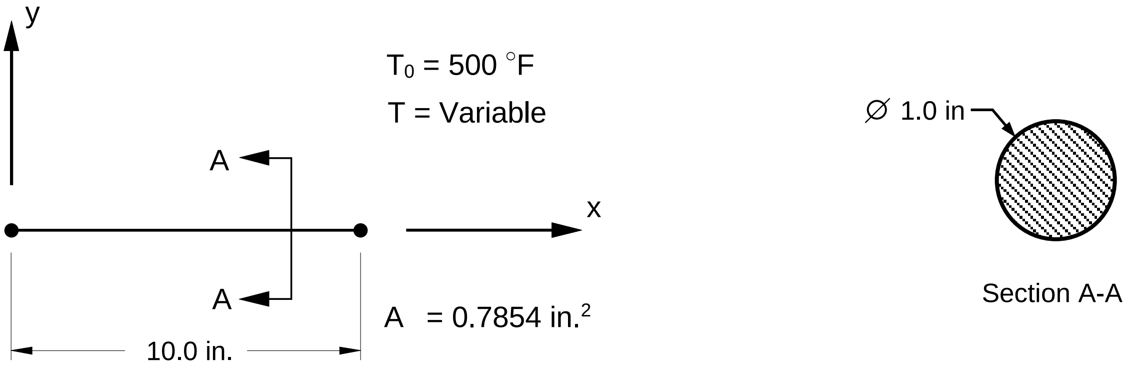


Figure 18-6. 1-Dimensional Bar Example Problem with Transient Cool Down Using a Time-Varying Convection Boundary Condition.

In the previous example the ambient temperature was constant. If the ambient temperature varies with time, a `TEMPBC` Bulk Data entry is used. The `TEMPBC` is treated in the same way as a thermal load for transient analysis with the `TEMPBC` set identification number specified directly on the `TLOADi` Bulk Data entry.

Listing 18-3. Model Input File for Bar Model with Transient Cool Down Using a Time-Varying Convection Boundary Condition.

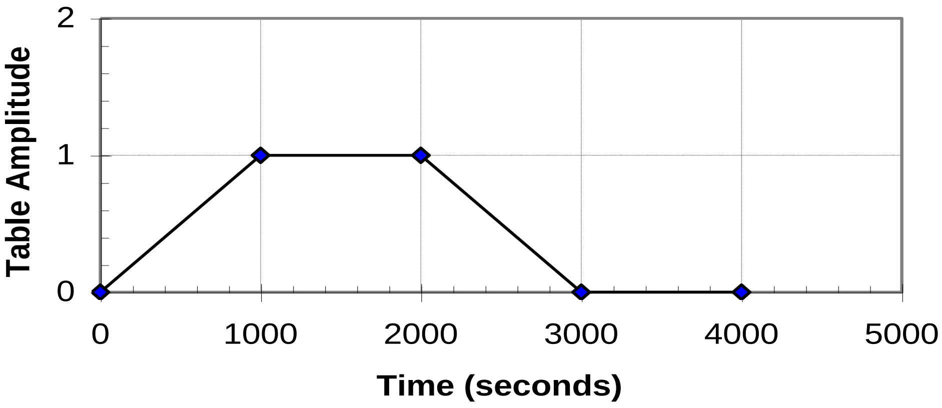
```
$
$ NONLINEAR TRANSIENT HEAT TRANSFER SOLUTION.
$
SOL NONLINEAR TRANSIENT HEAT TRANSFER
$
ANALYSIS = HEAT
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = BAR WITH NONLINEAR TEMPERATURE DEPENDENT MATERIAL
$
THERMAL = ALL
FLUX = ALL
$
IC = 10
SUBCASE 1
 LABEL = TRANSIENT COOL DOWN WITH A TRANSIENT CONVECTION BOUNDARY
 DLOAD = 1
 TSTEPNL = 1
$
BEGIN BULK
$
$ DEFINE NONLINEAR TRANSIENT SOLUTION PARAMETERS.
$
TSTEPNL,  1,  100,  100.
$
$ DEFINE TIME-DEPENDENT LOADING.
$
TLOAD1,  1,  100,  ,  ,  10
TABLED1, 10,
,    0.,  0.,  1000.,  1.,  2000.,  1.,  3000.,  0.,
, 4000.,  0.,  ENDT
$
$ TRANSIENT AMBIENT TEMPERATURE DEFINITION.
$
TEMPBC,  100,  TRAN,  500.,  12
$
$ GEOMETRY DEFINITION (10" BAR DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  ,   0.,  0.,  0.,
GRID,  2,  ,   1.,  0.,  0.,
GRID,  3,  ,   2.,  0.,  0.,
GRID,  4,  ,   3.,  0.,  0.,
GRID,  5,  ,   4.,  0.,  0.,
GRID,  6,  ,   5.,  0.,  0.,
GRID,  7,  ,   6.,  0.,  0.,
GRID,  8,  ,   7.,  0.,  0.,
GRID,  9,  ,   8.,  0.,  0.,
GRID, 10,  ,   9.,  0.,  0.,
GRID, 11,  ,  10.,  0.,  0.,
GRID, 12,  ,   0.,  0.,  0.,
$
$ CIRCULAR BAR MODELED WITH ROD ELEMENTS.
$
CROD,  1,  10,   1,   2
CROD,  2,  10,   2,   3
CROD,  3,  10,   3,   4
CROD,  4,  10,   4,   5
CROD,  5,  10,   5,   6
CROD,  6,  10,   6,   7
CROD,  7,  10,   7,   8
CROD,  8,  10,   8,   9
CROD,  9,  10,   9,  10
CROD, 10,  10,  10,  11
$
$ ROD ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" DIAMETER).
$
PROD,  10,  100,  0.7854
```

Listing 18-3. Model Input File for Bar Model with Transient Cool Down Using a Time-Varying Convection Boundary Condition. (Continued)

```
$
$ ELEMENT CONDUCTIVITY, SPECIFIC HEAT, AND DENSITY (MA956).
$
MAT4,  100,  1.,  1.,  0.26
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
MATT4,  100,  20,  30
$
$ TEMPERATURE DEPENDENT CONDUCTIVITY DATA.
$
TABLEM2,  20,
,    70.,  1.466E-4,   200.,  1.620E-4,   400.,  1.852E-4,   600.,  2.083E-4,
,   800.,  2.315E-4,  1000.,  2.508E-4,  1200.,  2.739E-4,  1400.,  2.951E-4,
,  1600.,  3.164E-4,  1800.,  3.376E-4,  ENDT
$
$ TEMPERATURE DEPENDENT SPECIFIC HEAT DATA.
$
TABLEM2,  30,
,    70.,  0.1166,   200.,  0.1214,   400.,  0.1288,   600.,  0.1362,
,   800.,  0.1435,  1000.,  0.1509,  1200.,  0.1583,  1400.,  0.1657,
,  1600.,  0.1731,  1800.,  0.1805,  ENDT
$
$ CONVECTION BOUNDARY DEFINITION.
$
CHBDYP,  11,  20,  LINE,  ,  ,   1,   2,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  12,  20,  LINE,  ,  ,   2,   3,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  13,  20,  LINE,  ,  ,   3,   4,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  14,  20,  LINE,  ,  ,   4,   5,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  15,  20,  LINE,  ,  ,   5,   6,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  16,  20,  LINE,  ,  ,   6,   7,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  17,  20,  LINE,  ,  ,   7,   8,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  18,  20,  LINE,  ,  ,   8,   9,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  19,  20,  LINE,  ,  ,   9,  10,
  ,  ,  ,  ,  ,  0.,  1.,  0.
CHBDYP,  20,  20,  LINE,  ,  ,  10,  11,
  ,  ,  ,  ,  ,  0.,  1.,  0.
$
$ AREA FACTOR TO DEFINE ROD SURFACE AREA
$ AF = PI*DIAMETER = 3.142, AREA = AF*LENGTH
$
PHBDY,  20,  3.142
$
$ CONVECTION LOAD.
$
CONV,  11,  30,  ,  ,  12,  12
CONV,  12,  30,  ,  ,  12,  12
CONV,  13,  30,  ,  ,  12,  12
CONV,  14,  30,  ,  ,  12,  12
CONV,  15,  30,  ,  ,  12,  12
CONV,  16,  30,  ,  ,  12,  12
CONV,  17,  30,  ,  ,  12,  12
CONV,  18,  30,  ,  ,  12,  12
CONV,  19,  30,  ,  ,  12,  12
CONV,  20,  30,  ,  ,  12,  12
$
$ CONVECTION PROPERTY REFERENCE.
$
PCONV,  30,  200
```

Listing 18-3. Model Input File for Bar Model with Transient Cool Down Using a Time-Varying Convection Boundary Condition. (Continued)

```
$
$ FREE CONVECTION HEAT TRANSFER COEFFICIENT (AIR).
$
MAT4,  200,  ,  ,  ,  1.
$
$ NONLINEAR TEMPERATURE DEPENDENT PROPERTIES.
$
MATT4,  200,  ,  ,  ,  40
$
$ TEMPERATURE DEPENDENT FREE CONVECTION HEAT TRANSFER COEFFICIENT.
$
TABLEM2,  40,
,   600.,  3.877E-6,   800.,  3.946E-6,  1000.,  3.993E-6,  1200.,  4.015E-6,
,  1400.,  4.039E-6,  1600.,  4.068E-6,  1800.,  4.142E-6,  2000.,  4.261E-6,
,  2200.,  4.380E-6,  2400.,  4.501E-6,  ENDT
$
$ INITIAL TEMPERATURE DISTRIBUTION.
$
TEMPD,  10,  0.
ENDDATA
```



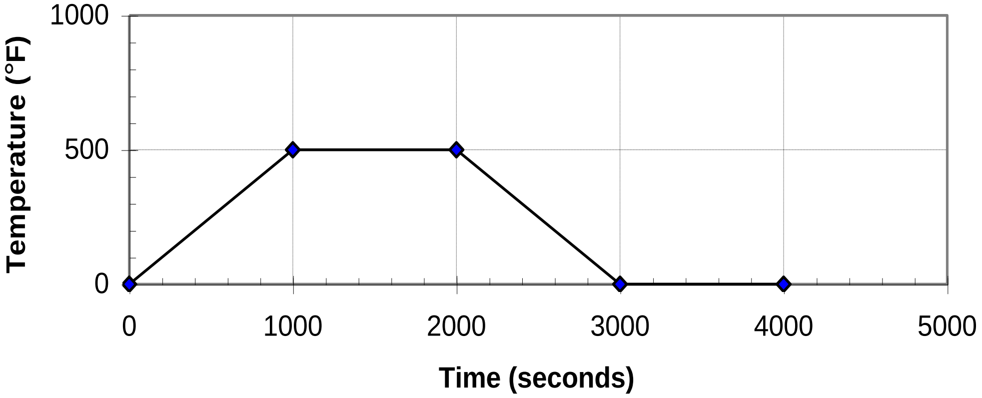
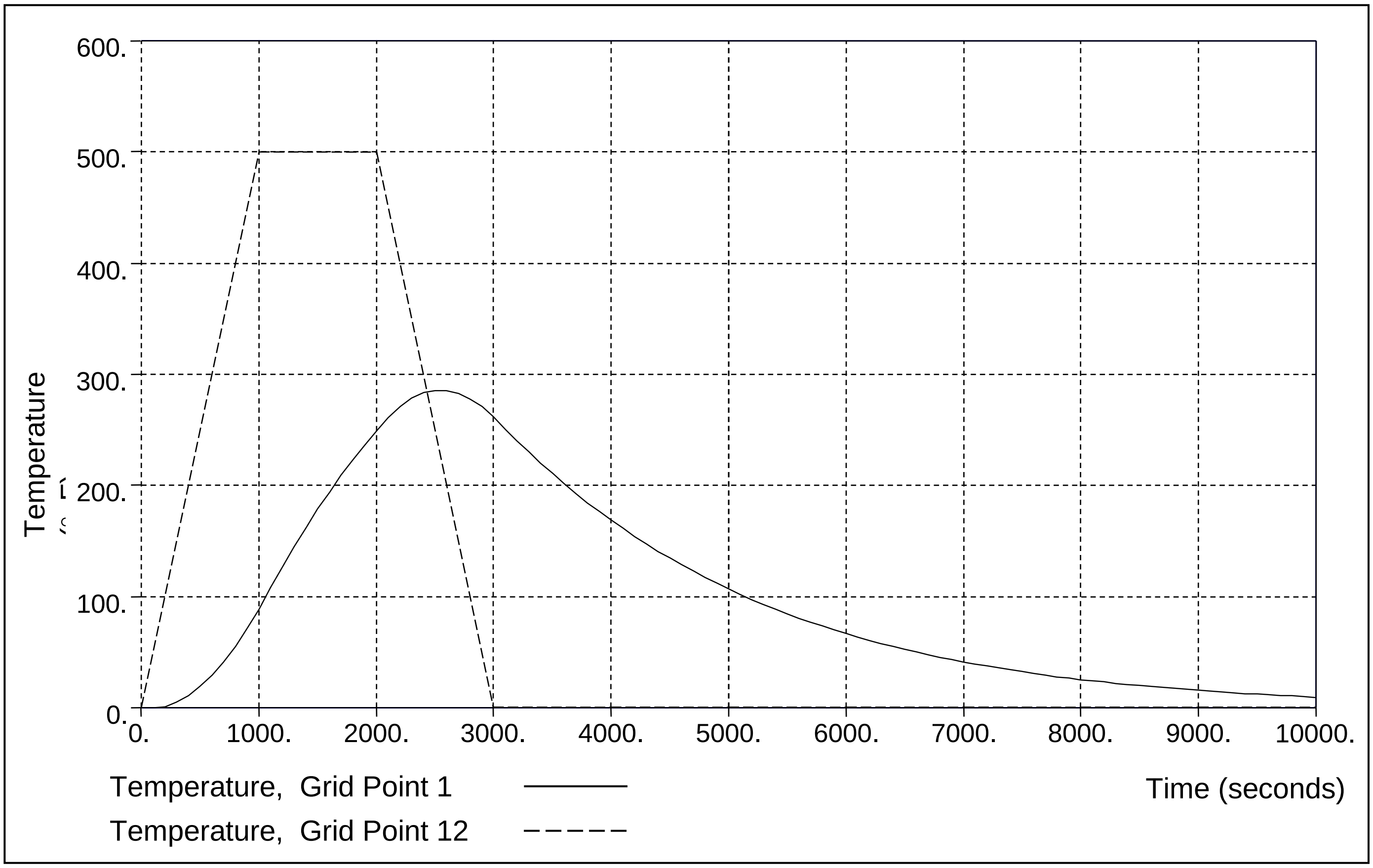


Figure 18-7. Time History from the `TABLED1` Entry (top) and Resulting Ambient Temperature (bottom).

Figure 18-8 gives the thermal response for both the bar surface and the ambient grid point. As expected the surface temperature approaches the ambient temperature of 0 °F.

Figure 18-8. Bar Temperature Versus Time.



19. NONLINEAR PRESTRESS MODAL ANALYSIS

19.1 Introduction

Nonlinear prestress modal analysis is similar to linear prestress modal except that a nonlinear solution is used to form the tangent stiffness matrix. Autodesk Inventor Nastran then determines prestress natural frequency by solving the eigenvalue problem:

$$\left| [K_t] + \lambda[M] \right| [\phi] = 0$$

$$\lambda_i = \omega_i^2$$

$$f_i = \frac{\omega_i}{2\pi}$$

where,

$[K_t]$	is the global tangent stiffness matrix
$[M]$	is the global mass matrix
λ_i	are the eigenvalues that yield the natural frequencies
ϕ_i	are the eigenvectors that represent the natural mode shapes
ω_i	are the circular frequencies (radians per second)
f_i	are the cyclic frequencies (hertz)

19.2 How to Setup a Model Input File for Nonlinear Prestress Modal Analysis

In Autodesk Inventor Nastran you can perform nonlinear prestress modal analysis by setting `SOLUTION = NONLINEAR PRESTRESS MODAL` in the Model Initialization File or by specifying `SOL 185` or `SOL NONLINEAR PRESTRESS MODAL` above the Case Control Section in the Model Input File. Multiple subcases are allowed to define the nonlinear prestress state.

19.3 Interpreting Results

The cantilever beam in Figure 19-1 is subjected to a shear load at its free end. The beam deflects normally until hitting a rigid support, which is modeled using a gap element (Figure 15-26). After contacting the support, the beam continues to deflect resulting in a reaction force in the gap element. It is desired to find the lowest natural frequency of the beam when fully loaded in its deflected state. Listing 19-1 contains the Model Input File and Listing 19-2 shows the extracted frequencies from the Model Results Output File.

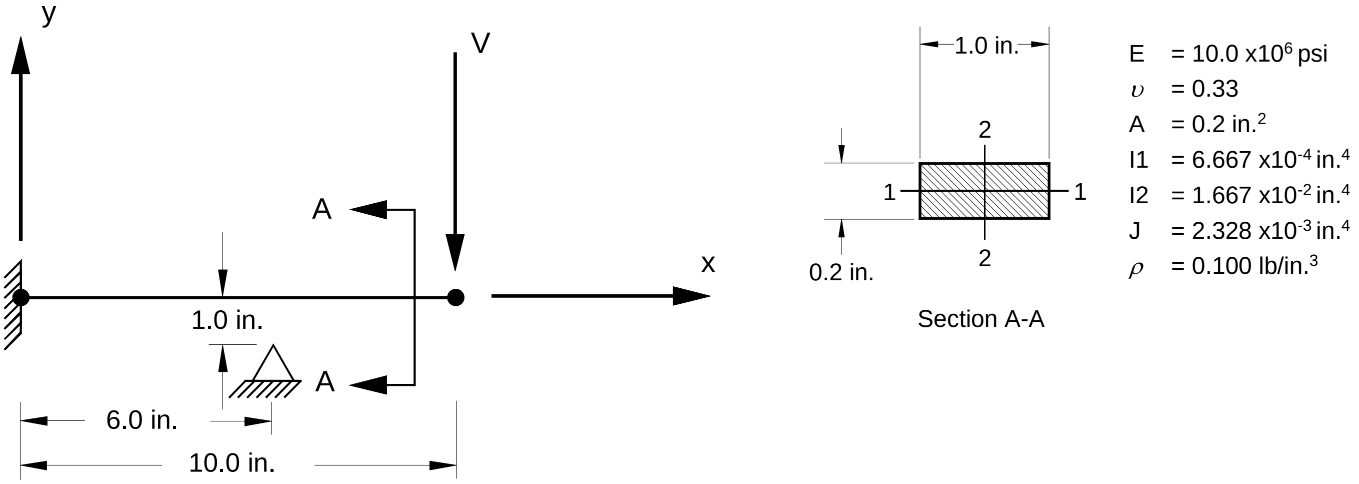


Figure 19-1. 2-D Cantilever Beam Example Problem with Contact.

Listing 19-1. Model Input File for the Cantilever Beam Problem with Contact.

```
$
$ NONLINEAR PRESTRESS MODAL SOLUTION.
$
SOL NONLINEAR PRESTRESS MODAL
$
TITLE = INSTALLATION TEST CASE
SUBTITLE= VIBRATION OF A SHEAR LOADED CANTILEVER BEAM IN CONTACT
$
DISPLACEMENT = ALL
$
SUBCASE 1
 LABEL = POINT LOAD AT FREE END (SHEAR)
 LOAD = 1
 NLPARM = 1
 SPC = 1
SUBCASE 2
 LABEL = MODAL
 METHOD = 1
 SPC = 1
$
BEGIN BULK
$
$ CONVERSION FACTOR FOR WEIGHT DENSITY TO MASS DENSITY
$ MASS = (1/g)*WEIGHT, G=32.2FT/SEC2, WTMASS = 1/(32.2*12) = 0.002588
$
PARAM, WTMASS, 0.002588
$
$ REQUEST DIAGONAL MASS MATRIX FORMULATION.
$
PARAM, COUPMASS, OFF
$
$ DEFINE EIGENVALUE EXTRACTION PARAMETERS.
$
EIGRL,  1,  ,  ,  5,  ,  ,
$
$ DEFINE NONLINEAR SOLUTION PARAMETERS.
$
NLPARM,  1,  10,  ,  ,  ,  ,  ,  YES
$
$ GEOMETRY DEFINITION (10" BEAM DIVIDED INTO 10 ELEMENTS).
$
GRID,  1,  0,   0.,  0.,  0.,  0
GRID,  2,  0,   1.,  0.,  0.,  0
GRID,  3,  0,   2.,  0.,  0.,  0
GRID,  4,  0,   3.,  0.,  0.,  0
GRID,  5,  0,   4.,  0.,  0.,  0
GRID,  6,  0,   5.,  0.,  0.,  0
GRID,  7,  0,   6.,  0.,  0.,  0
GRID,  8,  0,   7.,  0.,  0.,  0
GRID,  9,  0,   8.,  0.,  0.,  0
GRID, 10,  0,   9.,  0.,  0.,  0
GRID, 11,  0,  10.,  0.,  0.,  0
GRID, 12,  0,   6., -1.,  0.,  0
$
$ BEAM MODELED WITH BAR ELEMENTS.
$
CBAR,  1,  10,   1,   2,  0.,  0.,  1.
CBAR,  2,  10,   2,   3,  0.,  0.,  1.
CBAR,  3,  10,   3,   4,  0.,  0.,  1.
CBAR,  4,  10,   4,   5,  0.,  0.,  1.
CBAR,  5,  10,   5,   6,  0.,  0.,  1.
CBAR,  6,  10,   6,   7,  0.,  0.,  1.
CBAR,  7,  10,   7,   8,  0.,  0.,  1.
CBAR,  8,  10,   8,   9,  0.,  0.,  1.
CBAR,  9,  10,   9,  10,  0.,  0.,  1.
CBAR, 10,  10,  10,  11,  0.,  0.,  1.
$
$ GAP ELEMENT.
$
CGAP, 11,  20,   7,  12,  1.,  0.,  0.
```

Listing 19-1. Model Input File for the Cantilever Beam Problem with Contact. (Continued)

```
$
$ BAR ELEMENT MATERIAL AND SECTION PROPERTIES (1.0" X 0.2" CROSS-SECTION).
$
PBAR,  10,  100,  0.2,  1.667E-2,  6.667E-4,  2.328E-3,
,  -0.5,  0.1,  0.5,  0.1,  -0.5,  -0.1,  0.5,  -0.1
$
$ LINEAR ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ GAP ELEMENT PROPERTIES.
$
PGAP,  20,  1.,  0.,  1.E+7
$
$ FIXED AT ONE END, MOVEMENT CONSTRAINED TO X-Y PLANE ONLY.
$
SPC1,  1,  123456,   1
SPC1,  1,     345,   2,  THRU,  11
SPC1,  1,  123456,  12
$
$ POINT LOAD AT FREE END (SHEAR).
$
FORCE,  1,  11,  0,  1.5E+2,  0., -1.,  0.
ENDDATA
```

If the beam in Figure 19-1 was unloaded, it would have natural frequency of 240.0 Hz. The deflected beam has a frequency of 247.6 Hz.

Listing 19-2. Extracted Eigenvectors for the Cantilever Beam Problem with Contact.

```
MODAL                                                        SUBCASE 2

                                            R E A L   E I G E N V A L U E S

     MODE         EIGENVALUE       RADIANS          CYCLES        GENERALIZED     GENERALIZED     ORTHOGONALITY       ERROR
    NUMBER                                                            MASS          STIFFNESS          LOSS           MEASURE
       1        2.420853E+06    1.555909E+03    2.476306E+02    1.000000E+00    2.420853E+06    0.000000E+00    1.371168E-14
       2        3.438160E+07    5.863582E+03    9.332181E+02    1.000000E+00    3.438160E+07    2.263169E-14    5.030191E-11
       3        1.395707E+08    1.181400E+04    1.880257E+03    1.000000E+00    1.395707E+08    5.438358E-15    4.071945E-13
       4        3.174987E+08    1.781849E+04    2.835901E+03    1.000000E+00    3.174987E+08    1.383442E-16    6.524586E-10
       5        9.634014E+08    3.103871E+04    4.939964E+03    1.000000E+00    9.634014E+08    1.350482E-15    1.559315E-07
```

20. NONLINEAR PRESTRESS TRANSIENT RESPONSE ANALYSIS

20.1 Introduction

Nonlinear prestress transient response analysis is similar to linear prestress transient response except that a nonlinear solution is used to form the tangent stiffness matrix.

20.2 How to Setup a Model Input File for Nonlinear Prestress Transient Response Analysis

In Autodesk Inventor Nastran you can perform nonlinear prestress transient response analysis by setting `SOLUTION = NONLINEAR PRESTRESS TRANSIENT RESPONSE` in the Model Initialization File or by specifying `SOL 187` or `SOL NONLINEAR PRESTRESS TRANSIENT RESPONSE` above the Case Control Section in the Model Input File. Multiple subcases are allowed to define the nonlinear prestress state. Like linear prestress transient response, a modal transient response solution is required. Direct transient response is not supported.

21. NONLINEAR PRESTRESS FREQUENCY RESPONSE ANALYSIS

21.1 Introduction

Nonlinear prestress frequency response analysis is similar to linear prestress frequency response except that a nonlinear solution is used to form the tangent stiffness matrix.

21.2 How to Setup a Model Input File for Nonlinear Prestress Frequency Response Analysis

In Autodesk Inventor Nastran you can perform nonlinear prestress frequency response analysis by setting `SOLUTION = NONLINEAR PRESTRESS FREQUENCY RESPONSE` in the Model Initialization File or by specifying `SOL 186` or `SOL NONLINEAR PRESTRESS FREQUENCY RESPONSE` above the Case Control Section in the Model Input File. Multiple subcases are allowed to define the nonlinear prestress state. Like linear prestress frequency response, a modal frequency response solution is required. Direct frequency response is not supported.

22. NONLINEAR PRESTRESS COMPLEX EIGENVALUE ANALYSIS

22.1 Introduction

Nonlinear prestress complex eigenvalue analysis is similar to linear prestress complex eigenvalue analysis except that a nonlinear solution is used to form the tangent stiffness matrix.

22.2 How to Setup a Model Input File for Nonlinear Prestress Complex Eigenvalue Analysis

In Autodesk Inventor Nastran you can perform nonlinear prestress complex eigenvalue analysis by setting `SOLUTION = NONLINEAR PRESTRESS COMPLEX EIGENVALUE` in the Model Initialization File or by specifying `SOL 189` or `SOL NONLINEAR PRESTRESS COMPLEX EIGENVALUE` above the Case Control Section in the Model Input File. Multiple subcases can be specified, each requesting a different output set. Each subcase must also reference an `EIGC` Bulk Data entry via the `CMETHOD` Case Control command.

23. NONLINEAR BUCKLING ANALYSIS

23.1 Introduction

Nonlinear buckling analysis is similar to linear buckling except that a nonlinear solution is used to form the tangent stiffness matrix.

23.2 How to Setup a Model Input File for Nonlinear Buckling Analysis

In Autodesk Inventor Nastran you can perform nonlinear buckling analysis by setting `SOLUTION = NONLINEAR BUCKLING` in the Model Initialization File or by specifying `SOL 180` or `SOL NONLINEAR BUCKLING` above the Case Control Section in the Model Input File. Multiple subcases are allowed to define the nonlinear prestress state.

24. SPECIAL TOPICS

24.1 Stress Coordinate Systems

Element and grid point results in Autodesk Inventor Nastran can be output in any coordinate system through the use of the `SURFACE` and `VOLUME` Case Control commands. The `SURFACE` command is used to align shell element normals and define the shell element output coordinate system. The `VOLUME` command is used to define the solid element output coordinate system. Shell elements must be referenced on a `SURFACE` and solid elements must be referenced in a `VOLUME` for results to be output. Element results can be output in the element, basic, material, or a user specified coordinate system. Grid point results can be output in the grid, basic, material, or a user specified coordinate system. The default for element results output is the element coordinate system. The default for grid point results is the grid or global coordinate system. The global coordinate system is the collection of all displacement coordinate systems defined in field 7 of the `GRID` Bulk Data entry. The basic coordinate system is the default Cartesian coordinate system.

The following is an example of how to use the `SURFACE` command. An element patch test case is shown in Figure 24-1. It is desired to find the element normal-x stress and compare it to the theoretical value. The problem is that because the elements are skewed, the normal-x stress in the element coordinate system is not parallel to the basic or model coordinate system. In fact, two of the four elements have element x-directions that are closer to being parallel to the model y-axis. This can actually happen in an irregular or reflected (mirrored) mesh and has been done purposely here to emphasize the importance of stress coordinate systems. Listing 24-1 contains the Model Input File.

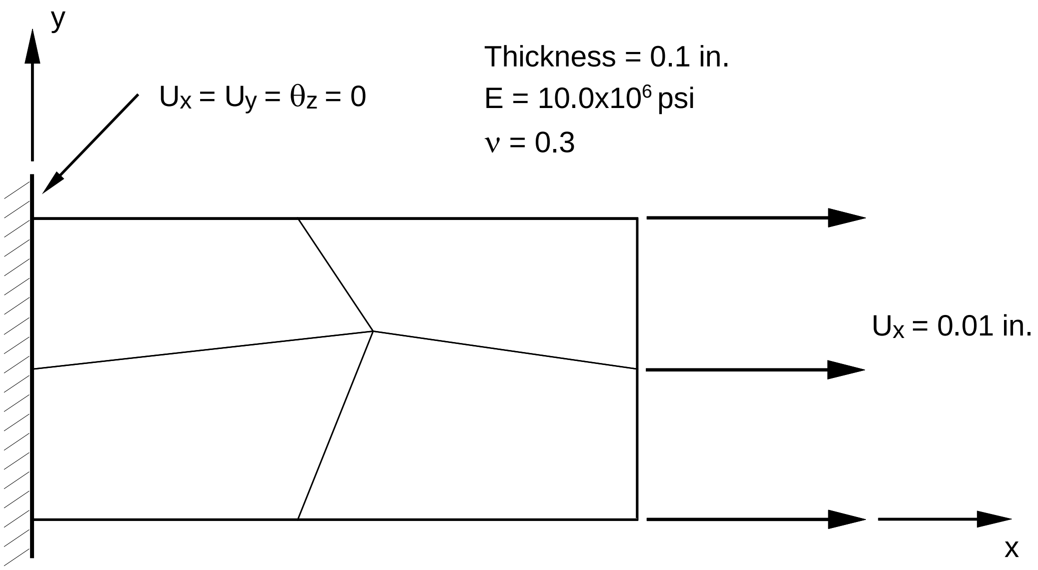


Figure 24-1. Element Patch Test Example.

Listing 24-1. Element Patch Test Model Input File.

```
$
$ STATIC SOLUTION.
$
SOL STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = ELEMENT PATCH TEST -QUAD4 ELEMENTS -2X2 MESH
$
DISPLACEMENT = ALL
ELSTRESS = ALL
GPSTRESS = ALL
$
SPC = 1
SUBCASE 1
 LABEL = 0.01 IN ENFORCED DISPLACEMENT IN X-DIRECTION
 LOAD = 1
$
$ ELEMENT AND GRID POINT STRESS COORDINATE SYSTEM (BASIC).
$
SET 1 = ALL
SURFACE 1, SET 1, SYSTEM BASIC, AXIS X, NORMAL Z
$
BEGIN BULK
$
$ GEOMETRY DEFINITION (4" X 2" RECTANGULAR FLAT PLATE WITH A 2 X 2 MESH).
$
GRID,  1,  0,  0.,    0.,    0.
GRID,  2,  0,  1.75,  0.,    0.
GRID,  3,  0,  4.,    0.,    0.
GRID,  4,  0,  0.,    1.,    0.
GRID,  5,  0,  2.25,  1.25,  0.
GRID,  6,  0,  4.,    1.,    0.
GRID,  7,  0,  0.,    2.,    0.
GRID,  8,  0,  1.75,  2.,    0.
GRID,  9,  0,  4.,    2.,    0.
$
$ FLAT PLATE MODELED WITH SHELL ELEMENTS.
$
CQUAD4,  1,  10,  2,  5,  4,  1
CQUAD4,  2,  10,  2,  3,  6,  5
CQUAD4,  3,  10,  4,  5,  8,  7
CQUAD4,  4,  10,  6,  9,  8,  5
$
$ ELEMENT MATERIAL AND THICKNESS (0.1").
$
PSHELL,  10,  100,  0.1,  100,  ,  100
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.000+7,  ,  0.30000,  0.1
$
$ FIXED AT BOTH ENDS -ONE END FREE TO TRANSLATE IN X-DIR.
$
SPC1,  1,  123456,  7
SPC1,  1,    1345,  1,  4
SPC1,  1,    2345,  9
SPC1,  1,     345,  3,  6
$
$ ENFORCED DISPLACEMENT ON OTHER END (X-DIRECTION).
$
SPCD,  1,  3,  1,  0.01
SPCD,  1,  6,  1,  0.01
SPCD,  1,  9,  1,  0.01
ENDDATA
```

The theoretical stress can be found using:

$$\sigma = \frac{\delta E}{\ell}$$

Table 24-1 demonstrates the importance of the `SURFACE` command. While most users are only concerned with stress and strain invariants, such as von Mises, many applications require knowing the stress or strain in a particular direction. An example of this would be predicting strain to compare with test data recorded from a strain gage oriented in a particular direction.

Table 24-1. Comparison of Theoretical Versus Predicted Element Normal-X Stress.

Element ID	Theoretical (psi)	Normal-X Stress w/out **`SURFACE`** (psi)	Normal-X Stress with **`SURFACE`** (psi)
1	25000.	9.	25000.
2	25000.	24746.	25000.
3	25000.	24757.	25000.
4	25000.	11.	25000.

24.2 Quad Element Formulation Options

The quad elements in Autodesk Inventor Nastran are designed to give accurate results regardless of shape (skew, taper, aspect ratio, etc.). The quad element has from 20 to 54 total degrees of freedom and the option for either 5 or 6 degrees of freedom per node. The 6th degree of freedom or vertex drill degree of freedom (`CQUADR` or `PARAM`, `QUADRNODE`, `ON`) should be specified when:

- Curved shells are modeled.
- Beam, bar, or rod elements are used.
- Rigid elements are used with all 6 degrees of freedom made dependent.

The following guidelines should be adhered to when using the `CQUADR` element:

- At least one grid point in the model must have the drill degree of freedom constrained, which is not the case for the bending degrees of freedom.
- Do not over constrain the drill degree of freedom.
- Use the `CTRIAR` element (`PARAM`, `TRIRNODE`, `ON`) when using the `CQUADR` element.

The element formulation option also includes an internal node (`PARAM`, `QUADINODE`, `ON`). This option may be used with either the `CQUAD4` or the `CQUADR` element and generally improves accuracy with a slight penalty in performance. Table 24-2 shows the various element formulations and the degrees of freedom for each.

Table 24-2. Quad Element Formulation Options.

Element Type	Geometry Processor Parameter Settings	Nodal DOF	Element DOF		
		External [3]	Internal [2]	External [3]	Total
`CQUAD4`	`QUADINODE = OFF`	5	0	20	20
`CQUAD4`	`QUADINODE = ON` [1]	5	5	20	25
`CQUADR`	`QUADINODE = OFF`	6	24	24	48
`CQUADR`	`QUADINODE = ON` [1]	6	30	24	54

Notes:

1. Default setting.
2. Internal degrees of freedom are generated by static condensation of internal element nodes.
3. External degrees of freedom are generated from corner and mid-side nodes, which have grid points associated with them.

Generally, the more degrees of freedom an element has, the more accurate it will be especially when it is irregularly shaped.

24.3 Hex Element Formulation Options

The HEX element has from 24 to 63 total degrees of freedom. The `HEXEGRID` option can be used to change eight node hex elements, which have 24 to 63 total degrees of freedom to 20 node hex elements, which have 60 to 63 total degrees of freedom. When nodes are automatically added to an element using this option, the corresponding grid points generated will have the same constraints as the adjacent grid points. This insures that the model boundary conditions are correct and that fictitious stress concentrations do not exist at the model boundaries.

The element formulation option also includes an internal node (`PARAM`, `HEXINODE`, `ON`) and/or edge nodes (`PARAM`, `HEXENODE`, `ON`). Both of these options statically condense out the additional nodes. These options generally improve accuracy with a slight penalty in performance. The `HEXENODE` option is ignored when the `HEXEGRID` option is set to `ON`. Table 24-3 shows the various element formulations and the degrees of freedom for each.

Table 24-3. Hex Element Formulation Options.

Element Type	Geometry Processor Parameter Settings	Nodal DOF	Element DOF		
		External [3]	Internal [2]	External [3]	Total
`HEXA`	`HEXINODE = OFF` `HEXENODE = OFF`	3	0	24	24
`HEXA`	`HEXINODE = ON` [1] `HEXENODE = OFF`	3	3	24	27
`HEXA`	`HEXINODE = OFF` `HEXENODE = ON`	3	36	24	60
`HEXA`	`HEXINODE = ON` `HEXENODE = ON`	3	39	24	63
`HEXA`	`HEXINODE = OFF` `HEXEGRID = ON`	3	0	60	60
`HEXA`	`HEXINODE = ON` `HEXEGRID = ON`	3	3	60	63

Notes:

1. Default setting.
2. Internal degrees of freedom are generated by static condensation of internal element nodes.
3. External degrees of freedom are generated from corner and mid-side nodes, which have grid points associated with them.

Generally, the more degrees of freedom an element has, the more accurate it will be especially when it is irregularly shaped.

24.4 2-Dimensional Composite Analysis

The `PCOMP` Bulk Data entry in Autodesk Inventor Nastran is used to model composite shell and solid element properties. In this section we will discuss the application to shells. Section 24.4 discusses the application to solid elements.

The composite shell is defined as a stacked group of lamina or plies, each having its own material properties (`MAT1`, `MAT2`, or `MAT8`), orientation, and stress limits. Each lamina may be considered as a group of unidirectional fibers. The principal material axes for the lamina are parallel and perpendicular to the fiber directions. The principal directions are referred to as "longitudinal" or the 1-direction of the fiber and as "transverse" or the 2-direction for the perpendicular direction (matrix direction).

A stacked group of lamina is called a laminate. The lamina are bonded together with a thin layer of zero thickness bonding material. Each lamina can be modeled as an isotropic material (`MAT1`), a 2-dimensional anisotropic material (`MAT2`), or a 2-dimensional orthotropic material (`MAT8`). The following assumptions are made in lamination theory:

- Each lamina is in a state of plane stress.
- The bonding is perfect.
- 2-Dimensional plate theory can be used.

The material properties for each of the lamina are used to generate equivalent `PSHELL` and `MAT2` Bulk Data entries. These equivalent properties are output as `PSHELL` and `MAT2` Bulk Data entries which are written to the Bulk Data Output File when `TRSLMODLDATA` is set to `ON` in the Model Initialization File or on the Nastran command line. Composite material property data is also written the Model Results Output File when `MODLDATAOUT` is set to `ON` in the Model Initialization File or on the Nastran command line.

Composite element output includes:

- Lamina (ply) and interlaminar (bond) stress or strain output.
- Failure index or strength ratio output (use `PARAM`, `STRENGTHRATIO`, `ON` to obtain strength ratio instead of failure index output).
- Equivalent plate stress, strain, or force output (use `PARAM`, `NOCOMPS`, `ON` to obtain equivalent plate instead of individual lamina output).
- Stability index for sandwich laminates.

Failure index and strength ratio output requires that the appropriate stress limits be specified on the lamina material property definition (`MAT1`, `MAT2`, or `MAT8`) and that the failure theory be specified on the `PCOMP` Bulk Data entry. Stability index will be generated for laminates with a minimum of 3 plies and HCS, FCS, or ACS specified in the LAM field of the `PCOMP` Bulk Data entry.

2-Dimensional composites are supported in all solutions. Individual ply output is not available in solutions with complex results output such as frequency and random response. For these solutions standard shell element results will be output.

We will now look at two examples of how to use the `PCOMP` Bulk Data entry. The first example is the cantilevered honeycomb sandwich plate shown in Figure 24-2. Sandwich materials are a form of composite lay-up that can be analyzed effectively using the `PCOMP` entry.

The section material properties are defined in Table 24-4. The Model Input File is shown in Listing 24-2. Note that the face sheets are not the same thickness. This results in a nonsymmetric plate and a nonblank `MID4` field on the generated equivalent `PSHELL` Bulk Data entry shown in Listing 24-3.

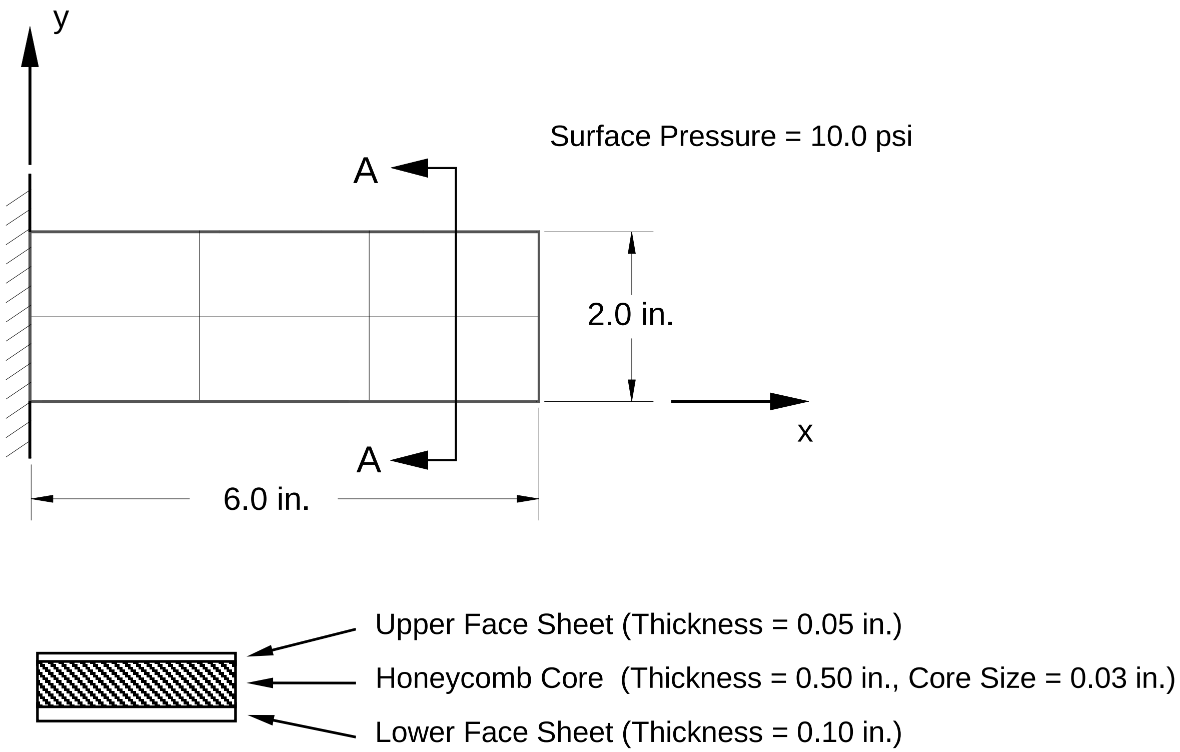


Figure 24-2. Honeycomb Sandwich Cantilever Plate Example Problem.

Table 24-4. Honeycomb Sandwich Material Properties.

Material	Modulus of Elasticity (msi)	Tensile Limit (ksi)	Compression Limit (ksi)	Shear Limit (ksi)
Aluminum Face Sheets	10.0	35.0	35.0	23.0
Honeycomb Core	0.1	0.1	0.3	0.2
Bonding Material	—	—	—	0.1

Listing 24-2. Honeycomb Sandwich Cantilever Plate Model Input File.

```
$
$ STATIC SOLUTION.
$
SOL STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = 3-D HONEYCOMB SANDWICH CANTILEVER PLATE -QUAD4 ELEMENTS -2X3 MESH
$
DISPLACEMENT = ALL
STRESS = ALL
$
SPC = 1
SUBCASE 1
 LABEL = 10 PSI SURFACE PRESSURE
 LOAD = 1
$
BEGIN BULK
$
$ GEOMETRY DEFINITION (6" X 2" RECTANGULAR FLAT PLATE WITH A 3 X 2 MESH).
$
GRID,   4,  ,  2.,  0.,  0.
GRID,   5,  ,  2.,  1.,  0.
GRID,   6,  ,  2.,  2.,  0.
GRID,   7,  ,  6.,  0.,  0.
GRID,   8,  ,  6.,  1.,  0.
GRID,   9,  ,  6.,  2.,  0.
GRID,  10,  ,  4.,  0.,  0.
GRID,  11,  ,  4.,  1.,  0.
GRID,  12,  ,  4.,  2.,  0.
GRID,  16,  ,  0.,  0.,  0.
GRID,  17,  ,  0.,  1.,  0.
GRID,  18,  ,  0.,  2.,  0.
$
$ FLAT PLATE MODELED WITH SHELL ELEMENTS.
$
CQUAD4,  1,  10,  16,   4,   5,  17
CQUAD4,  2,  10,   4,  10,  11,   5
CQUAD4,  3,  10,  10,   7,   8,  11
CQUAD4,  4,  10,  17,   5,   6,  18
CQUAD4,  5,  10,   5,  11,  12,   6
CQUAD4,  6,  10,  11,   8,   9,  12
$
$ COMPOSITE LAMINATE PROPERTY DEFINITION.
$
PCOMP,  10,  ,  ,  100.,  STRESS,  ,  ,  HCS,
,  110,  0.1,  0.,  YES,  120,  0.5,  0.,  YES
,  110,  0.05, 0.,  YES
$
$ FACE SHEET MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  110,  10.E+6,  ,  0.33,  ,  13.E-6,
,  35.E+3,  35.E+3,  23.E+3
$
$ CORE MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  120,  0.1E+6,  ,  0.33,  ,  0.1E-6,
,  0.1E+3,  0.3E+3,  0.2E+3,  ,  0.03

$
$ FIXED BOUNDARY CONDITION AT ONE END.
$
SPC1,  1,  123456,  16,  17,  18
$
$ SURFACE PRESSURE LOADING.
$
PLOAD2,  1,  10.0,  1,  THRU,  6
ENDDATA
```

Listing 24-3. Generated `PSHELL` and `MAT2` Bulk Data Entries Written to the Bulk Data Output File.

```
PSHELL        10     121 0.65000     122 1.00000     123 1.00000      0.+C    1A
+C    1A-0.32500 0.32500     124
MAT2         1210.2676+7 883092.      0.0.2676+7      0. 896472.      0.+C    4A
+C    4A0.1258-40.1258-4      0.      0.              0.      0.      0.
MAT2         1220.6014+70.1984+7      0.0.6014+7      0.0.2015+7      0.+C    5A
+C    5A0.1289-40.1289-4      0.      0.              0.      0.      0.
MAT2         123 45874.0      0.      0. 45874.0      0.      0.      0.+C    6A
+C    6A      0.      0.      0.      0.              0.      0.      0.
MAT2         124 328694. 108469.      0. 328694.      0. 110113.      0.+C    7A
+C    7A 5.74009 5.74009      0.      0.              0.      0.      0.
```

The plate is loaded with a 10 psi surface pressure, which results in bending about the model y-axis. Because the plate section is not symmetric, membrane-bending coupling is generated which results in an x-displacement along with the expected z-displacement and y-rotation. The displacements are shown in Listing 24-4. The element stresses are shown in Listing 24-5.

The element failure indexes are shown in Listing 24-6 and the strength ratios in Listing 24-7 (obtained by adding `PARAM`, `STRENGTHRATIO`, `ON` to the Case Control Section of the Model Input File). Note that for the STRESS, and STRAIN failure theories, the strength ratio is the inverse of the failure index. For all failure theories, the strength ratio is directly proportional to the applied loading. For example, if a laminate has a strength ratio of 2.0, the applied loading can be doubled before failure occurs. For more information on strength ratio see Reference 18.

The stability indexes are shown in Listing 24-8. Allowables for face sheet wrinkling modes are given by:

$$\sigma_{wr} = k_1 (E_f E_c G_c)^{1/3}$$

for foam core sandwich materials and

$$\sigma_{wr} = k_2 E_f \sqrt{\frac{E_c t_f}{E_f t_c}}$$

for honeycomb core materials where,

k_1 is given by `PARAM`, `COMPK1`

k_1 is given by `PARAM`, `COMPK2`

E_f is Young's Modulus for the face sheet

E_c is Young's Modulus for the core

G_c is the transverse shear modulus for the core

t_f is face sheet thickness

t_c is core thickness

Allowables for face sheet dimpling of sandwich materials with honeycomb cores is given by:

$$\sigma_{dp} = \frac{2E_f}{(1-\upsilon^2)}\left(\frac{t_f}{s}\right)^2$$

where,

s is the cell size of the honeycomb core specified on the `MATi` Bulk Data entry

υ is Poisson’s ratio for the face sheet

Allowables for face sheet crimping is given by:

$$\sigma_{cr} = \frac{t_c G_c}{t_{f1} + t_{f2}}$$

where,

t_{f_1} is the bottom face sheet thickness

t_{f_2} is the top face sheet thickness

For more information on face sheet stability in sandwich structures see Reference 9.

Listing 24-4. Honeycomb Sandwich Plate Displacements.

```
                                             D I S P L A C E M E N T   V E C T O R

      GRID      COORDINATE         T1             T2             T3             R1             R2             R3
       ID           ID
        4            0      -1.704834E-04 -1.944323E-05  6.095832E-03  3.307392E-04 -2.170236E-03 -3.815982E-06
        5            0      -1.704490E-04  0.000000E+00  6.255492E-03  0.000000E+00 -2.157586E-03  0.000000E+00
        6            0      -1.704834E-04  1.944323E-05  6.095832E-03 -3.307392E-04 -2.170236E-03  3.815982E-06
        7            0      -2.544360E-04  1.170822E-06  2.106090E-02  1.108142E-06 -3.249578E-03  5.738955E-06
        8            0      -2.597582E-04  0.000000E+00  2.104670E-02  0.000000E+00 -3.244511E-03  0.000000E+00
        9            0      -2.544360E-04 -1.170822E-06  2.106090E-02 -1.108142E-06 -3.249578E-03 -5.738955E-06
       10            0      -2.424810E-04 -7.402531E-06  1.389607E-02  3.828805E-05 -3.050755E-03  5.712113E-06
       11            0      -2.423619E-04  0.000000E+00  1.392396E-02  0.000000E+00 -3.101215E-03  0.000000E+00
       12            0      -2.424810E-04  7.402531E-06  1.389607E-02 -3.828805E-05 -3.050755E-03 -5.712113E-06
```

Listing 24-5. Honeycomb Sandwich Plate Stresses.

```
             S T R E S S E S   I N   C O M P O S I T E S   Q U A D   E L E M E N T S   O N   S U R F A C E   0

   SURFACE COORDINATE ID = ELEMENT  X-AXIS = X  NORMAL = Z

   ELEMENT   PLY  STRESSES IN FIBER AND MATRIX DIRECTIONS  INTER-LAMINAR STRESSES     PRINCIPAL STRESSES (ZERO SHEAR)    MAX SHEAR
     ID      ID      NORMAL-1      NORMAL-2      SHEAR-12    SHEAR XZ-MAT SHEAR YZ-MAT   ANGLE      MAJOR         MINOR

      1       1   2.25009E+03  3.84979E+02  6.07343E+01   8.65419E+01 -1.82116E-02    1.86  2.25206E+03  3.83004E+02   9.34531E+02
              2  -1.20873E+01 -2.60321E+00 -2.53857E-01   8.42174E+01 -1.77224E-02  -88.47 -2.59642E+00 -1.20940E+01   4.74881E+00
              3  -4.37931E+03 -8.51846E+02 -1.04329E+02   0.00000E+00  0.00000E+00  -88.31 -8.48763E+02 -4.38239E+03   1.76681E+03

      2       1   8.65419E+02 -8.75954E+01 -4.44369E+01   5.19251E+01 -4.56677E-01   -2.66  8.67487E+02 -8.96629E+01   4.78575E+02
              2  -4.64894E+00  2.69420E-01  1.66997E-01   5.05305E+01 -4.44411E-01   88.06  2.75084E-01 -4.65461E+00   2.46485E+00
              3  -1.68435E+03  1.31935E+02  7.27417E+01   0.00000E+00  0.00000E+00   87.71  1.34843E+02 -1.68726E+03   9.11050E+02

      3       1   1.73084E+02  3.41064E+01  1.21325E+01   1.73084E+01  3.29923E-01    4.95  1.74135E+02  3.30552E+01   7.05399E+01
              2  -9.29789E-01  5.40003E-02 -2.98213E-02   1.68435E+01  3.21062E-01  -88.27  5.49035E-02 -9.30692E-01   4.92798E-01
              3  -3.36870E+02 -2.09142E+01 -1.68372E+01   0.00000E+00  0.00000E+00  -86.96 -2.00195E+01 -3.37764E+02   1.58872E+02

      4       1   2.25009E+03  3.84979E+02 -6.07343E+01   8.65419E+01  1.82116E-02   -1.86  2.25206E+03  3.83004E+02   9.34531E+02
              2  -1.20873E+01 -2.60321E+00  2.53857E-01   8.42174E+01  1.77224E-02   88.47 -2.59642E+00 -1.20940E+01   4.74881E+00
              3  -4.37931E+03 -8.51846E+02  1.04329E+02   0.00000E+00  0.00000E+00   88.31 -8.48763E+02 -4.38239E+03   1.76681E+03

      5       1   8.65419E+02 -8.75954E+01  4.44369E+01   5.19251E+01  4.56677E-01    2.66  8.67487E+02 -8.96629E+01   4.78575E+02
              2  -4.64894E+00  2.69420E-01 -1.66997E-01   5.05305E+01  4.44411E-01  -88.06  2.75084E-01 -4.65461E+00   2.46485E+00
              3  -1.68435E+03  1.31935E+02 -7.27417E+01   0.00000E+00  0.00000E+00  -87.71  1.34843E+02 -1.68726E+03   9.11050E+02

      6       1   1.73084E+02  3.41064E+01 -1.21325E+01   1.73084E+01 -3.29923E-01   -4.95  1.74135E+02  3.30552E+01   7.05399E+01
              2  -9.29789E-01  5.40003E-02  2.98213E-02   1.68435E+01 -3.21062E-01   88.27  5.49035E-02 -9.30692E-01   4.92798E-01
              3  -3.36870E+02 -2.09142E+01  1.68372E+01   0.00000E+00  0.00000E+00   86.96 -2.00195E+01 -3.37764E+02   1.58872E+02
```

Listing 24-6. Honeycomb Sandwich Plate Failure Indexes.

```
        F A I L U R E   I N D E X E S   F O R   C O M P O S I T E   Q U A D   E L E M E N T S   O N   S U R F A C E   0

     ELEMENT    FAILURE   PLY    MATRIX FAILURE    FAILURE       FIBER FAILURE     FAILURE        BOND FAILURE    ELEMENT FAILURE
       ID       THEORY    ID         INDEX          MODE            INDEX            MODE            INDEX              INDEX

        1        STRESS     1      6.43E-02  1                   6.43E-02  1
                                                                                                  8.65419E-01
                            2      4.03E-02  1                   4.03E-02  1
                                                                                                  8.42174E-01
                            3      1.25E-01  1                   1.25E-01  1                                          8.65419E-01


        2        STRESS     1      2.47E-02  1                   2.47E-02  1
                                                                                                  5.19251E-01
                            2      1.55E-02  1                   1.55E-02  1
                                                                                                  5.05305E-01
                            3      4.81E-02  1                   4.81E-02  1                                          5.19251E-01


        3        STRESS     1      4.95E-03  1                   4.95E-03  1
                                                                                                  1.73084E-01
                            2      3.10E-03  1                   3.10E-03  1
                                                                                                  1.68435E-01
                            3      9.62E-03  1                   9.62E-03  1                                          1.73084E-01


        4        STRESS     1      6.43E-02  1                   6.43E-02  1
                                                                                                  8.65419E-01
                            2      4.03E-02  1                   4.03E-02  1
                                                                                                  8.42174E-01
                            3      1.25E-01  1                   1.25E-01  1                                          8.65419E-01


        5        STRESS     1      2.47E-02  1                   2.47E-02  1
                                                                                                  5.19251E-01
                            2      1.55E-02  1                   1.55E-02  1
                                                                                                  5.05305E-01
                            3      4.81E-02  1                   4.81E-02  1                                          5.19251E-01


        6        STRESS     1      4.95E-03  1                   4.95E-03  1
                                                                                                  1.73084E-01
                            2      3.10E-03  1                   3.10E-03  1
                                                                                                  1.68435E-01
                            3      9.62E-03  1                   9.62E-03  1                                          1.73084E-01
```

Listing 24-7. Honeycomb Sandwich Plate Strength Ratios.

```
         S T R E N G T H   R A T I O S   F O R   C O M P O S I T E   Q U A D   E L E M E N T S   O N   S U R F A C E   0

   ELEMENT    FAILURE    PLY   MATRIX STRENGTH   FAILURE       FIBER STRENGTH   FAILURE       BOND STRENGTH     ELEMENT STRENGTH
     ID       THEORY     ID         RATIO         MODE              RATIO        MODE             RATIO               RATIO

       1        STRESS      1     1.56E+01  1                  1.56E+01  1
                                                                                               1.15551E+00
                            2     2.48E+01  1                  2.48E+01  1
                                                                                               1.18740E+00
                            3     7.99E+00  1                  7.99E+00  1                                        1.15551E+00

       2        STRESS      1     4.04E+01  1                  4.04E+01  1
                                                                                               1.92585E+00
                            2     6.45E+01  1                  6.45E+01  1
                                                                                               1.97900E+00
                            3     2.08E+01  1                  2.08E+01  1                                        1.92585E+00

       3        STRESS      1     2.02E+02  1                  2.02E+02  1
                                                                                               5.77755E+00
                            2     3.23E+02  1                  3.23E+02  1
                                                                                               5.93701E+00
                            3     1.04E+02  1                  1.04E+02  1                                        5.77755E+00

       4        STRESS      1     1.56E+01  1                  1.56E+01  1
                                                                                               1.15551E+00
                            2     2.48E+01  1                  2.48E+01  1
                                                                                               1.18740E+00
                            3     7.99E+00  1                  7.99E+00  1                                        1.15551E+00

       5        STRESS      1     4.04E+01  1                  4.04E+01  1
                                                                                               1.92585E+00
                            2     6.45E+01  1                  6.45E+01  1
                                                                                               1.97900E+00
                            3     2.08E+01  1                  2.08E+01  1                                        1.92585E+00

       6        STRESS      1     2.02E+02  1                  2.02E+02  1
                                                                                               5.77755E+00
                            2     3.23E+02  1                  3.23E+02  1
                                                                                               5.93701E+00
                            3     1.04E+02  1                  1.04E+02  1                                        5.77755E+00
```

Listing 24-8. Honeycomb Sandwich Plate Stability Indexes.

```
      S T A B I L I T Y   I N D E X E S   F O R   C O M P O S I T E   Q U A D   E L E M E N T S   O N   S U R F A C E  0

   ELEMENT   PLY             STRESS LIMITS                        STABILITY INDEXES              CRITICAL     FAILURE
     ID      ID      WRINKLING    DIMPLING     CRIMPING      WRINKLING    DIMPLING     CRIMPING       INDEX        MODE

       1       1    2.59307E+05  6.23449E+07  1.25313E+05                                                      STRESS
               3    2.59307E+05  6.23449E+07  1.25313E+05   1.69004E-02  7.02926E-05  3.49718E-02   3.49718E-02   CRIMPING

       2       1    2.59307E+05  6.23449E+07  1.25313E+05   3.45779E-04  1.43817E-06  7.15510E-04   7.15510E-04   CRIMPING
               3    2.59307E+05  6.23449E+07  1.25313E+05   6.50680E-03  2.70633E-05  1.34643E-02   1.34643E-02   CRIMPING

       3       1    2.59307E+05  6.23449E+07  1.25313E+05                                                      STRESS
               3    2.59307E+05  6.23449E+07  1.25313E+05   1.30257E-03  5.41767E-06  2.69536E-03   2.69536E-03   CRIMPING

       4       1    2.59307E+05  6.23449E+07  1.25313E+05                                                      STRESS
               3    2.59307E+05  6.23449E+07  1.25313E+05   1.69004E-02  7.02926E-05  3.49718E-02   3.49718E-02   CRIMPING

       5       1    2.59307E+05  6.23449E+07  1.25313E+05   3.45779E-04  1.43817E-06  7.15510E-04   7.15510E-04   CRIMPING
               3    2.59307E+05  6.23449E+07  1.25313E+05   6.50680E-03  2.70633E-05  1.34643E-02   1.34643E-02   CRIMPING

       6       1    2.59307E+05  6.23449E+07  1.25313E+05                                                      STRESS
               3    2.59307E+05  6.23449E+07  1.25313E+05   1.30257E-03  5.41767E-06  2.69536E-03   2.69536E-03   CRIMPING
```

The second example is the cantilevered composite plate shown in Figure 24-3. The lamina and laminate properties are defined in Tables 24-5 and 24-6, respectively. The Model Input File is shown in Listing 24-9.

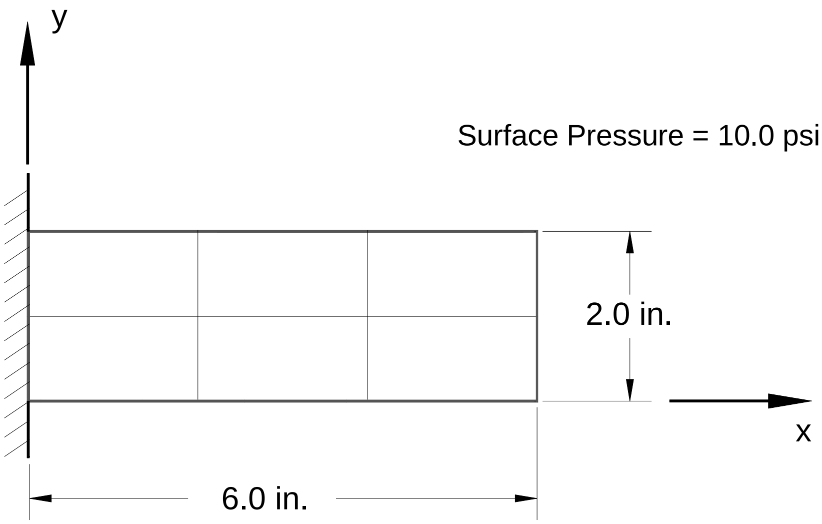


Figure 24-3. Composite Cantilever Plate Example Problem.

Table 24-5a. Individual Lamina Material Properties – Moduli.

Material ID	E_1 (msi)	E_2 (msi)	υ_{12}	G_{12} (msi)	G_{1z} (msi)	G_{2z} (msi)
110	10.0	10.0	0.1	1.0	1.0	1.0
120	10.0	1.0	0.2	4.0	2.0	2.0
130	10.0	2.0	0.1	3.0	1.5	1.5

Table 24-5b. Individual Lamina Material Properties – Stress Limits.

Material ID	X_t (ksi)	X_c (ksi)	Y_t (ksi)	Y_c (ksi)	S (ksi)
110	60.0	50.0	60.0	50.0	5.0
120	40.0	80.0	4.0	8.0	5.0
130	20.0	10.0	15.0	10.0	4.0

Table 24-6. Composite Laminate Material Properties.

Ply	Material ID	Thickness (inches)	Orientation (degrees)
1	110	0.05	0.0
2	120	0.07	45.0
3	130	0.06	90.0
4	120	0.05	60.0
5	110	0.02	0.0

Ply orientation is relative to the material axis of the element. The angles given in Table 24-6 are relative to this axis. The default material axis for shell elements is the element edge 1-2 defined by nodes 1 and 2. Since the quad elements in our example are rectangular we can use the default axis. Typically this is not the case and the material orientation must be defined explicitly. See the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for more information on `CQUAD4`, `CQUADR`, `CQUAD8`, `CTRIA3`, `CTRIAR`, `CTRIA6`, and `PSHELL` Bulk Data entries and material axis orientation.

Listing 24-9. 2-Dimensional Composite Cantilever Plate Model Input File.

```
$
$ STATIC SOLUTION.
$
SOL STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = COMPOSITE CANTILEVER BEAM -QUAD4 ELEMENTS -2X3 MESH
$
DISPLACEMENT = ALL
STRESS = ALL
$
SPC = 1
SUBCASE 1
 LABEL = 10 PSI SURFACE PRESSURE
 LOAD = 1
$
BEGIN BULK
$
$ GEOMETRY DEFINITION (6" X 2" RECTANGULAR FLAT PLATE WITH A 3 X 2 MESH).
$
GRID,   4,  ,  2.,  0.,  0.
GRID,   5,  ,  2.,  1.,  0.
GRID,   6,  ,  2.,  2.,  0.
GRID,   7,  ,  6.,  0.,  0.
GRID,   8,  ,  6.,  1.,  0.
GRID,   9,  ,  6.,  2.,  0.
GRID,  10,  ,  4.,  0.,  0.
GRID,  11,  ,  4.,  1.,  0.
GRID,  12,  ,  4.,  2.,  0.
GRID,  16,  ,  0.,  0.,  0.
GRID,  17,  ,  0.,  1.,  0.
GRID,  18,  ,  0.,  2.,  0.
$
$ FLAT PLATE MODELED WITH SHELL ELEMENTS.
$
CQUAD4,  1,  10,  16,   4,   5,  17
CQUAD4,  2,  10,   4,  10,  11,   5
CQUAD4,  3,  10,  10,   7,   8,  11
CQUAD4,  4,  10,  17,   5,   6,  18
CQUAD4,  5,  10,   5,  11,  12,   6
CQUAD4,  6,  10,  11,   8,   9,  12
$
$ COMPOSITE LAMINATE PROPERTY DEFINITION.
$
PCOMP,  10,  ,  ,  5.E+3,  HILL,
,  110,  0.05,   0.,  YES,  120,  0.07,  45.,  YES,
,  130,  0.06,  90.,  YES,  120,  0.05,  60.,  YES,
,  110,  0.02,   0.,  YES
$
$ LAMINA MATERIAL PROPERTIES.
$
MAT8,  110,  10.E+6,  10.E+6,  0.1,  1.E+6,  1.E+6,  1.E+6,
,  ,  ,  ,  60.E+3,  50.E+3,  60.E+3,  50.E+3,  5.E+3
MAT8,  120,  10.E+6,  1.E+6,  0.2,  4.E+6,  2.E+6,  2.E+6,
,  ,  ,  ,  40.E+3,  80.E+3,  4.E+3,  8.E+3,  5.E+3
MAT8,  130,  10.E+6,  2.E+6,  0.1,  3.E+6,  1.5E+6,  1.5E+6,
,  ,  ,  ,  20.E+3,  10.E+3,  15.E+3,  10.E+3,  4.E+3
$
$ FIXED BOUNDARY CONDITION AT ONE END.
$
SPC1,  1,  123456,  16,  17,  18
$
$ SURFACE PRESSURE LOADING.
$
PLOAD2,  1,  10.,  1,  THRU,  6
ENDDATA
```

Listing 24-10. Generated `PSHELL` and `MAT2` Bulk Data Entries Written to the Bulk Data Output File.

```
PSHELL        10     131 0.25000     132 1.00000     133 1.00000      0.+C    1A
+C    1A-0.12500 0.12500     134
MAT2         1310.6084+7-148614.0.1140+70.8911+7 907815.0.2344+7      0.+C    8A
+C    8A      0.      0.      0.      0.              0.      0.      0.
MAT2         1320.7926+7 283328. 887526.0.9414+7 593185.0.1771+7      0.+C    9A
+C    9A      0.      0.      0.      0.              0.      0.      0.
MAT2         1330.2280+7      0.      0.0.1818+7      0.      0.      0.+C   10A
+C   10A      0.      0.      0.      0.              0.      0.      0.
MAT2         134 422536. 39257.3-61121.0-59005.7 13113.7-101314.      0.+C   11A
+C   11A      0.      0.      0.      0.              0.      0.      0.
```

The plate is loaded with a 10 psi surface pressure which results in bending about the model y-axis. Because the composite lay-up is not symmetric, membrane-bending coupling is generated which results in an x-displacement along with the expected z-displacement and y-rotation. The lay-up is also not balanced which results in a slight twisting x-rotation. The displacements are shown in Listing 24-11. The element stresses are shown in Listing 24-12. The element failure indexes are shown in Listing 24-13 and the strength ratios in Listing 24-14 (obtained by adding `PARAM`, `STRENGTHRATIO`, `ON` to the Case Control Section of the Model Input File).

Listing 24-11. 2-Dimensional Composite Plate Displacements.

```
                                             D I S P L A C E M E N T   V E C T O R

     GRID       COORDINATE         T1             T2             T3             R1             R2             R3
      ID            ID
        4            0      -5.930695E-04  4.343331E-04  3.138304E-02 -2.249688E-03 -2.903187E-02  1.158399E-04
        5            0      -5.633355E-04  3.933851E-04  2.908152E-02 -2.181653E-03 -2.723822E-02  1.102060E-04
        6            0      -5.338815E-04  3.539528E-04  2.681530E-02 -2.205097E-03 -2.547819E-02  1.035768E-04
        7            0      -8.424426E-04  4.908367E-04  1.809799E-01 -6.204114E-03 -4.057298E-02  5.459421E-05
        8            0      -8.482638E-04  4.889086E-04  1.749577E-01 -6.108129E-03 -4.052350E-02  3.334295E-05
        9            0      -8.373268E-04  4.870649E-04  1.690166E-01 -6.002523E-03 -4.039096E-02  1.414558E-05
       10            0      -8.132125E-04  4.899697E-04  1.008572E-01 -5.262480E-03 -3.915736E-02 -7.165992E-05
       11            0      -7.928412E-04  4.896641E-04  9.579562E-02 -5.106808E-03 -3.821157E-02 -4.982979E-05
       12            0      -7.830227E-04  4.866978E-04  9.087225E-02 -4.959575E-03 -3.728016E-02 -2.853515E-05
```

Listing 24-12. 2-Dimensional Composite Plate Stresses.

```
             S T R E S S E S   I N   C O M P O S I T E S   Q U A D   E L E M E N T S   O N   S U R F A C E   0

SURFACE COORDINATE ID = ELEMENT  X-AXIS = X  NORMAL = Z

ELEMENT   PLY  STRESSES IN FIBER AND MATRIX DIRECTIONS  INTER-LAMINAR STRESSES     PRINCIPAL STRESSES (ZERO SHEAR)    MAX SHEAR
  ID      ID      NORMAL-1     NORMAL-2     SHEAR-12   SHEAR XZ-MAT SHEAR YZ-MAT   ANGLE     MAJOR        MINOR

    1       1   1.12722E+04  9.56493E+02  2.13307E+01   2.46547E+02  1.61686E+02    0.12  1.12722E+04  9.56449E+02   5.15788E+03
            2   1.99977E+03  9.64334E+01 -1.17085E+03   2.62642E+02  1.78571E+02  -25.45  2.55693E+03 -4.60728E+02   1.50883E+03
            3  -3.42086E+02 -1.28842E+03 -8.15739E+02   2.30154E+02  1.34206E+02  -29.94  1.27783E+02 -1.75829E+03   9.43036E+02
            4  -2.31106E+03 -1.27838E+03  4.05529E+03   1.91004E+02  9.59349E+01   48.63  2.29331E+03 -5.88275E+03   4.08803E+03
            5  -1.92859E+04 -2.17245E+03  4.52332E+02   2.68667E+01  2.39332E+01   88.49 -2.16050E+03 -1.92979E+04   8.56868E+03

    2       1   4.18298E+03  3.23883E+02 -2.22389E+02   1.31625E+02 -3.97567E+01   -3.29  4.19575E+03  3.11110E+02   1.94232E+03
            2   1.70473E+02  7.01896E+01 -4.58916E+02   1.40218E+02 -4.39083E+01  -41.88  5.81978E+02 -3.41316E+02   4.61647E+02
            3  -2.83655E+02 -4.94233E+02 -4.03160E+02   1.22873E+02 -3.29995E+01  -37.68  2.77382E+01 -8.05626E+02   4.16682E+02
            4  -4.04614E+02 -5.42444E+02  1.16599E+03   1.01973E+02 -2.35892E+01   43.31  6.94497E+02 -1.64155E+03   1.16803E+03
            5  -7.29650E+03 -1.06455E+03  3.91265E+02   1.43435E+01 -5.88487E+00   86.42 -1.04008E+03 -7.32096E+03   3.14044E+03

    3       1   7.39111E+02  1.88571E+02 -9.10345E+01   4.15318E+01 -6.24960E+00   -9.15  7.53773E+02  1.73908E+02   2.89932E+02
            2  -5.50317E+01  2.49297E+01 -4.87837E+01   4.42430E+01 -6.90222E+00  -64.67  4.80228E+01 -7.81249E+01   6.30738E+01
            3  -5.16206E+01 -8.99531E+01 -9.56618E+01   3.87703E+01 -5.18740E+00  -39.34  2.67761E+01 -1.68350E+02   9.75629E+01
            4   2.68609E+01 -1.11271E+02  1.20878E+02   3.21754E+01 -3.70813E+00   30.13  9.70128E+01 -1.81423E+02   1.39218E+02
            5  -1.31201E+03 -2.87071E+02  1.20391E+02   4.52580E+00 -9.25080E-01   83.39 -2.73120E+02 -1.32596E+03   5.26420E+02

    4       1   1.05204E+04  8.43154E+02  3.89141E+00   1.91809E+02  1.53827E+02    0.02  1.05204E+04  8.43153E+02   4.83861E+03
            2   1.79404E+03  9.09714E+01 -1.09218E+03   2.04330E+02  1.69891E+02  -26.03  2.32741E+03 -4.42402E+02   1.38491E+03
            3  -3.15619E+02 -1.21388E+03 -7.52937E+02   1.79055E+02  1.27682E+02  -29.59  1.11969E+02 -1.64146E+03   8.76716E+02
            4  -2.14485E+03 -1.19982E+03  3.81807E+03   1.48598E+02  9.12715E+01   48.53  2.17487E+03 -5.51953E+03   3.84720E+03
            5  -1.80984E+04 -1.99352E+03  4.28882E+02   2.09018E+01  2.27698E+01   88.48 -1.98210E+03 -1.81098E+04   8.06384E+03

    5       1   4.52748E+03  3.02649E+02 -1.99670E+02   1.31388E+02 -3.84315E+01   -2.70  4.53689E+03  2.93234E+02   2.12183E+03
            2   2.91653E+02  6.73834E+01 -5.07260E+02   1.39965E+02 -4.24448E+01  -38.77  6.99025E+02 -3.39988E+02   5.19507E+02
            3  -2.80429E+02 -5.29624E+02 -4.38099E+02   1.22652E+02 -3.18996E+01  -37.06  5.04461E+01 -8.60499E+02   4.55472E+02
            4  -4.58293E+02 -5.76179E+02  1.30557E+03   1.01789E+02 -2.28029E+01   43.71  7.89660E+02 -1.82413E+03   1.30690E+03
            5  -7.85135E+03 -1.06831E+03  3.94939E+02   1.43176E+01 -5.68872E+00   86.68 -1.04539E+03 -7.87427E+03   3.41444E+03

    6       1   1.10263E+03  2.12632E+02 -9.40237E+01   4.61395E+01 -5.37137E+00   -5.97  1.11245E+03  2.02807E+02   4.54822E+02
            2  -3.89568E+00  3.03663E+01 -9.65751E+01   4.91515E+01 -5.93228E+00  -50.03  1.11318E+02 -8.48474E+01   9.80827E+01
            3  -6.80561E+01 -1.24009E+02 -1.09070E+02   4.30716E+01 -4.45843E+00  -37.81  1.65687E+01 -2.08633E+02   1.12601E+02
            4  -5.71217E+01 -1.46770E+02  2.39909E+02   3.57451E+01 -3.18704E+00   39.71  1.42115E+02 -3.46007E+02   2.44061E+02
            5  -1.86898E+03 -3.56331E+02  1.30231E+02   5.02791E+00 -7.95082E-01   85.12 -3.45201E+02 -1.88011E+03   7.67456E+02
```

Listing 24-13. 2-Dimensional Composite Plate Failure Indexes.

```
    F A I L U R E   I N D E X E S   F O R   C O M P O S I T E   Q U A D   E L E M E N T S   O N   S U R F A C E   0

ELEMENT    FAILURE   PLY    MATRIX FAILURE   FAILURE      FIBER FAILURE    FAILURE       BOND FAILURE    ELEMENT FAILURE
  ID       THEORY    ID         INDEX         MODE            INDEX         MODE            INDEX             INDEX

    1       HILL       1     3.25723E-02                  3.25723E-02
                                                                                         4.93095E-02
                       2     5.77958E-02                  5.77958E-02
                                                                                         5.25284E-02
                       3     5.49524E-02                  5.49524E-02
                                                                                         4.60308E-02
                       4     6.83724E-01                  6.83724E-01
                                                                                         3.82009E-02
                       5     1.42091E-01                  1.42091E-01                                    6.83724E-01

    2       HILL       1     6.49145E-03                  6.49145E-03
                                                                                         2.63251E-02
                       2     8.74275E-03                  8.74275E-03
                                                                                         2.80436E-02
                       3     1.20040E-02                  1.20040E-02
                                                                                         2.45747E-02
                       4     5.89703E-02                  5.89703E-02
                                                                                         2.03945E-02
                       5     2.47654E-02                  2.47654E-02                                    5.89703E-02

    3       HILL       1     4.54399E-04                  4.54399E-04
                                                                                         8.30636E-03
                       2     1.34725E-04                  1.34725E-04
                                                                                         8.84859E-03
                       3     6.33076E-04                  6.33076E-04
                                                                                         7.75405E-03
                       4     7.80239E-04                  7.80239E-04
                                                                                         6.43508E-03
                       5     1.15061E-03                  1.15061E-03                                    8.84859E-03

    4       HILL       1     2.84781E-02                  2.84781E-02
                                                                                         3.83618E-02
                       2     5.01411E-02                  5.01411E-02
                                                                                         4.08661E-02
                       3     4.73320E-02                  4.73320E-02
                                                                                         3.58111E-02
                       4     6.05917E-01                  6.05917E-01
                                                                                         2.97196E-02
                       5     1.25536E-01                  1.25536E-01                                    6.05917E-01

    5       HILL       1     6.93346E-03                  6.93346E-03
                                                                                         2.62777E-02
                       2     1.06172E-02                  1.06172E-02
                                                                                         2.79931E-02
                       3     1.41019E-02                  1.41019E-02
                                                                                         2.45304E-02
                       4     7.33590E-02                  7.33590E-02
                                                                                         2.03578E-02
                       5     2.79980E-02                  2.79980E-02                                    7.33590E-02

    6       HILL       1     6.38769E-04                  6.38769E-04
                                                                                         9.22790E-03
                       2     4.30722E-04                  4.30722E-04
                                                                                         9.83029E-03
                       3     8.59222E-04                  8.59222E-04
                                                                                         8.61432E-03
                       4     2.63805E-03                  2.63805E-03
                                                                                         7.14901E-03
                       5     1.86004E-03                  1.86004E-03                                    9.83029E-03
```

Listing 24-14. 2-Dimensional Composite Plate Strength Ratios.

```
        S T R E N G T H   R A T I O S   F O R   C O M P O S I T E   Q U A D   E L E M E N T S   O N   S U R F A C E   0

    ELEMENT    FAILURE   PLY   MATRIX STRENGTH    FAILURE      FIBER STRENGTH   FAILURE      BOND STRENGTH    ELEMENT STRENGTH
      ID       THEORY    ID         RATIO          MODE            RATIO         MODE            RATIO             RATIO

        1       HILL        1     5.54084E+00                   5.54084E+00
                                                                                              2.02801E+01
                            2     4.15960E+00                   4.15960E+00
                                                                                              1.90373E+01
                            3     4.26586E+00                   4.26586E+00
                                                                                              2.17246E+01
                            4     1.20937E+00                   1.20937E+00
                                                                                              2.61774E+01
                            5     2.65287E+00                   2.65287E+00                                     1.20937E+00

        2       HILL        1     1.24116E+01                   1.24116E+01
                                                                                              3.79866E+01
                            2     1.06949E+01                   1.06949E+01
                                                                                              3.56588E+01
                            3     9.12720E+00                   9.12720E+00
                                                                                              4.06923E+01
                            4     4.11797E+00                   4.11797E+00
                                                                                              4.90328E+01
                            5     6.35444E+00                   6.35444E+00                                     4.11797E+00

        3       HILL        1     4.69117E+01                   4.69117E+01
                                                                                              1.20390E+02
                            2     8.61542E+01                   8.61542E+01
                                                                                              1.13012E+02
                            3     3.97440E+01                   3.97440E+01
                                                                                              1.28965E+02
                            4     3.58003E+01                   3.58003E+01
                                                                                              1.55398E+02
                            5     2.94805E+01                   2.94805E+01                                     2.94805E+01

        4       HILL        1     5.92577E+00                   5.92577E+00
                                                                                              2.60676E+01
                            2     4.46584E+00                   4.46584E+00
                                                                                              2.44702E+01
                            3     4.59645E+00                   4.59645E+00
                                                                                              2.79243E+01
                            4     1.28468E+00                   1.28468E+00
                                                                                              3.36479E+01
                            5     2.82238E+00                   2.82238E+00                                     1.28468E+00

        5       HILL        1     1.20095E+01                   1.20095E+01
                                                                                              3.80551E+01
                            2     9.70500E+00                   9.70500E+00
                                                                                              3.57231E+01
                            3     8.42096E+00                   8.42096E+00
                                                                                              4.07657E+01
                            4     3.69210E+00                   3.69210E+00
                                                                                              4.91213E+01
                            5     5.97635E+00                   5.97635E+00                                     3.69210E+00

        6       HILL        1     3.95665E+01                   3.95665E+01
                                                                                              1.08367E+02
                            2     4.81838E+01                   4.81838E+01
                                                                                              1.01726E+02
                            3     3.41152E+01                   3.41152E+01
                                                                                              1.16086E+02
                            4     1.94697E+01                   1.94697E+01
                                                                                              1.39879E+02
                            5     2.31867E+01                   2.31867E+01                                     1.94697E+01
```

24.5 3-Dimensional Composite Analysis

The `PCOMP` Bulk Data entry in Autodesk Inventor Nastran can also be used to model 3-dimensional composite solid element properties (`CHEXA` and `CPENTA` elements only). Like the composite shell, the composite solid is defined as a stacked group of lamina or plies, each having its own material properties (`MAT1`, `MAT9`, or `MAT12`), orientation, and stress limits. Each lamina may be considered as a group of unidirectional fibers. The principal material axes for the lamina are parallel and perpendicular to the fiber directions. The principal directions are referred to as "longitudinal" or the 1-direction of the fiber, as "transverse" or the 2-direction for the perpendicular direction (matrix direction), and as "thickness" or the 3-direction for the through thickness direction.

A stacked group of lamina is called a laminate. The lamina are bonded together with a thin layer of zero thickness bonding material. Each lamina can be modeled as an isotropic material (`MAT1`), a 3-dimensional anisotropic material (`MAT9`), or a 3-dimensional orthotropic material (`MAT12`). Each layer is in a state of general stress. The only assumption made is that the bonding is perfect.

Composite element output includes:

- Lamina (ply) and interlaminar (bond) stress or strain output.
- Failure index or strength ratio output (use `PARAM`, `STRENGTHRATIO`, `ON` to obtain strength ratio instead of failure index output).
- Equivalent plate stress, strain, or force output (use `PARAM`, `NOCOMPS`, `ON` to obtain equivalent plate instead of individual lamina output).

Failure index and strength ratio output requires that the appropriate stress limits be specified on the lamina material property definition (`MAT1` or `MAT12`) and that the failure theory be specified on the `PCOMP` Bulk Data entry.

3-Dimensional composites are supported in all linear solutions. In nonlinear solutions an equivalent anisotropic solid material property (`MAT9`) will be used. Individual ply output is not available in solutions with complex results output such as frequency and random response. For these solutions standard solid element results will be output.

We will now look at how to use the `PCOMP` and `PSOLID` Bulk Data entries for modeling 3-dimensional composites using the cantilevered composite plate example shown in Figure 24-3. The lamina and laminate properties are defined in Tables 24-7 and 24-6, respectively. The Model Input File is shown in Listing 24-15.

The definition of the ply orientation is handled differently for the 3-dimensional composite as compared to the 2-dimensional one. Ply orientation is relative to the projection of the material x-direction on the element surface where the surface normal is defined by the element z-axis. The angles given in Table 24-6 are relative to this axis. The default material coordinate system for solid elements is the element system. Since the hex elements in our example are rectangular we can use the default axis. Typically this is not the case and the material orientation must be defined explicitly on the MCID field of the `PSOLID` entry. See the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for more information on `PSOLID` Bulk Data entry and material axis orientation.

3-Dimensional composites are enabled when a `PCOMP` identification number is referenced in the PCPID field of the `PSOLID` Bulk Data entry. The thicknesses defined on the `PCOMP` entry are relative to the element z-axis. The thickness total on the `PCOMP` entry is not required to be equal to the element thickness. Instead these values are converted to percentages at each element integration point to accommodate element taper and twist. The number of integration points in the 3-direction of the ply is controlled using `PARAM`, `NSLDPLYINTPOINT` which is defaulted to 3.

Table 24-7a. Individual Lamina Material Properties – Moduli.

Material ID	E_1 (msi)	E_2 (msi)	E_3 (msi)	υ_{12}	υ_{23}	υ_{31}	G_{12} (msi)	G_{23} (msi)	G_{31} (msi)
110	10.0	10.0	0.1	0.1	0.0	0.0	1.0	1.0	1.0
120	10.0	1.0	0.1	0.2	0.0	0.0	4.0	2.0	2.0
130	10.0	2.0	0.1	0.1	0.0	0.0	3.0	1.5	1.5

Table 24-7b. Individual Lamina Material Properties – Stress Limits.

Material ID	X_t (ksi)	X_c (ksi)	Y_t (ksi)	Y_c (ksi)	Z_t (ksi)	Z_c (ksi)	S_{12} (ksi)	S_{23} (ksi)	S_{31} (ksi)
110	60.0	50.0	60.0	50.0	10.0	10.0	5.0	4.0	4.0
120	40.0	80.0	4.0	8.0	1.0	2.0	5.0	4.0	4.0
130	20.0	10.0	15.0	10.0	1.0	1.0	4.0	3.0	3.0

Because 3-dimensional composite analysis uses the `PCOMP` Bulk Data entry the output will include the generation of equivalent 2-dimensional `MAT2` anisotropic material and `PSHELL` shell properties as well as an equivalent `MAT9` anisotropic solid material property. The equivalent anisotropic solid property is only used in solutions which do not support laminated solid elements.

Listing 24-15. 3-Dimensional Composite Cantilever Plate Model Input File.

```
$
$ STATIC SOLUTION.
$
SOL STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = COMPOSITE CANTILEVER BEAM -HEX ELEMENTS -2X3 MESH
$
DISPLACEMENT = ALL
STRESS = ALL
$
SPC = 1
SUBCASE 1
 LABEL = 10 PSI SURFACE PRESSURE
 LOAD = 1
$
BEGIN BULK
$
$ GEOMETRY DEFINITION (6" X 2" RECTANGULAR FLAT PLATE WITH A 3 X 2 MESH).
$
GRID,   1,  ,  2.,  0.,    0.
GRID,   2,  ,  2.,  1.,    0.
GRID,   3,  ,  2.,  2.,    0.
GRID,   4,  ,  6.,  0.,    0.
GRID,   5,  ,  6.,  1.,    0.
GRID,   6,  ,  6.,  2.,    0.
GRID,   7,  ,  4.,  0.,    0.
GRID,   8,  ,  4.,  1.,    0.
GRID,   9,  ,  4.,  2.,    0.
GRID,  10,  ,  0.,  0.,    0.
GRID,  11,  ,  0.,  1.,    0.
GRID,  12,  ,  0.,  2.,    0.
GRID,  13,  ,  2.,  0.,  0.25
GRID,  14,  ,  2.,  1.,  0.25
GRID,  15,  ,  2.,  2.,  0.25
GRID,  16,  ,  6.,  0.,  0.25
GRID,  17,  ,  6.,  1.,  0.25
GRID,  18,  ,  6.,  2.,  0.25
GRID,  19,  ,  4.,  0.,  0.25
GRID,  20,  ,  4.,  1.,  0.25
GRID,  21,  ,  4.,  2.,  0.25
GRID,  22,  ,  0.,  0.,  0.25
GRID,  23,  ,  0.,  1.,  0.25
GRID,  24,  ,  0.,  2.,  0.25
$
$ FLAT PLATE MODELED WITH SOLID ELEMENTS.
$
CHEXA,   7,  10,  10,   1,   2,  11,  22,  13,
,  14,  23
CHEXA,   8,  10,   1,   7,   8,   2,  13,  19,
,  20,  14
CHEXA,   9,  10,   7,   4,   5,   8,  19,  16,
,  17,  20
CHEXA,  10,  10,  11,   2,   3,  12,  23,  14,
,  15,  24
CHEXA,  11,  10,   2,   8,   9,   3,  14,  20,
,  21,  15
CHEXA,  12,  10,   8,   5,   6,   9,  20,  17,
,  18,  21
$
$ SOLID ELEMENT PROPERTY DEFINITION IN ELEMENT COORDINATE SYSTEM.
$
PSOLID,  10,  1,  ,  20
$
$ COMPOSITE LAMINATE PROPERTY DEFINITION.
$
PCOMP,  20,  ,  ,  5.E+3,  HILL,
,  110,  0.05,   0.,  YES,  120,  0.07,  45.,  YES,
,  130,  0.06,  90.,  YES,  120,  0.05,  60.,  YES,
,  110,  0.02,   0.,  YES
```

Listing 24-15. 3-Dimensional Composite Cantilever Plate Model Input File. (Continued)

```
$
$ LAMINA MATERIAL PROPERTIES.
$
MAT12,  110,  10.E+6,  10.E+6,  1.E+4,  0.1,  0.,  0.,
,  1.E+6,  1.E+6,  1.E+6,
,  ,  ,  ,  ,  60.E+3,  60.E+3,  10.E+3,
,  5.E+3,  4.E+3,  4.E+3,  ,  50.E+3,  50.E+3,  10.E+3
MAT12,  120,  10.E+6,  1.E+6,  1.E+4,  0.2,  0.,  0.,
,  4.E+6,  2.E+6,  2.E+6,
,  ,  ,  ,  ,  40.E+3,  4.E+3,  1.E+3,
,  5.E+3,  4.E+3,  4.E+3,  ,  80.E+3,  8.E+3,  2.E+3
MAT12,  130,  10.E+6,  2.E+6,  1.E+4,  0.1,  0.,  0.,
,  3.E+6,  1.5E+6,  1.5E+6,
,  ,  ,  ,  ,  20.E+3,  15.E+3,  1.E+3,
,  4.E+3,  3.E+3,  3.E+3,  ,  10.E+3,  10.E+3,  1.E+3
$
$ FIXED BOUNDARY CONDITION AT ONE END.
$
SPC1,  1,  123456,  10,  11,  12,  22,  23,  24
$
$ SURFACE PRESSURE LOADING.
$
PLOAD4,  1,   7,  10.,  ,  ,  ,  10,  2
PLOAD4,  1,   8,  10.,  ,  ,  ,   1,  8
PLOAD4,  1,   9,  10.,  ,  ,  ,   7,  5
PLOAD4,  1,  10,  10.,  ,  ,  ,  11,  3
PLOAD4,  1,  11,  10.,  ,  ,  ,   2,  9
PLOAD4,  1,  12,  10.,  ,  ,  ,   8,  6
ENDDATA
```

The plate is loaded with a 10 psi surface pressure which results in bending about the model y-axis. The results are similar to the 2-dimensional example in the previous section (Figure 24-3) including the slight twisting x-rotation due to the unbalanced lay-up. The displacements are shown in Listing 24-16. The element stresses are shown in Listing 24-17. The element failure indexes are shown in Listing 24-18 and the strength ratios in Listing 24-19 (obtained by adding `PARAM`, `STRENGTHRATIO`, `ON` to the Case Control Section of the Model Input File). Note that ply 1-direction and 2-direction stress and strain output is taken at the ply center while interlaminar and 3-direction stress and strain output is taken at the ply transition.

Listing 24-16. 3-Dimensional Composite Plate Displacements.

```
                                       D I S P L A C E M E N T   V E C T O R

     GRID      COORDINATE         T1            T2            T3            R1            R2            R3
      ID           ID
       1           0      2.647045E-03 -7.963247E-05  2.551758E-02  0.000000E+00  0.000000E+00  0.000000E+00
       2           0      2.842610E-03 -1.250115E-04  2.758905E-02  0.000000E+00  0.000000E+00  0.000000E+00
       3           0      3.037259E-03 -1.726977E-04  2.961260E-02  0.000000E+00  0.000000E+00  0.000000E+00
       4           0      4.183717E-03  3.294545E-04  1.658251E-01  0.000000E+00  0.000000E+00  0.000000E+00
       5           0      4.226151E-03  3.321596E-04  1.720394E-01  0.000000E+00  0.000000E+00  0.000000E+00
       6           0      4.280705E-03  3.352250E-04  1.782917E-01  0.000000E+00  0.000000E+00  0.000000E+00
       7           0      3.883498E-03  1.424097E-04  8.831843E-02  0.000000E+00  0.000000E+00  0.000000E+00
       8           0      3.984041E-03  1.418669E-04  9.316656E-02  0.000000E+00  0.000000E+00  0.000000E+00
       9           0      4.077350E-03  1.453106E-04  9.803680E-02  0.000000E+00  0.000000E+00  0.000000E+00
      13           0     -3.694612E-03 -6.092479E-04  2.535928E-02  0.000000E+00  0.000000E+00  0.000000E+00
      14           0     -3.968037E-03 -6.423821E-04  2.743004E-02  0.000000E+00  0.000000E+00  0.000000E+00
      15           0     -4.246207E-03 -6.767892E-04  2.945430E-02  0.000000E+00  0.000000E+00  0.000000E+00
      16           0     -5.848107E-03 -1.218392E-03  1.656952E-01  0.000000E+00  0.000000E+00  0.000000E+00
      17           0     -5.924244E-03 -1.224177E-03  1.719098E-01  0.000000E+00  0.000000E+00  0.000000E+00
      18           0     -5.986584E-03 -1.231361E-03  1.781617E-01  0.000000E+00  0.000000E+00  0.000000E+00
      19           0     -5.439550E-03 -1.066713E-03  8.820406E-02  0.000000E+00  0.000000E+00  0.000000E+00
      20           0     -5.570186E-03 -1.071960E-03  9.305112E-02  0.000000E+00  0.000000E+00  0.000000E+00
      21           0     -5.710591E-03 -1.075929E-03  9.792051E-02  0.000000E+00  0.000000E+00  0.000000E+00
```

Listing 24-17. 3-Dimensional Composite Plate Stresses.

```
               S T R E S S E S   I N   C O M P O S I T E   H E X   E L E M E N T S   I N   V O L U M E   0

   VOLUME COORDINATE ID = MATERIAL

     ELEMENT   PLY  STRESSES IN FIBER AND MATRIX DIRECTIONS          INTER-LAMINAR STRESSES            HENCKY
       ID      ID      NORMAL-1     NORMAL-2     SHEAR-12      NORMAL-3  SHEAR XZ-MAT SHEAR YZ-MAT    VON MISES

        7       1   1.05192E+04  8.31145E+02 -3.00295E+00  -3.17302E+00  9.15680E+01 -1.23558E+01   1.01323E+04
                2   1.79817E+03  9.18431E+01  1.10030E+03  -3.17302E+00  1.70297E+02 -2.29792E+01   2.60962E+03
                3  -3.10869E+02 -1.20708E+03  7.53370E+02  -3.17302E+00  1.19469E+02 -1.61207E+01   1.70968E+03
                4  -2.12105E+03 -1.19571E+03 -3.80401E+03  -3.17302E+00  1.50122E+02 -2.02568E+01   6.84590E+03
                5  -1.80386E+04 -1.97198E+03 -4.29770E+02   0.00000E+00  0.00000E+00  0.00000E+00   1.71527E+04

        8       1   4.53625E+03  2.27785E+02  2.00876E+02  -5.47106E+00  8.21190E+01 -1.14378E+01   4.44578E+03
                2   2.72493E+02  6.56837E+01  5.18110E+02  -5.47106E+00  1.52724E+02 -2.12718E+01   9.71943E+02
                3  -2.60364E+02 -5.33756E+02  4.38484E+02  -5.47106E+00  1.07141E+02 -1.49229E+01   9.07621E+02
                4  -4.18219E+02 -5.77102E+02 -1.33968E+03  -5.47106E+00  1.34631E+02 -1.87518E+01   2.38837E+03
                5  -7.89350E+03 -9.82765E+02 -3.96028E+02   0.00000E+00  0.00000E+00  0.00000E+00   7.48005E+03

        9       1   1.04041E+03  1.08255E+02  1.31305E+02  -4.89408E+00  3.42376E+01  4.19288E+00   1.02111E+03
                2  -1.18080E+02  3.36187E+01  1.03295E+02  -4.89408E+00  6.36748E+01  7.79788E+00   2.52898E+02
                3  -4.08855E+01 -1.21137E+02  1.23861E+02  -4.89408E+00  4.46701E+01  5.47049E+00   2.51300E+02
                4   1.15620E+02 -1.48479E+02 -2.07575E+02  -4.89408E+00  5.61312E+01  6.87407E+00   4.38059E+02
                5  -1.79929E+03 -2.32451E+02 -1.65553E+02   0.00000E+00  0.00000E+00  0.00000E+00   1.71694E+03

       10       1   1.12656E+04  8.94770E+02 -2.62802E+01  -3.17313E+00  1.80170E+02 -1.02822E+01   1.08526E+04
                2   1.99202E+03  9.28781E+01  1.17421E+03  -3.17313E+00  3.35078E+02 -1.91228E+01   2.88073E+03
                3  -3.28080E+02 -1.29486E+03  8.16504E+02  -3.17313E+00  2.35069E+02 -1.34153E+01   1.87946E+03
                4  -2.29524E+03 -1.27970E+03 -4.09224E+03  -3.17313E+00  2.95381E+02 -1.68573E+01   7.38074E+03
                5  -1.93370E+04 -2.10840E+03 -4.49207E+02   0.00000E+00  0.00000E+00  0.00000E+00   1.83902E+04

       11       1   4.16158E+03  1.97879E+02  2.19534E+02  -5.49048E+00  8.09235E+01 -1.04549E+01   4.08949E+03
                2   1.54620E+02  6.71452E+01  4.81680E+02  -5.49048E+00  1.50501E+02 -1.94439E+01   8.88662E+02
                3  -2.52485E+02 -4.88705E+02  4.01400E+02  -5.49048E+00  1.05582E+02 -1.36406E+01   8.33612E+02
                4  -3.28405E+02 -5.34607E+02 -1.19077E+03  -5.49048E+00  1.32671E+02 -1.71404E+01   2.12702E+03
                5  -7.23316E+03 -9.16371E+02 -3.88201E+02   0.00000E+00  0.00000E+00  0.00000E+00   6.85205E+03

       12       1   8.55711E+02  1.09286E+02  1.34257E+02  -4.91361E+00  2.01099E+01  3.23881E+00   8.43099E+02
                2  -1.47041E+02  3.26150E+01  7.87817E+01  -4.91361E+00  3.74002E+01  6.02350E+00   2.24049E+02
                3  -3.04946E+01 -1.00532E+02  1.18133E+02  -4.91361E+00  2.62376E+01  4.22569E+00   2.26903E+02
                4   1.72737E+02 -1.29511E+02 -1.32619E+02  -4.91361E+00  3.29694E+01  5.30989E+00   3.54309E+02
                5  -1.48767E+03 -2.01001E+02 -1.64394E+02   0.00000E+00  0.00000E+00  0.00000E+00   1.42414E+03
```

Listing 24-18. 3-Dimensional Composite Plate Failure Indexes.

```
           F A I L U R E   I N D E X E S   F O R   C O M P O S I T E   H E X   E L E M E N T S   O N   V O L U M E   0

     ELEMENT    FAILURE   PLY    MATRIX FAILURE   FAILURE      FIBER FAILURE    FAILURE       BOND FAILURE    ELEMENT FAILURE
       ID       THEORY    ID         INDEX         MODE            INDEX         MODE            INDEX             INDEX

         7       HILL       1     1.14396E-01                  1.14396E-01
                                                                                              1.83136E-02
                            2     5.27927E-02                  5.27927E-02
                                                                                              3.40595E-02
                            3     2.76366E-02                  2.76366E-02
                                                                                              2.38939E-02
                            4     6.05851E-01                  6.05851E-01
                                                                                              3.00244E-02
                            5     4.66360E-01                  4.66360E-01                                    6.05851E-01

         8       HILL       1     1.77938E-02                  1.77938E-02
                                                                                              1.64240E-02
                            2     1.24333E-02                  1.24333E-02
                                                                                              3.05452E-02
                            3     9.79860E-03                  9.79860E-03
                                                                                              2.14285E-02
                            4     7.53143E-02                  7.53143E-02
                                                                                              2.69265E-02
                            5     1.02951E-01                  1.02951E-01                                    1.02951E-01

         9       HILL       1     2.18811E-03                  2.18811E-03
                                                                                              6.84733E-03
                            2     6.48890E-04                  6.48890E-04
                                                                                              1.27346E-02
                            3     7.84648E-04                  7.84648E-04
                                                                                              8.93377E-03
                            4     1.77434E-03                  1.77434E-03
                                                                                              1.12259E-02
                            5     6.26075E-03                  6.26075E-03                                    1.27346E-02

        10       HILL       1     1.33126E-01                  1.33126E-01
                                                                                              3.60339E-02
                            2     6.51094E-02                  6.51094E-02
                                                                                              6.70155E-02
                            3     3.47027E-02                  3.47027E-02
                                                                                              4.70137E-02
                            4     7.04999E-01                  7.04999E-01
                                                                                              5.90762E-02
                            5     5.34503E-01                  5.34503E-01                                    7.04999E-01

        11       HILL       1     1.51828E-02                  1.51828E-02
                                                                                              1.61845E-02
                            2     1.08383E-02                  1.08383E-02
                                                                                              3.00998E-02
                            3     8.36903E-03                  8.36903E-03
                                                                                              2.11160E-02
                            4     5.95614E-02                  5.95614E-02
                                                                                              2.65339E-02
                            5     8.82715E-02                  8.82715E-02                                    8.82715E-02

        12       HILL       1     1.88451E-03                  1.88451E-03
                                                                                              4.02217E-03
                            2     2.96709E-04                  2.96709E-04
                                                                                              7.48039E-03
                            3     6.28783E-04                  6.28783E-04
                                                                                              5.24775E-03
                            4     5.81843E-04                  5.81843E-04
                                                                                              6.59419E-03
                            5     4.73347E-03                  4.73347E-03                                    7.48039E-03
```

Listing 24-19. 3-Dimensional Composite Plate Strength Ratios.

```
          S T R E N G T H   R A T I O S   F O R   C O M P O S I T E   H E X   E L E M E N T S   O N   V O L U M E   0

    ELEMENT    FAILURE   PLY   MATRIX STRENGTH   FAILURE      FIBER STRENGTH   FAILURE      BOND STRENGTH    ELEMENT STRENGTH
      ID       THEORY    ID         RATIO         MODE            RATIO          MODE            RATIO               RATIO

        7       HILL       1     2.95661E+00                   2.95661E+00
                                                                                             5.46042E+01
                           2     4.35224E+00                   4.35224E+00
                                                                                             2.93604E+01
                           3     6.01531E+00                   6.01531E+00
                                                                                             4.18517E+01
                           4     1.28474E+00                   1.28474E+00
                                                                                             3.33062E+01
                           5     1.46433E+00                   1.46433E+00                                     1.28474E+00

        8       HILL       1     7.49663E+00                   7.49663E+00
                                                                                             6.08865E+01
                           2     8.96823E+00                   8.96823E+00
                                                                                             3.27384E+01
                           3     1.01022E+01                   1.01022E+01
                                                                                             4.66668E+01
                           4     3.64386E+00                   3.64386E+00
                                                                                             3.71381E+01
                           5     3.11662E+00                   3.11662E+00                                     3.11662E+00

        9       HILL       1     2.13779E+01                   2.13779E+01
                                                                                             1.46042E+02
                           2     3.92568E+01                   3.92568E+01
                                                                                             7.85262E+01
                           3     3.56995E+01                   3.56995E+01
                                                                                             1.11935E+02
                           4     2.37400E+01                   2.37400E+01
                                                                                             8.90794E+01
                           5     1.26383E+01                   1.26383E+01                                     1.26383E+01

       10       HILL       1     2.74074E+00                   2.74074E+00
                                                                                             2.77516E+01
                           2     3.91903E+00                   3.91903E+00
                                                                                             1.49219E+01
                           3     5.36807E+00                   5.36807E+00
                                                                                             2.12704E+01
                           4     1.19098E+00                   1.19098E+00
                                                                                             1.69273E+01
                           5     1.36781E+00                   1.36781E+00                                     1.19098E+00

       11       HILL       1     8.11566E+00                   8.11566E+00
                                                                                             6.17875E+01
                           2     9.60549E+00                   9.60549E+00
                                                                                             3.32228E+01
                           3     1.09311E+01                   1.09311E+01
                                                                                             4.73573E+01
                           4     4.09749E+00                   4.09749E+00
                                                                                             3.76877E+01
                           5     3.36581E+00                   3.36581E+00                                     3.36581E+00

       12       HILL       1     2.30357E+01                   2.30357E+01
                                                                                             2.48622E+02
                           2     5.80543E+01                   5.80543E+01
                                                                                             1.33683E+02
                           3     3.98795E+01                   3.98795E+01
                                                                                             1.90558E+02
                           4     4.14569E+01                   4.14569E+01
                                                                                             1.51649E+02
                           5     1.45348E+01                   1.45348E+01                                     1.45348E+01
```

24.6 Using Rigid and Interpolation Elements

Autodesk Inventor Nastran contains a wide variety of rigid and interpolation elements. The rigid elements are referenced using the `RROD`, `RBAR`, `RTRPLT`, and `RBE2` Bulk Data entries. The interpolation elements are referenced using the `RBE3` and `RSPLINE` Bulk Data entries. The general form of the rigid element is the `RBE2`, which provides a rigid connection between independent degrees of freedom at a single grid point and corresponding dependent degrees of freedom at multiple grid points. The `RBE3` element is a linear interpolation element often used to distribute loading or mass at a single reference point to several non-collinear averaged points. The `RSPLINE` element uses beam equations to interpolate displacements along a curve and is normally used to model mesh transitions.

All rigid and interpolation elements are reduced to multipoint constraint equations (`MPC`). The simplest description of an `MPC` equation is that used to describe the motion of one (dependent) degree of freedom in a model as the linear combination of the motions of one or more (dependent) degrees of freedom.

We will now look at two examples comparing the differences between the `RBE2` and `RBE3` entry. For each example we will use the cantilever beam shown in Figure 24-4.

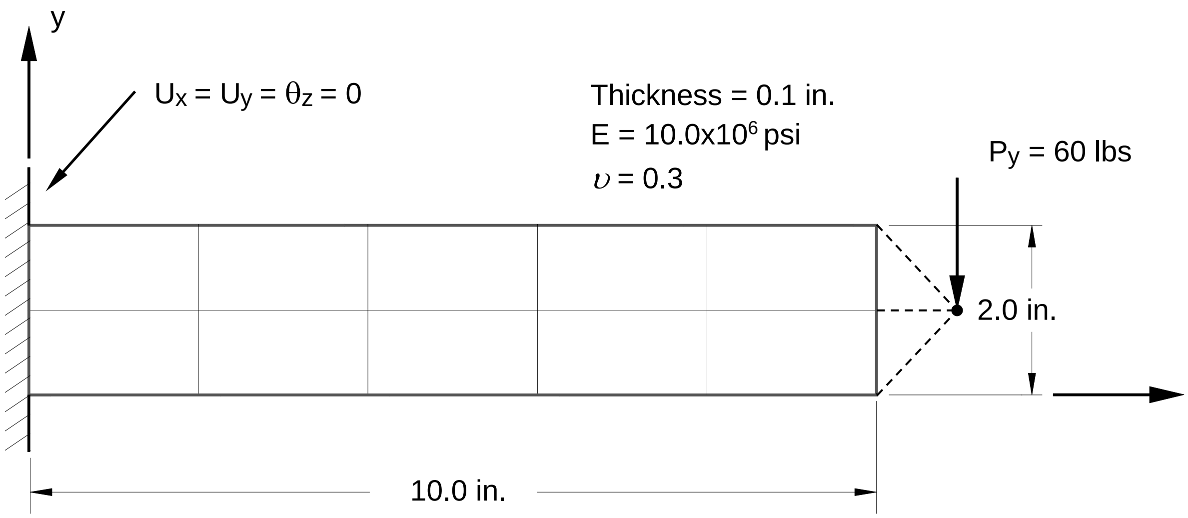


Figure 24-4. Cantilever Beam Example Problem.

In the first example the beam is point loaded at the free end through a rigid connection from a single point. For the `RBE2` element only one point can be defined as independent. This point is the grid where the load is applied (grid point 19). The dependent grid points are chosen to be the three nodes at the end of the beam (grid points 1, 2, and 3). Component numbers for the independent degrees of freedom must also be specified. Since in this example we are only concerned with motion in the xy-plane, components 1, 2, 6 (x-translation, y-translation, and z-rotation) are chosen. The Model Input File is shown in Listing 24-20.

Listing 24-20. Model Input File for the Rigid Element Example Problem.

```
$
$ STATIC SOLUTION.
$
SOL STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = 2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH
$
DISPLACEMENT = ALL
ELSTRESS = ALL
MPCFORCES = ALL
$
SPC = 1
SUBCASE 1
 LABEL = 60 LB POINT LOAD IN Y-DIRECTION AT RIGID ELEMENT INDEPENDENT GRID POINT
 LOAD = 2

BEGIN BULK
$
$ GEOMETRY DEFINITION (10" X 2" RECTANGULAR FLAT PLATE WITH A 5 X 2 MESH).
$
GRID,   1,  ,  10.,  0.,  0.
GRID,   2,  ,  10.,  1.,  0.
GRID,   3,  ,  10.,  2.,  0.
GRID,   4,  ,   2.,  0.,  0.
GRID,   5,  ,   2.,  1.,  0.
GRID,   6,  ,   2.,  2.,  0.
GRID,   7,  ,   6.,  0.,  0.
GRID,   8,  ,   6.,  1.,  0.
GRID,   9,  ,   6.,  2.,  0.
GRID,  10,  ,   4.,  0.,  0.
GRID,  11,  ,   4.,  1.,  0.
GRID,  12,  ,   4.,  2.,  0.
GRID,  13,  ,   8.,  0.,  0.
GRID,  14,  ,   8.,  1.,  0.
GRID,  15,  ,   8.,  2.,  0.
GRID,  16,  ,   0.,  0.,  0.
GRID,  17,  ,   0.,  1.,  0.
GRID,  18,  ,   0.,  2.,  0.
GRID,  19,  ,  11.,  1.,  0.
$
$ FLAT PLATE MODELED WITH SHELL ELEMENTS.
$
CQUADR,   1,  10,  16,   4,   5,  17
CQUADR,   2,  10,   4,  10,  11,   5
CQUADR,   3,  10,  10,   7,   8,  11
CQUADR,   4,  10,   7,  13,  14,   8
CQUADR,   5,  10,  13,   1,   2,  14
CQUADR,   6,  10,  17,   5,   6,  18
CQUADR,   7,  10,   5,  11,  12,   6
CQUADR,   8,  10,  11,   8,   9,  12
CQUADR,   9,  10,   8,  14,  15,   9
CQUADR,  10,  10,  14,   2,   3,  15
$
$ RIGID ELEMENT CONNECTION.
$
RBE2,  11,  19,  126,  1,  2,  3
$
$ ELEMENT MATERIAL AND THICKNESS (0.1").
$
PSHELL,  10,  100,  0.1,  100,  ,  100
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
$
$ FIXED BOUNDARY CONDITION AT ONE END.
$
SPC1,  1,  123456,  16,  17,  18
$
$ POINT LOAD ON FREE END (Y-DIRECTION) AT INDEPENDENT GRID POINT.
$
FORCE,  2,  19,  0,  60.,  0.,  1.,  0.
ENDDATA
```

The `RBE2` element generates `MPC` equations for the motion of the independent grid points as a function of the motion of the dependent grid point. The equivalent `MPC` equations used are written to the Model Output File and the Bulk Data File (Listing 24-21).

Listing 24-21. Generated `MPC` Bulk Data Entries Written to the Bulk Data Output File.

```
MPC            0       1       1 1.00000      19       1-1.00000        +C     3A
+C    3A              19       6-1.00000
MPC            0       1       2 1.00000      19       2-1.00000        +C     4A
+C    4A              19       6 1.00000
MPC            0       1       6 1.00000      19       6-1.00000
MPC            0       2       1 1.00000      19       1-1.00000
MPC            0       2       2 1.00000      19       2-1.00000        +C     5A
+C    5A              19       6 1.00000
MPC            0       2       6 1.00000      19       6-1.00000
MPC            0       3       1 1.00000      19       1-1.00000        +C     6A
+C    6A              19       6 1.00000
MPC            0       3       2 1.00000      19       2-1.00000        +C     7A
+C    7A              19       6 1.00000
MPC            0       3       6 1.00000      19       6-1.00000
```

In this example the beam is subjected to a 60 pound point load at the independent grid point resulting in deflections in the x and y directions. Note that because the `RBE2` element creates a rigid connection, stiffness is added to the model and all nodes move as a rigid body. The rigid element displacements from the Model Output File are shown in Listing 24-22 and as expected display rigid body motion.

Listing 24-22. Rigid Element Displacements.

```
                                         D I S P L A C E M E N T   V E C T O R

     GRID       COORDINATE         T1              T2              T3              R1              R2              R3
      ID            ID
       1             0       5.359975E-03   3.479293E-02   0.000000E+00   0.000000E+00   0.000000E+00   5.359975E-03
       2             0       0.000000E+00   3.479293E-02   0.000000E+00   0.000000E+00   0.000000E+00   5.359975E-03
       3             0      -5.359975E-03   3.479293E-02   0.000000E+00   0.000000E+00   0.000000E+00   5.359975E-03
      19             0       0.000000E+00   4.015290E-02   0.000000E+00   0.000000E+00   0.000000E+00   5.359975E-03
```

The forces of multipoint constraint from the Model Output File are shown in Listing 24-23. These are the internal forces generated to enforce rigid body motion. Note that these forces are in equilibrium.

Listing 24-23. Rigid Element Forces of Multipoint Constraint.

```
                                  F O R C E S   O F   M U L T I P O I N T   C O N S T R A I N T

     GRID      COORDINATE        T1             T2             T3             R1             R2             R3
      ID           ID
       1           0      2.488690E+01  2.285701E+01  0.000000E+00  0.000000E+00  0.000000E+00 -6.327615E+00
       2           0     -7.389644E-13  1.428599E+01  0.000000E+00  0.000000E+00  0.000000E+00  2.288144E+01
       3           0     -2.488690E+01  2.285701E+01  0.000000E+00  0.000000E+00  0.000000E+00 -6.327615E+00
      19           0      0.000000E+00 -6.000000E+01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
```

Interpolation elements behave differently than rigid elements and do not add stiffness to the model or force rigid body motion. As an example we replace the `RBE2` rigid element with an `RBE3` interpolation element. The `RBE3` element is very versatile and has many uses. The most common use is to transmit forces and moments from a reference point to several averaged points. In our example the load is applied to the single reference point (grid point 19) and the grid points to be averaged are the three at the beam end (1, 2, and 3). Component numbers for the reference degrees of freedom must also be specified. Since in this example we are only concerned with motion in the xy-plane, components 1, 2, 6 (x-translation, y-translation, and z-rotation) are chosen. Additionally, we must specify weighting factors and components for the averaged grid points. Normally the weighting factors should be set to 1.0. The component numbers determine in what directions the averaged points can react loads. Since our problem is limited to the xy-plane, components 1 and 2 are chosen. The Model Input File is shown in Listing 24-24.

Listing 24-24. Model Input File for the Interpolation Element Example Problem.

```
$
$ STATIC SOLUTION.
$
SOL STATIC
$
TITLE = INSTALLATION TEST CASE
SUBTITLE = 2-D CANTILEVER BEAM -QUADR ELEMENTS -2X5 MESH
$
DISPLACEMENT = ALL
ELSTRESS = ALL
MPCFORCES = ALL
$
SPC = 1
SUBCASE 1
 LABEL = 60 LB POINT LOAD IN Y-DIRECTION AT INTERPOLATION ELEMENT REFERENCE GRID POINT
 LOAD = 2

BEGIN BULK
$
$ GEOMETRY DEFINITION (10" X 2" RECTANGULAR FLAT PLATE WITH A 5 X 2 MESH).
$
GRID,   1,  ,  10.,  0.,  0.
GRID,   2,  ,  10.,  1.,  0.
GRID,   3,  ,  10.,  2.,  0.
GRID,   4,  ,   2.,  0.,  0.
GRID,   5,  ,   2.,  1.,  0.
GRID,   6,  ,   2.,  2.,  0.
GRID,   7,  ,   6.,  0.,  0.
GRID,   8,  ,   6.,  1.,  0.
GRID,   9,  ,   6.,  2.,  0.
GRID,  10,  ,   4.,  0.,  0.
GRID,  11,  ,   4.,  1.,  0.
GRID,  12,  ,   4.,  2.,  0.
GRID,  13,  ,   8.,  0.,  0.
GRID,  14,  ,   8.,  1.,  0.
GRID,  15,  ,   8.,  2.,  0.
GRID,  16,  ,   0.,  0.,  0.
GRID,  17,  ,   0.,  1.,  0.
GRID,  18,  ,   0.,  2.,  0.
GRID,  19,  ,  11.,  1.,  0.
$
$ FLAT PLATE MODELED WITH SHELL ELEMENTS.
$
CQUADR,   1,  10,  16,   4,   5,  17
CQUADR,   2,  10,   4,  10,  11,   5
CQUADR,   3,  10,  10,   7,   8,  11
CQUADR,   4,  10,   7,  13,  14,   8
CQUADR,   5,  10,  13,   1,   2,  14
CQUADR,   6,  10,  17,   5,   6,  18
CQUADR,   7,  10,   5,  11,  12,   6
CQUADR,   8,  10,  11,   8,   9,  12
CQUADR,   9,  10,   8,  14,  15,   9
CQUADR,  10,  10,  14,   2,   3,  15
$
$ INTERPOLATION ELEMENT CONNECTION.
$
RBE3,  11,  ,  19,  126,  1.,  12,  1,  2,
,  3
$
$ ELEMENT MATERIAL AND THICKNESS (0.1").
$
PSHELL,  10,  100,  0.1,  100,  ,  100
$
$ ELEMENT MATERIAL PROPERTIES (ALUMINUM).
$
MAT1,  100,  1.E+7,  ,  0.33,  0.1
```

Listing 24-24. Model Input File for the Interpolation Element Example Problem. (Continued)

```
$
$ FIXED BOUNDARY CONDITION AT ONE END.
$
SPC1,  1,  123456,  16,  17,  18
$
$ POINT LOAD ON FREE END (Y-DIRECTION) AT REFERENCE GRID POINT.
$
FORCE,  2,  19,  0,  60.,  0.,  1.,  0.
ENDDATA
```

The `RBE3` element generates `MPC` equations for the motion of one or more grid points as a function of the motion of other connected points. The equivalent `MPC` equations used are written to the Model Output File and the Bulk Data File (Listing 24-25).

Listing 24-25. Generated `MPC` Bulk Data Entries Written to the Bulk Data Output File.

```
MPC            0      19       1 1.00000       1       1-0.33333        +C    3A
+C    3A               2       1-0.33333       3       1-0.33333
MPC            0      19       2 1.00000       1       1-0.50000        +C    4A
+C    4A               1       2-0.33333       2       2-0.33333        +C    5A
+C    5A               3       1 0.50000       3       2-0.33333
MPC            0      19       6 1.00000       1       1-0.50000        +C    6A
+C    6A               3       1 0.50000
```

As with the previous example the beam is subjected to a 60 pound point load at the reference grid point resulting in deflections in the x and y directions. Note that the `RBE3` element does not force rigid body motion and stiffness is not added to the model. The interpolation element displacements are shown in Listing 24-26. As expected the `RBE3` element rotations about the z-direction differ, unlike the `RBE2` example where they were all the same (rigid body motion).

Listing 24-26. Interpolation Element Displacements.

```
                                       D I S P L A C E M E N T   V E C T O R

      GRID      COORDINATE        T1            T2            T3            R1            R2            R3
       ID          ID
        1           0      5.454999E-03  3.499772E-02  0.000000E+00  0.000000E+00  0.000000E+00  5.341274E-03
        2           0      0.000000E+00  3.489725E-02  0.000000E+00  0.000000E+00  0.000000E+00  4.668649E-03
        3           0     -5.454999E-03  3.499772E-02  0.000000E+00  0.000000E+00  0.000000E+00  5.341274E-03
       19           0      0.000000E+00  4.041923E-02  0.000000E+00  0.000000E+00  0.000000E+00  5.454999E-03
```

The forces of multipoint constraint from the Model Output File are shown in Listing 24-27. These are the internal forces generated at the connected grid points that are in equilibrium with the load(s) applied at the reference point. Unlike rigid elements, they are a direct result of the magnitude and direction of the applied loads and the interpolation element geometry, not the adjoining stiffness. Note the difference between these forces and those in Listing 24-23.

Listing 24-27. Interpolation Element Forces of Multipoint Constraint.

```
                                   F O R C E S   O F   M U L T I P O I N T   C O N S T R A I N T

      GRID       COORDINATE         T1            T2            T3            R1            R2            R3
       ID           ID
        1           0      3.000000E+01  2.000000E+01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        2           0      3.410605E-12  2.000000E+01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        3           0     -3.000000E+01  2.000000E+01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
       19           0      0.000000E+00 -6.000000E+01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
```

Last, a discussion of common errors using rigid and interpolation elements is necessary. The most common error is the application of additional constraints through either single point or multipoint constraints or other rigid or interpolation elements. A dependent degree of freedom assigned by one element cannot be assigned dependent by another or by an `MPC`. For example, you cannot specify a reference degree of freedom on a `RBE3` as the dependent degree of freedom on an `RBE2`. Additionally, you cannot reference dependent degrees of freedom on `RBE2` or `RBE3` entries on single point constraint entries. This error occurs most often when using symmetric boundary conditions and rigid elements together. The solution is to specify the boundary conditions on the independent degrees of freedom and not the dependent ones.

24.7 Sparse Solver Operation

Operation of the sparse direct and iterative solvers is discussed in Table 24-8. If configured correctly, these solvers can dramatically increase performance over the older blocked profile solvers. The sparse direct solver requires more physical memory for large models than the blocked profile solvers did. It is recommended that if you intend to run large, complex models, you upgrade your computers physical memory to the maximum allowed. Table 24-9 shows which solvers are available in each solution.

Table 24-8 Sparse Solver Operation.

Solver	`DECOMPMETHOD` Directive Setting	Associated Directives and Parameters	Remarks
Parallel Sparse Iterative	PCGLSS	`MAXSPARSEITER`, `MINSPARSEITER`, `SPARSEITERTOL`, `SPARSEITERMODE`, `SPARSEITERMETHOD`, `NPROCESSORS`	This is a parallel, multi-mode solver which uses either an iterative (Preconditioned Conjugate Gradient) or direct solution technique. Its advanced preconditioner is optimized for all element types and is available in all linear and nonlinear static solutions. The iterative solver mode requires less physical memory and disk storage than the direct mode and other direct solvers (VSS). It is especially effective for large models comprised mainly of parabolic tetrahedron elements (`CTETRA`). The `MAXSPARSEITER` model parameter controls the maximum number of iterations permitted. The `MINSPARSEITER` model parameter controls the minimum number of iterations that will be performed regardless of convergence. The `SPARSEITERTOL` parameter controls the convergence tolerance. Solution accuracy and performance can be controlled with these parameters. Note the `RAM` directive setting has no effect on performance for this solver in the iterative mode. The direct solver requires more memory but is less sensitive to element initial distortion and may provide a faster solution when sufficient memory is available. Like the VSS solver it has an out of core mode that is controlled by the `RAM` directive. Both modes support multiple processors using the `NPROCESSORS` directive. If the `PCGLSS` solver is selected for a nonlinear transient solution, the `VSS` solver will be used. This solver is recommended for large problems and will generally be faster than the `VSS` solver.
Sparse Direct	VSS	`SPARSEMETHOD`, `RAM`, `RESEQGRIDMETHOD`	This solver is optimized for all element types and solutions. Performance is dependent on available physical memory and can be adversely affected by the `RAM` directive setting. The solver has two modes of operation: in core and out of core. If the factored stiffness matrix size, as determined by the solver, exceeds the available system memory, as set by the `RAM` directive, an out of core solution will be used. An in core solution is usually faster than an out of core solution, even if some virtual memory is used. To force an in core solution, increase the `RAM` directive value. If an out of core solution is performed, the solver will state "OUT OF CORE SOLUTION USED". Significant performance degradation can occur if physical memory is limited regardless of which mode is used. The PCGLSS solver will usually be faster for these types of problems since it requires less memory.

Table 24-8 Sparse Solver Operation. (Continued)

Solver	`DECOMPMETHOD` Directive Setting	Associated Directives and Parameters	Remarks
Sparse Iterative	VIS	`MAXSPARSEITER`, `SPARSEITERTOL`	This solver uses an iterative solution technique (Preconditioned Conjugate Gradient) which requires less physical memory and disk storage than the direct solvers. It is optimized for models consisting of mostly solid elements running non-eigenvalue solutions or all element types in heat transfer solutions. It is not recommended for nonlinear solutions where material nonlinearity is specified or for models with shell and/or line element types. If the `VIS` solver is used for these types of problems, convergence may be slow or the solution may diverge. The `MAXSPARSEITER` model parameter controls the maximum number of iterations permitted. The `SPARSEITERTOL` parameter controls the convergence tolerance. Solution accuracy and performance can be controlled with these parameters. This solver can be very useful when other solvers fail because it does not require the matrix to be nonsingular or positive definite. For ill-conditioned problems it is recommended that `MAXSPARSEITER` be set to 500-1000 and that the solution error measure, epsilon, be checked. Note the `RAM` directive setting has no effect of performance for this solver.
Parallel Sparse Direct	PSS	`RAM`, `RESEQGRIDMETHOD`, `NPROCESSORS`	This is a parallel solver and is optimized for all element types and solutions. Performance is dependent on available physical memory and can be adversely affected by the RAM directive setting. The solver has two modes of operation: in core and out of core. If the factored stiffness matrix size, as determined by the solver, exceeds the available system memory, as set by the RAM directive, an out of core solution will be used. An in core solution is usually faster than an out of core solution, even if some virtual memory is used. To force an in core solution, increase the RAM directive value. If an out of core solution is performed, the solver will state "OUT OF CORE SOLUTION USED". Significant performance degradation can occur if physical memory is limited regardless of which mode is used. The PCGLSS solver may be faster for these types of problems since it requires less memory.

Table 24-9 Solution/Solver Applicability Matrix.

Solution		Solver			
Label	**Number**	**PCGLSS**	**VSS**	**VIS**	**PSS**
LINEAR STATIC or STEADY STATE HEAT TRANSFER	101	✓	✓	✓	✓
MODAL	103	✓	✓		✓
LINEAR BUCKLING	105	✓	✓	✓	✓
NONLINEAR STATIC	106	✓	✓	✓	✓
DIRECT FREQUENCY RESPONSE	108		✓	✓	✓
DIRECT TRANSIENT RESPONSE	109		✓	✓	✓
MODAL COMPLEX EIGENVALUE	110	✓	✓		✓
MODAL FREQUENCY RESPONSE	111	✓	✓		✓
MODAL TRANSIENT RESPONSE	112	✓	✓		✓
NONLINEAR TRANSIENT RESPONSE	129		✓	✓	✓
NONLINEAR STEADY STATE HEAT TRANSFER	153	✓	✓	✓	✓
NONLINEAR TRANSIENT HEAT TRANSFER	159		✓	✓	✓
NONLINEAR BUCKLING	180	✓	✓	✓	✓
PRESTRESS STATIC	181	✓	✓	✓	✓
LINEAR PRESTRESS MODAL	182	✓	✓	✓	✓
LINEAR PRESTRESS FREQUENCY RESPONSE	183	✓	✓	✓	✓
LINEAR PRESTRESS TRANSIENT RESPONSE	184	✓	✓	✓	✓
NONLINEAR PRESTRESS MODAL	185	✓	✓	✓	✓
NONLINEAR PRESTRESS FREQUENCY RESPONSE	186	✓	✓	✓	✓
NONLINEAR PRESTRESS TRANSIENT RESPONSE	187	✓	✓	✓	✓
LINEAR PRESTRESS COMPLEX EIGENVALUE	188	✓	✓	✓	✓
NONLINEAR PRESTRESS COMPLEX EIGENVALUE	189	✓	✓	✓	✓

24.8 Optimal Parameter Settings

A single model parameter (`OPTIMIZESETTINGS`) can be used to force certain model parameters and directives to optimize a solution for speed, accuracy, or a combination of both. Table 24-10 shows which parameters and directives are modified and the values set for the four different `OPTIMIZESETTINGS` options.

Table 24-10 `OPTIMIZESETTINGS` Parameter Matrix.

OPTIMIZESETTINGS	**NONE**	**ACCURACY**	**SPEED**	**BOTH**
ALIGNEDGENODE	OFF	ON	OFF	ON
AUTOFIXRIGIDELEM	OFF	ON	ON	ON
BAREQVLOAD	ON	ON	ON	ON
BISECT	ON	ON	OFF	ON
COUPMASS	AUTO	ON	OFF	AUTO
DECOMPMETHOD	AUTO	AUTO	AUTO	AUTO
ELEMGEOMCHECKS	ON	ON	OFF	ON
ENHCQUADRSLT	OFF	ON	OFF	ON
EXTRACTMETHOD	AUTO	AUTO	LANCZOS	AUTO
FREQRESPRSLTOUT	ON	ON	OFF	ON
HEXINODE	AUTO	ON	AUTO	OFF
MAXSPARSEITER	AUTO	AUTO	1000	AUTO
MODLDATAOUT	ON	ON	OFF	ON
NBEAMINTNODE	2	4	2	2
NLAYERS	6	12	6	9
NLINSOLACCEL	4	4	4	4
NLINSOLTOL	2	1	3	2
QUADEQVLOAD	OFF	ON	OFF	OFF
QUADINODE	AUTO	ON	AUTO	ON
QUADRNODE	OFF	ON	OFF	ON
QUADSECT	OFF	ON	OFF	ON
RANDRESPRSLTOUT	ON	ON	OFF	ON
SLINEMAXACTDIST	1.0E+30	1.0E+30	AUTO	1.0E+30
SPARSEITERMETHOD	AUTO	AUTO	AUTO	AUTO
SPARSEITERMODE	AUTO	AUTO	3	AUTO
SPARSEITERTOL	1.0E-05	1.0E-06	1.0E-04	1.0E-05
TRIEQVLOAD	OFF	ON	OFF	OFF
TRIRNODE	OFF	ON	ON	ON

APPENDIX A - OUTPUT FORMATS

Examples of most of the Model Results Output File formats that are available are contained in this section. Each example is annotated with comments and identified by page number and title. A summary of these figures is shown in the listing below, Listing A-1.

Listing A-1. List of Figures for Output Formats.

Figure A-1. Database Subcase Constraint and Load Set Definition. A-6
Figure A-2. Database Rectangular Coordinate System Definition. A-7
Figure A-3. Database Cylindrical Coordinate System Definition. A-8
Figure A-4. Database Spherical Coordinate System Definition. A-9
Figure A-5. Database Grid Point Definition. A-10
Figure A-6. Database Concentrated Mass Element Definition. A-11
Figure A-7. Database Gap Element Definition. A-12
Figure A-8. Database Spring Element Definition. A-13
Figure A-9. Database Rod Element Definition. A-14
Figure A-10. Database Bar Element Definition. A-15
Figure A-11. Database Beam Element Definition. A-16
Figure A-12. Database Quad Element Definition. A-17
Figure A-13. Database Tri Element Definition. A-18
Figure A-14. Database Shear Element Definition. A-19
Figure A-15. Database Hex Element Definition. A-20
Figure A-16. Database Pent Element Definition. A-21
Figure A-17. Database Tet Element Definition. A-22
Figure A-18. Database Gap Element Property Definition. A-23
Figure A-19. Database Spring Element Property Definition. A-24
Figure A-20. Database Rod Element Property Definition. A-25
Figure A-21. Database Bar Element Property Definition. A-26
Figure A-22. Database Beam Element Property Definition. A-27
Figure A-23. Database Composite Shell Element Property Definition. A-28
Figure A-24. Database Shell Element Property Definition. A-29
Figure A-25. Database Shear Element Property Definition. A-30
Figure A-26. Database Solid Element Material Property Definition. A-31
Figure A-27. Database Isotropic Material Definition. A-32
Figure A-28. Database Anisotropic Shell Element Material Definition. A-33
Figure A-29. Database Isotropic Material Definition (Heat Transfer Analysis). A-34
Figure A-30. Database Anisotropic Element Material Definition (Heat Transfer Analysis). A-35
Figure A-31. Database Orthotropic Material Definition. A-36
Figure A-32. Database Anisotropic Solid Element Material Definition. A-37
Figure A-33. Database Nonlinear Material Definition. A-38

Figure A-34. Database Isotropic Material Table Definition. A-39
Figure A-35. Database Anisotropic Shell Element Material Table Definition. A-40
Figure A-36. Database Isotropic Material Table Definition (Heat Transfer Analysis). A-41
Figure A-37. Database Orthotropic Material Table Definition (Heat Transfer Analysis). A-42
Figure A-38. Database Orthotropic Shell Element Material Table Definition. A-43
Figure A-39. Database Anisotropic Solid Element Material Table Definition. A-44
Figure A-40. Database Material Property Table Definition. A-45
Figure A-41. Database Dynamic Load Table Definition. A-46
Figure A-42. Database Modal Damping Table Definition. A-47
Figure A-43. Database Single Point Constraint Definition. A-48
Figure A-44. Database Single Point Constraint Addition Definition. A-49
Figure A-45. Database Multipoint Constraint Definition. A-50
Figure A-46. Database Multipoint Constraint Addition Definition. A-51
Figure A-47. Database Grid Point Force Vector Definition. A-52
Figure A-48. Database Grid Point Moment Vector Definition. A-53
Figure A-49. Database Gravity Load Vector Definition. A-54
Figure A-50. Database Centrifugal Load Vector Definition. A-55
Figure A-51. Database Grid Point Temperature Definition. A-56
Figure A-52. Database Dynamic Load Scale Factor Definition. A-57
Figure A-53. Database Dynamic Load Time Delay Definition. A-58
Figure A-54. Database Dynamic Load Phase Lead Definition. A-59
Figure A-55. Database Transient Time Step Definition. A-60
Figure A-56. Database Frequency List Definition. A-61
Figure A-57. Database Frequency List Generation Definition. A-62
Figure A-58. Database Element Initial Deformation Definition. A-63
Figure A-59. Database Bar Element Pressure Definition. A-64
Figure A-60. Database Shell Element Pressure Definition. A-65
Figure A-61. Database Solid Element Pressure Definition. A-66
Figure A-62. Database Bar Element Thermal Definition. A-67
Figure A-63. Database Shell Element Thermal Definition. A-68
Figure A-64. Database Static Load Addition Definition. A-69
Figure A-65. Database Transient Response Dynamic Load Definition. A-70
Figure A-66. Database Frequency Response Dynamic Load Definition. A-71
Figure A-67. Database Power Spectral Density Definition. A-72
Figure A-68. Database Power Spectral Density Definition. A-73
Figure A-69. Database Static Load Sequence Definition. A-74
Figure A-70. Database Grid Point Scalar Load Definition. A-75
Figure A-71. Database Line and Point Grid Point Boundary Heat Flux Definition. A-76
Figure A-72. Database Surface Grid Point Boundary Heat Flux Definition. A-77
Figure A-73. Database Element Boundary Heat Flux Definition. A-78
Figure A-74. Database Element Volume Heat Addition Definition. A-79
Figure A-75. Database Element Boundary Convection Definition. A-80
Figure A-76. Database Subcase Vector Output Set Definition. A-81

Figure A-77. Database Subcase Vector Output Set Definition (Heat Transfer Analysis). A-82
Figure A-78. Database Subcase Element Output Set Definition. A-83
Figure A-79. Database Subcase Element Output Set Definition (Heat Transfer Analysis). A-84
Figure A-80. Database Subcase Grid Output Set Definition. A-85
Figure A-81. Database Set Definition. A-86
Figure A-82. Database Volume Definition. A-87
Figure A-83. Database Surface Definition. A-88
Figure A-84. Database Spring Element Stress Recovery Property Definition. A-89
Figure A-85. Database Rod Element Stress Recovery Property Definition. A-90
Figure A-86. Database Bar Element Axis Output Location Definition. A-91
Figure A-87. Database Bar Element Stress Recovery Property Definition. A-92
Figure A-88. Database Beam Element Stress Recovery Property Definition. A-93
Figure A-89. Database Shell Element Stress Recovery Property Definition. A-94
Figure A-90. Database Result Limits Definition. A-95
Figure A-91. Database Model Parameter Definition. A-96
Figure A-92. Model Database Size. A-97
Figure A-93. Grid Point Resequencer Output. A-98
Figure A-94. Element Geometry Statistics. A-99
Figure A-95. Global Stiffness Matrix Assembly Statistics. A-100
Figure A-96. Global Mass Matrix Assembly Statistics. A-101
Figure A-97. Global Stiffness Matrix Factorization Statistics. A-102
Figure A-98. Eigenvalue Extraction Statistics. A-103
Figure A-99. Real Eigenvalue Output. A-104
Figure A-100. Modal Participation Factor Output. A-105
Figure A-101. Rigid-Body Eigenvalue Output. A-106
Figure A-102. Modal Effective Mass Output. A-107
Figure A-103. Real Eigenvector Output. A-108
Figure A-104. Grid Point Weight Output. A-109
Figure A-105. Part Definition Output. A-110
Figure A-106. Part Mass Properties Output. A-111
Figure A-107. Part Element Geometry Summary. A-112
Figure A-108. Grid Point Singularity Table. A-113
Figure A-109. Mass Matrix Singularity Table. A-114
Figure A-110. Load Vector Output. A-115
Figure A-111. Load Vector Output (Heat Transfer Analysis). A-116
Figure A-112. Displacement Vector Output. A-117
Figure A-113. Complex Displacement Vector Output. A-118
Figure A-114. Displacement Vector Power Spectral Density Output. A-119
Figure A-115. Displacement Vector RMS Output. A-120
Figure A-116. Displacement Vector Number of Positive Crossings Output. A-121
Figure A-117. Temperature Vector Output. A-122
Figure A-118. Single Point Constraint Force Vector Output. A-123
Figure A-119. Single Point Constraint Heat Flow Vector Output. A-124

Figure A-120. Multipoint Constraint Force Vector Output. A-125
Figure A-121. Multipoint Constraint Heat Flow Vector Output. A-126
Figure A-122. Load Vector Resultant. A-127
Figure A-123. Single Point Constraint Force Vector Resultant. A-128
Figure A-124. Maximum Applied Loads. A-129
Figure A-125. Maximum Displacements. A-130
Figure A-126. Maximum Single Point Constraint Forces. A-131
Figure A-127. Nonlinear Forces in Gap Elements. A-132
Figure A-128. Forces in Spring Elements. A-133
Figure A-129. Stresses in Spring Elements. A-134
Figure A-130. Forces in Rod Elements. A-135
Figure A-131. Stresses in Rod Elements. A-136
Figure A-132. Forces in Bar Elements. A-137
Figure A-133. Stresses in Bar Elements. A-138
Figure A-134. Thermal Gradients and Heat Fluxes in Bar Elements. A-139
Figure A-135. Forces in Shear Elements. A-140
Figure A-136. Stresses in Shear Elements. A-141
Figure A-137. Heat Flow Into Hbdy Elements. A-142
Figure A-138. Forces in Quad Elements (Without Corner Option). A-143
Figure A-139. Forces in Quad Elements (With Corner Option). A-144
Figure A-140. Stresses in Quad Elements (With Corner Option). A-145
Figure A-141. Stresses in Quad Elements (Without Corner Option). A-146
Figure A-142. Strains in Quad Elements. A-147
Figure A-143. Strain Energy in Quad Elements. A-148
Figure A-144. Thermal Gradients and Heat Fluxes in Quad Elements. A-149
Figure A-145. Surface Grid Point Stresses. A-150
Figure A-146. Surface Grid Point Thermal Gradients and Heat Fluxes. A-151
Figure A-147. Stresses in Composite Quad Elements. A-152
Figure A-148. Failure Indexes in Composite Quad Elements. A-153
Figure A-149. Strength Ratios in Composite Quad Elements. A-154
Figure A-150. Stresses in Composite Hex Elements. A-155
Figure A-151. Nonlinear Stresses in Quad Elements. A-156
Figure A-152. Stresses in Hex Elements (Without Corner Option). A-157
Figure A-153. Strains in Hex Elements (Without Corner Option). A-158
Figure A-154. Strain Energy in Hex Elements. A-159
Figure A-155. Thermal Gradients and Heat Fluxes in Hex Elements. A-160
Figure A-156. Volume Grid Point Stresses. A-161
Figure A-157. Volume Grid Point Thermal Gradients and Heat Fluxes. A-162
Figure A-158. Nonlinear Stresses in Hex Elements. A-163
Figure A-159. Complex Stresses in Hex Elements. A-164
Figure A-160. Element Result Limits (Subcase Search). A-165
Figure A-161. Element Result Limits (Global Search). A-166
Figure A-162. Grid Point Result Limits (Subcase Search). A-167

Figure A-163. Grid Point Result Limits (Global Search). A-168
Figure A-164. Execution Summary. A-169

```
                    S U B C A S E   C O N S T R A I N T   A N D   L O A D   S E T   D E F I N I T I O N

SUBCASE                          LABEL                               CONSTRAINT SET IDS             LOAD SET IDS
  ID                                                                   SPC        MPC        LOAD      DEFORM   TEMPERATURE
  100     ENFORCED DISPLACEMENT                                         10       NONE       NONE       NONE       NONE
  101     POINT LOADS                                                    1       NONE         10       NONE       NONE
  102     PRESSURE LOADS                                                 1         22          4       NONE       NONE
  103     GRAVITY LOADS                                                  1       NONE         50       NONE       NONE
  104     ELEMENT INITIAL DEFORMATIONS                                   1       NONE       NONE          5       NONE
  105     GRAV LOAD                                                      1       NONE          6       NONE       NONE
  106     TEMPERATURES                                                   1       NONE       NONE       NONE          3
  107     SCALED TEMPERATURES                                            1         45       NONE       NONE        300
  108     GENERATED THERMAL GRADIENT IN Z-DIR                            1       NONE       NONE       NONE         60
```

Remarks:

1. See `SPC`, `LOAD`, `DEFORM`, and `TEMPERATURE` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for command format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `5` in the Model Initialization File or on the Nastran command line.

Figure A-1. Database Subcase Constraint and Load Set Definition.

```
                        R E C T A N G U L A R   C O O R D I N A T E   S Y S T E M   D E F I N I T I O N

COORDINATE   ORIGIN COORDINATES          X-AXIS DIRECTION COSINES      Y-AXIS DIRECTION COSINES       Z-AXIS DIRECTION COSINES
    ID     X1         X2         X3         V1         V2         V3         V1         V2         V3          V1         V2         V3
      1 0.500E+00 0.500E+00 0.200E+01  0.000E+00-0.100E+01 0.000E+00-0.100E+01 0.000E+00 0.000E+00  0.000E+00 0.000E+00-0.100E+01
      4 0.000E+00 0.000E+00 0.000E+00  0.100E+01 0.000E+00 0.000E+00 0.000E+00 0.100E+01 0.000E+00  0.000E+00 0.000E+00 0.100E+01
```

Remarks:

1. See `CORD1R` and `CORD2R` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `3` in the Model Initialization File or on the Nastran command line.

Figure A-2. Database Rectangular Coordinate System Definition.

```
                        C Y L I N D R I C A L   C O O R D I N A T E   S Y S T E M   D E F I N I T I O N

COORDINATE    ORIGIN COORDINATES          R-AXIS DIRECTION COSINES       T-AXIS DIRECTION COSINES       Z-AXIS DIRECTION COSINES
    ID      X1        X2        X3         V1        V2        V3         V1        V2        V3         V1        V2        V3
    2  0.500E+00  0.500E+00  0.000E+00 -0.100E+01 0.000E+00 0.000E+00  0.000E+00 0.100E+01 0.000E+00  0.000E+00 0.000E+00-0.100E+01
    5  0.000E+00  0.000E+00  0.000E+00  0.100E+01 0.000E+00 0.000E+00  0.000E+00 0.100E+01 0.000E+00  0.000E+00 0.000E+00 0.100E+01
```

Remarks:

1. See `CORD1C` and `CORD2C` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `3` in the Model Initialization File or on the Nastran command line.

Figure A-3. Database Cylindrical Coordinate System Definition.

```
                             S P H E R I C A L   C O O R D I N A T E   S Y S T E M   D E F I N I T I O N

COORDINATE    ORIGIN COORDINATES          R-AXIS DIRECTION COSINES       P-AXIS DIRECTION COSINES       T-AXIS DIRECTION COSINES
    ID      X1         X2         X3         V1         V2         V3         V1         V2         V3         V1         V2         V3
    3   0.5000E+00 0.500E+00 0.100E+01 -0.100E+01 0.000E+00 0.000E+00  0.000E+00 0.100E+01 0.000E+00  0.000E+00 0.000E+00 -0.100E+01
    6   0.0000E+00 0.000E+00 0.000E+00  0.100E+01 0.000E+00 0.000E+00  0.000E+00 0.100E+01 0.000E+00  0.000E+00 0.000E+00  0.100E+01
```

Remarks:

1. See `CORD1S` and `CORD2S` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `3` in the Model Initialization File or on the Nastran command line.

Figure A-4. Database Spherical Coordinate System Definition.

```
                                      G R I D   P O I N T   D E F I N I T I O N

     GRID      COORDINATE                   COORDINATES
      ID          ID            X1            X2            X3
       1           0        0.000E+00     0.000E+00     0.000E+00
       2           1       -0.119E-06    -0.500E+00     0.300E+01
       3           1        0.500E+00    -0.500E+00     0.300E+01
       4           1       -0.500E+00    -0.500E+00     0.350E+01
       5           1       -0.119E-06    -0.500E+00     0.350E+01
       6           1        0.500E+00    -0.500E+00     0.350E+01
       7           1       -0.500E+00    -0.500E+00     0.400E+01
       8           1       -0.119E-06    -0.500E+00     0.400E+01
       9           1        0.500E+00    -0.500E+00     0.400E+01
      10           1        0.500E+00     0.500E+00     0.300E+01
```

Remarks:

1. See `GRID` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` in the Model Initialization File or on the Nastran command line.

Figure A-5. Database Grid Point Definition.

```
                                C O N C E N T R A T E D   M A S S   E L E M E N T   D E F I N I T I O N

 ELEMENT    GRID   COORDINATE                        MASS MATRIX                                          OFFSET VECTOR
   ID        ID        ID                                                                           V1          V2          V3
  4000        84         3  1  0.110E+03                                                          0.140E+00  0.990E+00  0.000E+00
                            2  0.000E+00  0.110E+03
                            3  0.000E+00  0.000E+00  0.110E+03
                            4  0.000E+00  0.000E+00  0.000E+00  0.000E+00
                            5  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
                            6  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
  4001        86         3  1  0.110E+03                                                          0.140E+00  0.990E+00  0.000E+00
                            2  0.000E+00  0.110E+03
                            3  0.000E+00  0.000E+00  0.110E+03
                            4  0.000E+00  0.000E+00  0.000E+00  0.000E+00
                            5  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
                            6  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
```

Remarks:

1. See `CONM1` and `CONM2` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` in the Model Initialization File or on the Nastran command line.

Figure A-6. Database Concentrated Mass Element Definition.

```
                                              G A P   E L E M E N T   D E F I N I T I O N

   ELEMENT      PROPERTY      GRID-1       GRID-2                X-Y PLANE VECTOR
     ID            ID            ID           ID            V1            V2            V3
      20           20             5           19        0.000E+00     1.000E+00     0.000E+00
      21           21             4           20        1.000E+00     0.000E+00     0.000E+00
```

Remarks:

1. See `CGAP` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` in the Model Initialization File or on the Nastran command line.

Figure A-7. Database Gap Element Definition.

```
                                  S P R I N G   E L E M E N T   D E F I N I T I O N

  ELEMENT      PROPERTY      GRID-1       COMPONENT    GRID-2       COMPONENT
    ID            ID           ID           NUMBER       ID           NUMBER
   1136           70          119             1         115             1
   2136           70          119             2         115             2
   3136           70          119             3         115             3
   1134           71          210             1         117             1
   1135           72          211             1         119             1
   2134           73          210             2         117             2
   2135           74          211             2         119             2
```

Remarks:

1. See `CELAS1` and `CELAS2` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` in the Model Initialization File or on the Nastran command line.

Figure A-8. Database Spring Element Definition.

```
                              R O D   E L E M E N T   D E F I N I T I O N

   ELEMENT     PROPERTY      GRID-1       GRID-2
     ID           ID           ID           ID
      45          70           59           64
      46          70           64           69
      47          70           69           58
      48          70           58           59
      49          70           58           84
      50          70           59           86
      51          70           69           88
      52          70           64           90
      53          70           86           90
```

Remarks:

1. See `CROD` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` in the Model Initialization File or on the Nastran command line.

Figure A-9. Database Rod Element Definition.

```
                                        B A R   E L E M E N T   D E F I N I T I O N

   ELEMENT   PROPERTY  GRID-1    GRID-2      X-Y PLANE VECTOR              END-A OFFSET VECTOR             END-B OFFSET VECTOR
     ID         ID       ID        ID      V1         V2         V3         V1         V2         V3         V1         V2         V3
      61         80      115         86  0.707E+00 0.707E+00-0.233E-05  0.000E+00 0.000E+00 0.000E+00  0.000E+00 0.000E+00 0.000E+00
      62         80      117         90  0.727E+00 0.294E+00-0.619E+00  0.500E+00 0.000E+00 0.000E+00  0.500E+00 0.000E+00 0.000E+00
      63         80      119         88 -0.656E-01 0.969E+00-0.235E+00  0.200E+00 0.200E+00 0.000E+00  0.200E+00 0.200E+00 0.000E+00
      64         80      121         84  0.000E+00-0.100E+01 0.000E+00  0.000E+00 0.000E+00 0.000E+00  0.000E+00 0.000E+00 0.000E+00
      65         80      121        115 -0.990E-01-0.990E-01-0.990E+00  0.000E+00 0.000E+00 0.000E+00  0.000E+00 0.000E+00 0.000E+00
      66         80      119        121  0.000E+00 0.000E+00-0.100E+01  0.000E+00 0.000E+00 0.000E+00  0.000E+00 0.000E+00 0.000E+00
      67         80      117        119 -0.990E-01-0.990E-01-0.990E+00  0.000E+00 0.000E+00 0.000E+00  0.000E+00 0.000E+00 0.000E+00
      68         80      115        117 -0.192E+00-0.192E+00-0.962E+00  0.000E+00 0.000E+00 0.000E+00  0.000E+00 0.000E+00 0.000E+00
```

Remarks:

1. See `CBAR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` in the Model Initialization File or on the Nastran command line.

Figure A-10. Database Bar Element Definition.

```
                                        B E A M   E L E M E N T   D E F I N I T I O N

ELEMENT   PROPERTY  GRID-1    GRID-2       X-Y PLANE VECTOR             END-A OFFSET VECTOR            END-B OFFSET VECTOR
  ID         ID       ID        ID      V1        V2        V3         V1        V2        V3         V1        V2        V3
   11         10      11       12  0.000E+00 0.100E+01 0.000E+00  0.000E+00 0.000E+00 0.000E+00  0.000E+00 0.000E+00 0.000E+00
   12         10      11       14  0.000E+00 0.100E+01 0.000E+00  0.000E+00 0.000E+00 0.000E+00  0.000E+00 0.000E+00 0.000E+00
   23         80     119       88 -0.656E-01 0.970E+00-0.236E+00  0.200E+00 0.200E+00 0.000E+00  0.200E+00 0.200E+00 0.000E+00
   24         80     121       84 -0.100E+01 0.000E+00 0.330E-05  0.000E+00 0.000E+00 0.000E+00  0.000E+00 0.000E+00 0.000E+00
   26         30     211      111  0.000E+00 0.100E+01 0.000E+00  0.000E+00 0.000E+00 0.000E+00  0.000E+00 0.000E+00 0.000E+00
   27         40     121      111  0.000E+00 0.100E+01 0.000E+00  0.000E+00 0.000E+00 0.000E+00  0.000E+00 0.000E+00 0.000E+00
   28         40     117      190  0.727E+00 0.294E+00-0.619E+00  0.500E+00 0.000E+00 0.000E+00  0.500E+00 0.000E+00 0.000E+00
   29         40     127      200  0.727E+00 0.294E+00-0.619E+00  0.500E+00 0.000E+00 0.000E+00  0.500E+00 0.000E+00 0.000E+00
```

Remarks:

1. See `CBEAM` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` in the Model Initialization File or on the Nastran command line.

Figure A-11. Database Beam Element Definition.

```
                                        Q U A D   E L E M E N T   D E F I N I T I O N

ELEMENT     PROPERTY      GRID-1       GRID-2        GRID-3       GRID-4       MATERIAL     ORIENTATION    OFFSET
  ID           ID           ID           ID            ID           ID       COORDINATE ID     ANGLE
    1           40            1            2             5            4               1         0.00      0.000E+00
    2           50            2            3             6            5               1         0.00      0.000E+00
    5           50           10           11            14           13               2         0.00      0.600E+00
    6           40           11           12            15           14               2         0.00      0.600E+00
    7           40           13           14            17           16               2         0.00      0.600E+00
    8           40           14           15            18           17         ELEMENT        45.00      0.000E+00
   25           40           12           38            41           15         ELEMENT        45.00      0.000E+00
   26           40           38            1             4           41         ELEMENT        60.00      0.000E+00
```

Remarks:

1. See `CQUAD4` and `CQUADR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` in the Model Initialization File or on the Nastran command line.

Figure A-12. Database Quad Element Definition.

```
                                         T R I   E L E M E N T   D E F I N I T I O N

ELEMENT     PROPERTY      GRID-1       GRID-2       GRID-3       MATERIAL     ORIENTATION    OFFSET
  ID           ID           ID           ID           ID       COORDINATE ID     ANGLE
   33           50           18           44           58               1       -90.00      0.000E+00
   34           50           59           58           44               1        90.00      0.000E+00
   35           50           44            7           59               1       -90.00      0.000E+00
   36           50            7            8           59               1         0.00      0.000E+00
   37           50           64           59            8               1         0.00      0.000E+00
   38           50            8            9           64               1         0.00      0.000E+00
   39           50            9           53           64         ELEMENT        90.00      0.000E+00
   40           50           69           64           53         ELEMENT       -90.00      0.000E+00
```

Remarks:

1. See `CTRIA3` and `CTRIAR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` in the Model Initialization File or on the Nastran command line.

Figure A-13. Database Tri Element Definition.

```
                                    S H E A R   E L E M E N T   D E F I N I T I O N

    ELEMENT     PROPERTY      GRID-1       GRID-2       GRID-3       GRID-4
      ID           ID           ID           ID           ID           ID
       57          60           59           64           90           86
       58          60           64           69           88           90
       59          60           69           58           84           88
       60          60           58           59           86           84
     1157          60           59           64           90           86
     1158          60           64           69           88           90
```

Remarks:

1. See `CSHEAR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` in the Model Initialization File or on the Nastran command line.

Figure A-14. Database Shear Element Definition.

```
                                       H E X   E L E M E N T   D E F I N I T I O N

   ELEMENT      PROPERTY      GRID-1       GRID-2       GRID-3       GRID-4       GRID-5       GRID-6       GRID-7       GRID-8
     ID            ID           ID           ID           ID           ID           ID           ID           ID           ID
      69           10          131          132          135          134          140          141          144          143
      70           10          132          133          136          135          141          142          145          144
      71           10          134          135          138          137          143          144          147          146
      72           10          135          136          139          138          144          145          148          147
      73           10          140          141          144          143          149          150          153          152
      74           10          141          142          145          144          150          151          154          153
      75           10          143          144          147          146          152          153          156          155
      76           10          144          145          148          147          153          154          157          156
```

Remarks:

1. See `CHEXA` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. Only the grid points for the corner nodes are displayed.
3. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` in the Model Initialization File or on the Nastran command line.

Figure A-15. Database Hex Element Definition.

```
                                   P E N T   E L E M E N T   D E F I N I T I O N

  ELEMENT      PROPERTY      GRID-1        GRID-2        GRID-3        GRID-4        GRID-5        GRID-6
    ID            ID           ID            ID            ID            ID            ID            ID
     77            20          158           159           161           167           168           170
     78            20          162           161           159           171           170           168
     79            20          132           162           159           135           171           168
     80            20          159           131           132           168           134           135
     81            20          161           162           164           170           171           173
     82            20          165           164           162           174           173           171
     83            20          133           165           162           136           174           171
     84            20          162           132           133           171           135           136
     85            20          167           168           170           176           177           179
```

Remarks:

1. See `CPENTA` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. Only the grid points for the corner nodes are displayed.
3. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` in the Model Initialization File or on the Nastran command line.

Figure A-16. Database Pent Element Definition.

```
                         T E T   E L E M E N T   D E F I N I T I O N

  ELEMENT     PROPERTY      GRID-1        GRID-2        GRID-3        GRID-4
    ID           ID           ID            ID            ID            ID
     93          30          167           170           158           195
     94          30          198           195           197           170
     95          30          161           158           170           197
     96          30          194           197           195           158
     97          30          158           195           170           197
     98          30          167           176           170           195
     99          30          196           195           199           176
    100          30          179           170           176           199
```

Remarks:

1. See `CTETRA` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. Only the grid points for the corner nodes are displayed.
3. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` in the Model Initialization File or on the Nastran command line.

Figure A-17. Database Tet Element Definition.

```
                                          G A P   E L E M E N T   P R O P E R T Y   D E F I N I T I O N

  PROPERTY        INITIAL       PRELOAD        AXIAL STIFFNESS        TRANSVERSE      MU-X         MU-Y         MAXIMUM
     ID           OPENING                     CLOSED         OPEN     STIFFNESS                               PENETRATION
      20         2.000E-02     0.000E+00     1.000E+08     1.000E+02     0.000E+00     0.000E+00    0.000E+00    0.000E+00
      21         0.000E+00     0.000E+00     1.000E+08     1.000E+02     1.000E+07     1.000E-01    1.000E-01    0.000E+00
```

Remarks:

1. See `PGAP` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `4` or `6` - `7` in the Model Initialization File or on the Nastran command line.

Figure A-18. Database Gap Element Property Definition.

```
                              S P R I N G   E L E M E N T   P R O P E R T Y   D E F I N I T I O N

  PROPERTY          K              DEPENDENT TABLE IDS
     ID                       K-FREQ     GE-FREQ    K-FORCE
      10       1.000E+04        10          20
      20       2.000E+04        10          20
      30       5.000E+03                               30
```

Remarks:

1. See `PELAS` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `4` or `6` - `7` in the Model Initialization File or on the Nastran command line.

Figure A-19. Database Spring Element Property Definition.

```
                                         R O D   E L E M E N T   P R O P E R T Y   D E F I N I T I O N

    PROPERTY      MATERIAL           AREA           J            NSM
       ID            ID
        70           100         0.900E-01    0.608E-02    0.000E+00
        85           145         0.500E-01    0.308E-02    0.100E+00
```

Remarks:

1. See `PROD` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `4` or `6` - `7` in the Model Initialization File or on the Nastran command line.

Figure A-20. Database Rod Element Property Definition.

```
                                 B A R   E L E M E N T   P R O P E R T Y   D E F I N I T I O N

PROPERTY     MATERIAL         AREA           I1           I2          I12            J           K1           K2          NSM
   ID           ID
    80          100         0.900E-01    0.608E-02    0.750E-04    0.000E+00    0.267E-03    0.000E+00    0.000E+00    0.000E+00
    90          140         0.400E-01    0.408E-02    0.350E-04    0.000E+00    0.567E-03    0.000E+00    0.000E+00    0.100E+00
```

Remarks:

1. See `PBAR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `4` or `6` - `7` in the Model Initialization File or on the Nastran command line.

Figure A-21. Database Bar Element Property Definition.

```
                                B E A M   E L E M E N T   P R O P E R T Y   D E F I N I T I O N

PROPERTY     MATERIAL     DISTANCE     AREA         I1         I2        I12          J          K1         K2         NSM
   ID           ID
    11            1        0.0000    0.500E+01   0.104E+02  0.417E+00  0.000E+00  0.000E+00   0.000E+00  0.000E+00   0.000E+00
                           0.1111    0.456E+01   0.788E+01  0.380E+00  0.000E+00  0.000E+00
                           0.2222    0.411E+01   0.579E+01  0.343E+00  0.000E+00  0.000E+00
                           0.3333    0.367E+01   0.411E+01  0.306E+00  0.000E+00  0.000E+00
                           0.4444    0.322E+01   0.279E+01  0.269E+00  0.000E+00  0.000E+00
                           0.5555    0.278E+01   0.179E+01  0.232E+00  0.000E+00  0.000E+00
                           0.6666    0.233E+01   0.106E+01  0.194E+00  0.000E+00  0.000E+00
                           0.7777    0.189E+01   0.562E+00  0.157E+00  0.000E+00  0.000E+00
                           0.8888    0.144E+01   0.251E+00  0.120E+00  0.000E+00  0.000E+00
                           1.0000    0.100E+01   0.833E-01  0.833E-01  0.000E+00  0.000E+00
```

Remarks:

1. See `PBEAM` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `4` or `6` - `7` in the Model Initialization File or on the Nastran command line.

Figure A-22. Database Beam Element Property Definition.

```
                  C O M P O S I T E   S H E L L   E L E M E N T   P R O P E R T Y   D E F I N I T I O N

 PROPERTY       Z0          NSM         T-REF        PLY       PLY    MATERIAL  THICKNESS  ORIENTATION    SHEAR      FAILURE     PLY
    ID                                              NUMBER     ID        ID                   ANGLE     ALLOWABLE    THEORY    OUTPUT
    10     -1.000E-01   2.000E+00   7.000E+01       1         1       100    4.000E-02      0.00     1.000E+03    TSAI-WU      ON
                                                    2         2       105    2.000E-02     45.00     1.000E+03    TSAI-WU      ON
                                                    3         3       105    2.000E-02     60.00     1.000E+03    TSAI-WU      ON
                                                    4         4       105    2.000E-02     90.00     1.000E+03    TSAI-WU      ON
                                                    5         5       105    2.000E-02     90.00     1.000E+03    TSAI-WU      ON
                                                    6         6       105    2.000E-02     60.00     1.000E+03    TSAI-WU      ON
                                                    7         7       105    4.000E-02     45.00     1.000E+03    TSAI-WU      ON
                                                    8         8       100    2.000E-02      0.00     1.000E+03    TSAI-WU      ON
    11     -7.000E-02   3.000E+00   7.000E+01       1         1       100    1.000E-02      0.00     5.000E+02    LARC02       ON
                                                    2         2       105    2.000E-02     45.00     5.000E+02    LARC02       ON
                                                    3         3       105    2.000E-02     60.00     5.000E+02    LARC02       ON
                                                    4         4       105    2.000E-02     90.00     5.000E+02    LARC02       ON
                                                    5         5       105    2.000E-02     90.00     5.000E+02    LARC02       ON
                                                    6         6       105    2.000E-02     60.00     5.000E+02    LARC02       ON
                                                    7         7       105    2.000E-02     45.00     5.000E+02    LARC02       ON
                                                    8         8       100    1.000E-02      0.00     5.000E+02    LARC02       ON
```

Remarks:

1. See `PCOMP` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `4` or `6` - `7` in the Model Initialization File or on the Nastran command line.

Figure A-23. Database Composite Shell Element Property Definition.

```
                               S H E L L   E L E M E N T   P R O P E R T Y   D E F I N I T I O N

PROPERTY                       MATERIAL IDS                         THICKNESS      TS/T        12I/T3         NSM
   ID         MEMBRANE      BENDING    TRANSVERSE    COUPLING
    40          500          500           500                      0.150E+00    0.833E+00    0.150E+01    0.000E+00
    50          100          100           100                      0.200E+00    0.833E+00    0.100E+01    0.000E+00
    90          511          512           513                      0.280E+00    0.100E+01    0.100E+01    0.000E+00
   150          471          472           473          474         0.100E+00    0.100E+01    0.100E+01    0.000E+00
```

Remarks:

1. See `PSHELL` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `4` or `6` - `7` in the Model Initialization File or on the Nastran command line.

Figure A-24. Database Shell Element Property Definition.

```
                              S H E A R   E L E M E N T   P R O P E R T Y   D E F I N I T I O N

   PROPERTY     MATERIAL       THICKNESS       NSM       EFFECTIVENESS FACTORS
      ID           ID                                   X-DIRECTION  Y-DIRECTION
       60          100         0.500E-01    0.000E+00    0.500E+00    1.200E+00
       70          120         0.700E-01    0.100E+00    0.700E+00    0.500E+00
```

Remarks:

1. See `PSHEAR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `4` or `6` - `7` in the Model Initialization File or on the Nastran command line.

Figure A-25. Database Shear Element Property Definition.

```
                                   S O L I D   E L E M E N T   P R O P E R T Y   D E F I N I T I O N

     PROPERTY     MATERIAL      MATERIAL
        ID           ID       COORDINATE ID
         10         100          ELEMENT
         20         100                2
         30         400                1
```

Remarks:

1. See `PSOLID` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `4` or `6` - `7` in the Model Initialization File or on the Nastran command line.

Figure A-26. Database Solid Element Material Property Definition.

```
                              I S O T R O P I C   M A T E R I A L   D E F I N I T I O N

 MATERIAL            E            G            NU           RHO          ALPHA        T-REF
    ID
    100         0.100E+08    0.385E+07    0.300E+00    0.100E+00    0.200E-05    0.120E+03
    120         0.300E+08    0.113E+08    0.330E+00    0.320E+00    0.700E-05    0.120E+03
```

Remarks:

1. See `MAT1` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `8` in the Model Initialization File or on the Nastran command line.

Figure A-27. Database Isotropic Material Definition.

```
                   A N I S O T R O P I C   S H E L L   E L E M E N T   M A T E R I A L   D E F I N I T I O N

MATERIAL            MATERIAL PROPERTY MATRIX            ALPHA        T-REF         RHO
   ID              1            2            3
   200     1   0.914E+07                               0.152E-05    0.000E+00    0.700E-01
           2   0.535E+06    0.111E+08                  0.140E-05
           3   0.787E+05   -0.787E+05    0.423E+07    -0.350E-06

   220     1   0.103E+07                               0.150E-05    0.110E+03    0.150E+00
           2   0.135E+07    0.217E+07                  0.250E-05
           3   0.121E+07    0.224E+07    0.311E+07     0.330E-05

   511     1   0.914E+07                               0.152E-05    0.000E+00    0.700E-01
           2   0.535E+06    0.111E+08                  0.140E-05
           3   0.787E+05   -0.787E+05    0.423E+07    -0.350E-06

   512     1   0.124E+08                               0.147E-05    0.000E+00    0.700E-01
           2   0.573E+06    0.943E+07                  0.142E-05
           3   0.607E+05   -0.607E+05    0.458E+07    -0.386E-06
```

Remarks:

1. See `MAT2` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `8` in the Model Initialization File or on the Nastran command line.

Figure A-28. Database Anisotropic Shell Element Material Definition.

```
                                        I S O T R O P I C   M A T E R I A L   D E F I N I T I O N

       MATERIAL           K            CP            RHO            H            MU           H-GEN
          ID
           10        0.000E+00    0.000E+00    0.000E+00    1.500E-06    0.000E+00    1.000E+00
           20        0.000E+00    0.000E+00    0.000E+00    2.500E-06    0.000E+00    1.000E+00
          200        1.167E-03    4.637E+01    4.145E-04    0.000E+00    0.000E+00    1.000E+00
          300        5.556E-04    4.053E+01    7.331E-04    0.000E+00    0.000E+00    1.000E+00
```

Remarks:

1. See `MAT4` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 8` in the Model Initialization File or on the Nastran command line.

Figure A-29. Database Isotropic Material Definition (Heat Transfer Analysis).

```
                    A N I S O T R O P I C   E L E M E N T   M A T E R I A L   D E F I N I T I O N

 MATERIAL           MATERIAL PROPERTY MATRIX             CP           RHO          H-GEN
    ID              1            2            3
     3     1   2.060E-03                              0.000E+00    0.000E+00    1.000E+00
           2   1.200E-03    3.120E-03
           3  -5.800E-04   -2.400E-04    6.100E-03

     4     1   1.060E-03                              0.000E+00    0.000E+00    1.000E+00
           2   2.200E-03    2.120E-03
           3  -5.400E-04   -3.500E-04    4.500E-03

     5     1   4.060E-03                              0.000E+00    0.000E+00    1.000E+00
           2   3.100E-03    3.420E-03
           3  -5.750E-04   -1.400E-04    5.400E-03

     6     1   2.070E-03                              0.000E+00    0.000E+00    1.000E+00
           2   3.500E-03    1.420E-03
           3  -1.300E-04   -3.570E-04    2.670E-03
```

Remarks:

1. See `MAT5` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `8` in the Model Initialization File or on the Nastran command line.

Figure A-30. Database Anisotropic Element Material Definition (Heat Transfer Analysis).

```
                     O R T H O T R O P I C   S H E L L   E L E M E N T   M A T E R I A L   D E F I N I T I O N

 MATERIAL        E1           E2          NU12          G12          G1Z          G2Z          RHO          A1           A2           T-REF
    ID
    500      0.140E+08   0.140E+08   0.100E+00   0.700E+07   0.130E+07   0.170E+07   0.750E-01   0.120E-05   0.170E-05   0.700E+01
    510      0.100E+08   0.100E+07   0.200E+00   0.200E+07   0.100E+07   0.170E+07   0.650E-01   0.140E-05   0.190E-05   0.000E+00
```

Remarks:

1. See `MAT8` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `8` in the Model Initialization File or on the Nastran command line.

Figure A-31. Database Orthotropic Material Definition.

```
                   A N I S O T R O P I C   S O L I D   E L E M E N T   M A T E R I A L   D E F I N I T I O N

MATERIAL                                 MATERIAL PROPERTY MATRIX                                ALPHA        T-REF         RHO
   ID               1            2            3             4            5            6
   400     1   0.107E+08                                                                       0.100E-02    0.100E+03    0.100E+00
           2   0.691E+07     0.190E+08                                                         0.244E-02
           3   0.689E+07     0.690E+07    0.143E+08                                            0.344E-02
           4   0.125E+07     0.127E+07    0.123E+07    0.527E+07                               0.403E-02
           5  -0.900E+05    -0.120E+06   -0.150E+06    0.210E+06    0.463E+07                  0.234E-02
           6   0.860E+06     0.830E+06    0.820E+06   -0.110E+06    0.520E+06     0.572E+07    0.502E-02

   470     1   0.117E+08                                                                       0.100E-02    0.120E+03    0.130E+00
           2   0.631E+07     0.193E+08                                                         0.600E-02
           3   0.699E+07     0.790E+07    0.143E+08                                            0.350E-02
           4   0.135E+07     0.177E+07    0.143E+07    0.327E+07                               0.440E-02
           5  -0.140E+06    -0.620E+06   -0.450E+06    0.310E+06    0.433E+07                  0.530E-02
           6   0.850E+06     0.530E+06    0.420E+06   -0.310E+06    0.530E+06     0.573E+07    0.420E-02
```

Remarks:

1. See `MAT9` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 8` in the Model Initialization File or on the Nastran command line.

Figure A-32. Database Anisotropic Solid Element Material Definition.

```
                                   N O N L I N E A R   M A T E R I A L   D E F I N I T I O N

MATERIAL      NONLINEARITY TYPE        TABLE     HARDENING          YIELD           HARDENING         YIELD       FRICTION
   ID                                   ID         SLOPE           FUNCTION           RULE            POINT        ANGLE
   100        PLASTIC                     0      5.000E+06        VON MISES         ISOTROPIC       3.000E+03
   110        PLASTIC                     0      1.000E+06        VON MISES         ISOTROPIC       1.500E+03
   120        NONLINEAR ELASTIC          10
   130        NONLINEAR ELASTIC          20
```

Remarks:

1. See `MATS1` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `8` in the Model Initialization File or on the Nastran command line.

Figure A-33. Database Nonlinear Material Definition.

```
                                    I S O T R O P I C   M A T E R I A L   T A B L E   D E F I N I T I O N

    MATERIAL              E              G              NU             RHO           ALPHA
       ID
       100               101            102            103             104
       200               108            108                            106             107
```

Remarks:

1. See `MATT1` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 8` in the Model Initialization File or on the Nastran command line.

Figure A-34. Database Isotropic Material Table Definition.

```
            A N I S O T R O P I C   S H E L L   E L E M E N T   M A T E R I A L   T A B L E   D E F I N I T I O N

 MATERIAL            MATERIAL PROPERTY MATRIX             ALPHA        RHO
    ID              1            2            3
    200     1       101                                    108         107
            2       102          104                       108
            3       103          105          106          109
```

Remarks:

1. See `MATT2` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 8` in the Model Initialization File or on the Nastran command line.

Figure A-35. Database Anisotropic Shell Element Material Table Definition.

```
                             I S O T R O P I C   M A T E R I A L   T A B L E   D E F I N I T I O N

MATERIAL           K           CP           RHO           H           MU         H-GEN
   ID
   100           10
   200                                                                            30
   300           20
```

Remarks:

1. See `MATT4` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 8` in the Model Initialization File or on the Nastran command line.

Figure A-36. Database Isotropic Material Table Definition (Heat Transfer Analysis).

```
                A N I S O T R O P I C   E L E M E N T   M A T E R I A L   T A B L E   D E F I N I T I O N

  MATERIAL            MATERIAL PROPERTY MATRIX            CP           RHO        H-GEN
     ID              1            2            3
      3     1         10                                                         90
            2         20           10
            3         30           20           10

      4     1         50
            2        120          110
            3        130           20          110
```

Remarks:

1. See `MATT5` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `8` in the Model Initialization File or on the Nastran command line.

Figure A-37. Database Orthotropic Material Table Definition (Heat Transfer Analysis).

```
                   O R T H O T R O P I C   S H E L L   E L E M E N T   T A B L E   D E F I N I T I O N

MATERIAL         E1          E2         NU12         G12         G1Z         G2Z         RHO         A1          A2
   ID
   300           101         102         103                     104                     107         108         106
   500            24          45                                             356         107                     106
```

Remarks:

1. See `MATT8` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `8` in the Model Initialization File or on the Nastran command line.

Figure A-38. Database Orthotropic Shell Element Material Table Definition.

```
             A N I S O T R O P I C   S O L I D   E L E M E N T   M A T E R I A L   T A B L E   D E F I N I T I O N

MATERIAL                                MATERIAL PROPERTY MATRIX                                ALPHA        RHO
   ID              1            2            3            4            5            6
   400     1       101
           2       102                                                                           102
           3       103          105
           4       104          106          107
           5                                                                                     103
           6                                 108                       109
```

Remarks:

1. See `MATT9` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 8` in the Model Initialization File or on the Nastran command line.

Figure A-39. Database Anisotropic Solid Element Material Table Definition.

```
                         M A T E R I A L   P R O P E R T Y   T A B L E   D E F I N I T I O N

 TABLE          X-VALUE      Y-VALUE
  ID
  101        -0.500E+03    0.400E+00
              0.000E+00    0.900E+00
              0.100E+04    0.118E+01
              0.100E+05    0.130E+01
              0.200E+05    0.179E+01
  102        -0.500E+03    0.580E+00
              0.000E+00    0.880E+00
              0.100E+04    0.116E+01
              0.100E+05    0.179E+01
              0.200E+05    0.200E+01
  103        -0.500E+03    0.580E+00
              0.000E+00    0.880E+00
              0.100E+04    0.116E+01
              0.100E+05    0.179E+01
              0.200E+05    0.200E+01
  201        -0.500E+03    0.880E+00
              0.000E+00    0.980E+00
              0.100E+04    0.118E+01
              0.100E+05    0.130E+01
              0.200E+05    0.140E+01
```

Remarks:

1. See `TABLEM1`, `TABLEM2`, `TABLEM3`, and `TABLEM4` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. For the specified table x value the calculated y value is shown.
3. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `6` in the Model Initialization File or on the Nastran command line.

Figure A-40. Database Material Property Table Definition.

```
                                        D Y N A M I C   L O A D   T A B L E   D E F I N I T I O N

     TABLE         X-VALUE      Y-VALUE
      ID
       11         0.000E+00    0.000E+00
                  0.100E-02    0.100E+01
                  0.100E+03    0.100E+01
       12         0.000E+00    0.000E+00
                  0.300E-01    0.000E+00
                  0.310E-01    0.100E+01
                  0.100E+03    0.100E+01
```

Remarks:

1. See `TABLED1`, `TABLED2`, `TABLED3`, and `TABLED4` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. For the specified table x value the calculated y value is shown.
3. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `6` in the Model Initialization File or on the Nastran command line.

Figure A-41. Database Dynamic Load Table Definition.

```
                              M O D A L   D A M P I N G   T A B L E   D E F I N I T I O N

    TABLE         X-VALUE       Y-VALUE
     ID
      20        0.100E+01    0.500E-01
                0.100E+05    0.500E-01
      30        0.100E+01    0.250E-01
                0.100E+05    0.250E-01
```

Remarks:

1. See `TABDMP1` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. For the specified table x value the calculated y value is shown.
3. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `6` in the Model Initialization File or on the Nastran command line.

Figure A-42. Database Modal Damping Table Definition.

```
                              S I N G L E   P O I N T   C O N S T R A I N T   D E F I N I T I O N

       SET          GRID         COMPONENT       ENFORCED
        ID           ID           NUMBERS      DISPLACEMENT
          2          119                1        0.100E+00
          2          121                1        0.100E+00
          1          149           123456        0.000E+00
          1          150           123456        0.000E+00
          1          153           123456        0.000E+00
          1          154           123456        0.000E+00
```

Remarks:

1. See `SPC`, `SPC1`, and `SPCD` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `2` in the Model Initialization File or on the Nastran command line.

Figure A-43. Database Single Point Constraint Definition.

```
                    S I N G L E   P O I N T   C O N S T R A I N T   A D D I T I O N   D E F I N I T I O N

    SET            SPC SET IDS
     ID
      10             1            2
      20             3            4            5
```

Remarks:

1. See `SPCADD` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-44. Database Single Point Constraint Addition Definition.

```
                              M U L T I P O I N T   C O N S T R A I N T   D E F I N I T I O N

               DEPENDENT DEGREES OF FREEDOM                               INDEPENDENT DEGREES OF FREEDOM
SET           GRID    COMPONENT   COEFFICIENT         GRID    COMPONENT   COEFFICIENT         GRID    COMPONENT   COEFFICIENT
 ID            ID      NUMBER                          ID      NUMBER                          ID      NUMBER
   1            15        1        0.100E+01            14        1       -0.100E+01
                15        2        0.100E+01            14        2       -0.100E+01            14        6         0.500E+00
                15        3        0.100E+01            14        3       -0.100E+01            14        5        -0.500E+00
                15        2        0.100E+01            14        2       -0.100E+01            14        6         0.500E+00
                15        3        0.100E+01            14        3       -0.100E+01            14        5        -0.500E+00
                15        4        0.100E+01            14        4       -0.100E+01
                15        5        0.100E+01            14        5       -0.100E+01
                15        6        0.100E+01            14        6       -0.100E+01
    2          161        1        0.100E+01            11        1       -0.100E+01            11        5         0.100E+01
                                                        11        6       -0.500E+00
               161        2        0.100E+01            11        2       -0.100E+01            11        5        -0.500E+00
                                                        11        6       -0.100E+01
               161        3        0.100E+01            11        3       -0.100E+01            11        4         0.500E+00
                                                        11        5        0.500E+00
               164        1        0.100E+01            11        1       -0.100E+01            11        5         0.100E+01
                                                        11        6       -0.100E+01
               164        2        0.100E+01            11        2       -0.100E+01            11        5        -0.500E+00
                                                        11        6       -0.100E+01
               164        3        0.100E+01            11        3       -0.100E+01            11        4         0.100E+01
                                                        11        5        0.500E+00
```

Remarks:

1. See `MPC` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `2` in the Model Initialization File or on the Nastran command line.

Figure A-45. Database Multipoint Constraint Definition.

```
                          M U L T I P O I N T   C O N S T R A I N T   A D D I T I O N   D E F I N I T I O N

     SET             MPC SET IDS
      ID
       10              1            2
       20              3            4            5
```

Remarks:

1. See `MPCADD` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-46. Database Multipoint Constraint Addition Definition.

```
                          G R I D   P O I N T   F O R C E   V E C T O R   D E F I N I T I O N

   SET          GRID       COORDINATE                 FORCE VECTOR
    ID            ID           ID            V1            V2            V3
     1            11            1         0.000E+00     0.000E+00    -0.100E+02
     1            84            3         0.990E-01     0.700E+00     0.707E+00
     1            86            3         0.990E-01     0.700E+00    -0.707E+00
     2             5            1         0.000E+00     0.000E+00    -0.100E+02
```

Remarks:

1. See `FORCE` and `FORCE1` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-47. Database Grid Point Force Vector Definition.

```
                              G R I D   P O I N T   M O M E N T   V E C T O R   D E F I N I T I O N

   SET           GRID       COORDINATE                MOMENT VECTOR
    ID            ID            ID              V1            V2            V3
     2           115             2          0.707E+00     0.707E+00     0.000E+00
     2           121             2         -0.707E+00     0.707E+00     0.000E+00
```

Remarks:

1. See `MOMENT` and `MOMENT1` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-48. Database Grid Point Moment Vector Definition.

```
                                     G R A V I T Y   L O A D   V E C T O R   D E F I N I T I O N

     SET       COORDINATE                GRAVITY VECTOR
      ID            ID             V1            V2            V3
        6             0         0.000E+00    0.100E+02    0.000E+00
        9             0         0.150E+02    0.150E+03    0.000E+00
```

Remarks:

1. See `GRAV` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-49. Database Gravity Load Vector Definition.

```
                                  C E N T R I F U G A L   L O A D   V E C T O R   D E F I N I T I O N

      SET          GRID         COORDINATE        OMEGA        ALPHA                ROTATION VECTOR
       ID           ID              ID                                        V1            V2            V3
        7            1               1         0.300E+02    0.500E+01      0.000E+00     0.000E+00     0.100E+01
        7            5               1         0.600E+02    0.700E+01      0.000E+00     0.000E+00     0.100E+01
```

Remarks:

1. See `RFORCE` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-50. Database Centrifugal Load Vector Definition.

```
                                  G R I D   P O I N T   T E M P E R A T U R E   D E F I N I T I O N

     SET          GRID        TEMPERATURE
      ID           ID
        3            1         0.100E+02
        3            2         0.150E+04
        3            3         0.150E+04
        3            4         0.100E+02
        7            5         0.250E+04
        7            6         0.250E+04
        7            7         0.250E+04
        7            8         0.250E+04
```

Remarks:

1. See `TEMP`, `TEMPD`, `TEMPP1`, and `TEMPRB` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `2` in the Model Initialization File or on the Nastran command line.

Figure A-51. Database Grid Point Temperature Definition.

```
                              D Y N A M I C   L O A D   S C A L E   F A C T O R   D E F I N I T I O N

     SET          GRID         COMPONENT    AREA SCALE
      ID           ID            NUMBER       FACTOR
      100            1             3         0.100E+01
      200            5             6         0.100E+01
```

Remarks:

1. See `DAREA` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-52. Database Dynamic Load Scale Factor Definition.

```
                                D Y N A M I C    L O A D    T I M E    D E L A Y    D E F I N I T I O N

     SET          GRID        COMPONENT       DELAY
      ID           ID          NUMBER
       30            5            3        0.300E-01
       40           15            3        0.600E-01
```

Remarks:

1. See `DELAY` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `2` in the Model Initialization File or on the Nastran command line.

Figure A-53. Database Dynamic Load Time Delay Definition.

```
                                       D Y N A M I C   L O A D   P H A S E   L E A D   D E F I N I T I O N

          SET            GRID         COMPONENT        PHASE
           ID             ID           NUMBER          LEAD
            45              6              2         4.500E+01
            55              3              1         1.300E+00
```

Remarks:

1. See `DPHASE` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `2` in the Model Initialization File or on the Nastran command line.

Figure A-54. Database Dynamic Load Phase Lead Definition.

```
                                        T R A N S I E N T   T I M E   S T E P   D E F I N I T I O N

     SET          TIME           TIME            SKIP
      ID          STEPS        INCREMENT        FACTOR
       25          600         0.100E-03          1
       30          300         0.200E-03          5
```

Remarks:

1. See `TSTEP` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-55. Database Transient Time Step Definition.

```
                                               F R E Q U E N C Y   L I S T   D E F I N I T I O N

    SET          FREQUENCY
     ID
      25         0.000E+00
                 1.000E+00
                 2.000E+00
                 3.000E+00
                 4.000E+00
                 5.000E+00
                 6.000E+00
                 7.000E+00
                 8.000E+00
                 9.000E+00
                 1.000E+01
```

Remarks:

1. See `FREQ`, `FREQ1`, `FREQ2` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `2` in the Model Initialization File or on the Nastran command line.

Figure A-56. Database Frequency List Definition.

```
                           F R E Q U E N C Y   L I S T   G E N E R A T I O N   D E F I N I T I O N

   SET           MODAL FREQUENCY RANGE    INTERPOLATION    SUBRANGE      CLUSTER
    ID             LOWER        UPPER         TYPE        FREQUENCIES
     25         1.000E+00    1.000E+02       LINEAR           20        1.000E+00
```

Remarks:

1. See `FREQ3` and `FREQ4` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-57. Database Frequency List Generation Definition.

```
                          E L E M E N T   I N I T I A L   D E F O R M A T I O N   D E F I N I T I O N

    SET          ELEMENT      DEFORMATION
     ID            ID
       5            66          0.100E+00
       5            68          0.100E+00
       5           133          0.100E+00
      25            66          0.300E+00
      25            68          0.140E+00
      25           133          0.160E+00
```

Remarks:

1. See `DEFORM` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `2` in the Model Initialization File or on the Nastran command line.

Figure A-58. Database Element Initial Deformation Definition.

```
                               B E A M   E L E M E N T   P R E S S U R E   D E F I N I T I O N

     SET         ELEMENT        LOAD        COORDINATE          POINT-1                       POINT-2
      ID           ID           TYPE            ID        DISTANCE     PRESSURE     DISTANCE     PRESSURE
        4          61        FORCE-Z             0        0.000E+00    0.700E+02    0.100E+01    0.350E+02
        4          61        MOMENT-Z            0        0.000E+00    0.700E+02    0.100E+01    0.350E+02
        4          62        FORCE-X             0        0.170E+00    0.500E+02    0.848E+00    0.500E+02
        6          62        MOMENT-X            0        0.170E+00    0.500E+02    0.848E+00    0.500E+02
        6          63        FORCE-Y             0        0.000E+00    0.000E+00    0.585E+00    0.600E+02
        6          63        MOMENT-Y            0        0.000E+00    0.000E+00    0.585E+00    0.600E+02
```

Remarks:

1. See `PLOAD1` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `2` in the Model Initialization File or on the Nastran command line.

Figure A-59. Database Bar Element Pressure Definition.

```
                               S H E L L   E L E M E N T   P R E S S U R E   D E F I N I T I O N

     SET       ELEMENT               PRESSURE MAGNITUDE                 COORDINATE      ORIENTATION VECTOR
      ID         ID         GRID-1    GRID-2    GRID-3    GRID-4          ID         V1         V2         V3
        4          1      -0.100E+01-0.100E+01-0.100E+01-0.100E+01     NORMAL   0.000E+00 0.000E+00 0.100E+01
        4          2      -0.100E+01-0.100E+01-0.100E+01-0.100E+01     NORMAL   0.000E+00 0.000E+00 0.100E+01
        4          3      -0.100E+01-0.100E+01-0.100E+01-0.100E+01     NORMAL   0.000E+00 0.000E+00 0.100E+01
        8          4      -0.100E+01-0.100E+01-0.100E+01-0.100E+01     NORMAL   0.000E+00 0.000E+00 0.100E+01
        8         36      -0.100E+01-0.100E+01-0.100E+01               NORMAL   0.000E+00 0.000E+00 0.100E+01
        8         37      -0.100E+01-0.100E+01-0.100E+01               NORMAL   0.000E+00 0.000E+00 0.100E+01
```

Remarks:

1. See `PLOAD2` and `PLOAD4` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `2` in the Model Initialization File or on the Nastran command line.

Figure A-60. Database Shell Element Pressure Definition.

```
                            S O L I D   E L E M E N T   P R E S S U R E   D E F I N I T I O N

SET       ELEMENT  FACE GRID-1  FACE GRID-2            PRESSURE MAGNITUDE              COORDINATE      ORIENTATION VECTOR
 ID         ID         ID           ID        GRID-1    GRID-2    GRID-3    GRID-4        ID        V1        V2        V3
   4         69        131          132      0.100E+03 0.120E+03 0.140E+03 0.160E+03     NORMAL   0.000E+00 0.000E+00 0.100E+01
   4         70        132          133      0.100E+03 0.120E+03 0.140E+03 0.160E+03     NORMAL   0.000E+00 0.000E+00 0.100E+01
   8         69        131          132      0.200E+03 0.120E+03 0.140E+03 0.160E+03     NORMAL   0.000E+00 0.100E+00 0.000E+01
   8         70        132          133      0.200E+03 0.120E+03 0.140E+03 0.160E+03     NORMAL   0.000E+00 0.100E+00 0.000E+01
```

Remarks:

1. See `PLOAD4` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-61. Database Solid Element Pressure Definition.

```
                         B E A M   E L E M E N T   T H E R M A L   D E F I N I T I O N

   SET         ELEMENT           TEMPERATURE           GRADIENT DIRECTION-1      GRADIENT DIRECTION-2
    ID            ID          END-A        END-B        END-A        END-B        END-A        END-B
     3            64       0.200E+02    0.200E+02    0.000E+00    0.000E+00    0.000E+00    0.000E+00
     3            64       0.250E+02    0.100E+02    0.000E+00    0.000E+00    0.000E+00    0.000E+00
     6            65       0.200E+02    0.200E+02    0.000E+00    0.100E+00    0.000E+00    0.200E+00
    12            65       0.250E+02    0.100E+02    0.000E+00    0.100E+00    0.000E+00    0.200E+00
```

Remarks:

1. See `TEMPRB` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-62. Database Bar Element Thermal Definition.

```
                              S H E L L   E L E M E N T   T H E R M A L   D E F I N I T I O N

     SET         ELEMENT      TEMPERATURE   GRADIENT
      ID           ID
        3           25         0.100E+02    0.500E+00
        3           26         0.100E+02    0.500E+00
       15           25         0.150E+02    0.600E+00
       15           26         0.150E+02    0.600E+00
```

Remarks:

1. See `TEMPP1` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-63. Database Shell Element Thermal Definition.

```
                                   S T A T I C   L O A D   A D D I T I O N   D E F I N I T I O N

    SET          SET SCALE     LOAD SCALE    LOAD SET    LOAD SCALE    LOAD SET
     ID           FACTOR         FACTOR         ID         FACTOR         ID
      10         0.100E+01      0.200E+01        1       0.300E+01         2
      30         0.100E+01      0.800E+01        2
      50         0.200E+01      0.100E+01        7
```

Remarks:

1. See `LOAD` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `2` in the Model Initialization File or on the Nastran command line.

Figure A-64. Database Static Load Addition Definition.

```
                     T R A N S I E N T   R E S P O N S E   D Y N A M I C   L O A D   D E F I N I T I O N

    SET      AREA SET   DELAY SET   EXCITATION      TABLE    TIME-1      TIME-2     FREQUENCY     PHASE     EXPONENTIAL    GROWTH
     ID         ID         ID          TYPE           ID    CONSTANT    CONSTANT                  ANGLE     COEFFICIENT  COEFFICIENT
      12        200         12     FORCE/MOMENT        2
      11        100                FORCE/MOMENT             0.000E+00   0.100E+00   0.318E+03   0.000E+00   0.000E+00    0.000E+00
```

Remarks:

1. See `TLOAD1` and `TLOAD2` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `2` in the Model Initialization File or on the Nastran command line.

Figure A-65. Database Transient Response Dynamic Load Definition.

```
                         F R E Q U E N C Y   R E S P O N S E   D Y N A M I C   L O A D   D E F I N I T I O N

     SET        AREA SET        DELAY SET    PHASE SET     TABLE-1      TABLE-2
      ID           ID              ID           ID            ID           ID
      101          101               5                          3
      102          102                            10            3
```

Remarks:

1. See `RLOAD1` and `RLOAD2` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-66. Database Frequency Response Dynamic Load Definition.

```
                      P O W E R   S P E C T R A L   D E N S I T Y   D E F I N I T I O N

   SET            SUBCASE ID          COMPLEX COEFFICIENT       TABLE
    ID        EXCITED    APPLIED        REAL      IMAGINARY       ID
    200           1          1       1.000E+00   0.000E+00        40
    200           2          2       1.000E+00   0.000E+00        50
    200           1          2       1.000E+00   0.000E+00        60
```

Remarks:

1. See `RANDPS` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `2` in the Model Initialization File or on the Nastran command line.

Figure A-67. Database Power Spectral Density Definition.

```
                          P O W E R   S P E C T R A L   D E N S I T Y   T A B L E   D E F I N I T I O N

     TABLE        FREQUENCY     DENSITY
      ID
       40         0.000E+00    0.000E+00
                  9.990E+00    0.000E+00
                  1.000E+01    1.000E+00
                  9.000E+01    1.000E+00
                  9.001E+01    0.000E+00
                  1.000E+02    0.000E+00
```

Remarks:

1. See `TABRND1` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-68. Database Power Spectral Density Definition.

```
                                        S T A T I C   L O A D   S E Q U E N C E   D E F I N I T I O N

     SET        AREA SET          LOAD SET IDS
      ID           ID         STATIC       THERMAL
       10          100            1            0
       10          200            2            0
       20          100            1           10
       20          200            2           10
```

Remarks:

1. See `LSEQ` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `2` in the Model Initialization File or on the Nastran command line.

Figure A-69. Database Static Load Sequence Definition.

```
                                    G R I D   P O I N T   S C A L A R   L O A D   D E F I N I T I O N

     SET          GRID           LOAD
      ID           ID
       1            18        6.500E-01
       1            18        7.500E-01
       1            19        6.400E-01
       2            42        8.100E-01
       2            48        6.500E-01
       2            54        6.500E-01
```

Remarks:

1. See `SLOAD` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `2` in the Model Initialization File or on the Nastran command line.

Figure A-70. Database Grid Point Scalar Load Definition.

```
        L I N E   A N D   P O I N T   G R I D   P O I N T   B O U N D A R Y   H E A T   F L U X   D E F I N I T I O N

    SET        SURFACE      HEAT        AREA        GRID-1      GRID-2
     ID         TYPE        FLUX       FACTOR         ID          ID
      1         LINE      4.000E-02   1.500E-01        27          77
      1         LINE      5.000E-02   1.000E-01        34          89
      1         LINE      5.000E-02   2.000E-01        36          98
      1         LINE      5.000E-02   1.000E-01        25          45
      1         POINT     5.000E-02   4.500E-01        32
      1         POINT     1.000E-02   1.000E-01        33
      2         POINT     2.000E-02   2.750E-01        31
      2         POINT     3.000E-02   1.000E-02        24
```

Remarks:

1. See `QHBDY` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `2` in the Model Initialization File or on the Nastran command line.

Figure A-71. Database Line and Point Grid Point Boundary Heat Flux Definition.

```
                 S U R F A C E   G R I D   P O I N T   B O U N D A R Y   H E A T   F L U X   D E F I N I T I O N

   SET        SURFACE      HEAT        GRID-1      GRID-2      GRID-3      GRID-4      GRID-5      GRID-6      GRID-7      GRID-8
    ID         TYPE        FLUX          ID          ID          ID          ID          ID          ID          ID          ID
     1         AREA3     1.000E+02         11          12          25
     1         AREA3     1.000E+02         15          23          21
     1         AREA4     2.500E+02         45          22          30          66
     5         AREA4     2.500E+02         51          32          20          54
     5         AREA6     4.000E+02         67          49          24          69          93          87
     5         AREA6     4.000E+02        121          94          78          99         143         111
```

Remarks:

1. See `QHBDY` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-72. Database Surface Grid Point Boundary Heat Flux Definition.

```
                         E L E M E N T   B O U N D A R Y   H E A T   F L U X   D E F I N I T I O N

SET         ELEMENT                                               HEAT FLUX MAGNITUDE
 ID           ID           GRID-1       GRID-2       GRID-3       GRID-4       GRID-5       GRID-6       GRID-7       GRID-8
  10           30         3.400E-03    1.400E-03    1.400E-03    3.400E-03    3.400E-03    1.400E-03    1.400E-03    3.400E-03
  10           24         4.000E-03    3.000E-03    2.000E-03    1.000E-03    4.000E-03    3.000E-03    2.000E-03    1.000E-03
  10           29         1.200E-02    1.200E-02    1.000E-02    1.000E-02    1.000E-02    1.000E-02
  20           34         4.500E-02    4.500E-02    4.500E-02    4.500E-02    4.500E-02    4.500E-02
  20           22         4.500E-02    4.500E-02    4.500E-02    4.500E-02
  20           23         5.500E-02    5.500E-02    5.500E-02    5.500E-02
```

Remarks:

1. See `QBDY1` and `QBDY2` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-73. Database Element Boundary Heat Flux Definition.

```
                           E L E M E N T   V O L U M E   H E A T   A D D I T I O N   D E F I N I T I O N

     SET         ELEMENT         POWER
      ID            ID          DENSITY
        5             9         1.000E-01
        5             4         1.000E-01
        5            10         1.000E-01
       10            19         5.000E-02
       10            20         5.000E-02
       10            15         2.500E-02
```

Remarks:

1. See `QVOL` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1 - 2` in the Model Initialization File or on the Nastran command line.

Figure A-74. Database Element Volume Heat Addition Definition.

```
                        E L E M E N T   B O U N D A R Y   C O N V E C T I O N   D E F I N I T I O N

ELEMENT     MATERIAL      FILM       AMBIENT-1  AMBIENT-2  AMBIENT-3  AMBIENT-4  AMBIENT-5  AMBIENT-6  AMBIENT-7  AMBIENT-8
  ID           ID       GRID ID       GRID ID    GRID ID    GRID ID    GRID ID    GRID ID    GRID ID    GRID ID    GRID ID
   21            1           18          101        101        101        101        101        101        101        101
   22            1           27          101        101        101        101        101        101        101        101
   23            1                       101        101        101        101
   25            2                       100        110        120        130
   27            2                       140        150        160        170
   28            2                       102        102        102        102
```

Remarks:

1. See `CONV` and `PCONV` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `2` in the Model Initialization File or on the Nastran command line.

Figure A-75. Database Element Boundary Convection Definition.

```
                                   S U B C A S E   V E C T O R   O U T P U T   S E T   D E F I N I T I O N

        SUBCASE          LOAD          DISPLACEMENT         VELOCITY        ACCELERATION        SPC FORCE         MPC FORCE
          ID            VECTOR            VECTOR             VECTOR            VECTOR             VECTOR            VECTOR
          101             ALL               ALL                ALL               102                ALL               ALL
          105             234               238               NONE               ALL                ALL              NONE
          110             ALL               ALL               NONE              NONE                ALL              NONE
```

Remarks:

1. See `OLOAD`, `DISPLACEMENT`, `VELOCITY`, `ACCELERATION`, `SPCFORCES`, and `MPCFORCES` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for command format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `5` in the Model Initialization File or on the Nastran command line.

Figure A-76. Database Subcase Vector Output Set Definition.

```
                            S U B C A S E   V E C T O R   O U T P U T   S E T   D E F I N I T I O N

   SUBCASE          LOAD          TEMPERATURE       SPC FORCE        MPC FORCE
     ID            VECTOR           VECTOR           VECTOR           VECTOR
      15             ALL              ALL              ALL             NONE
      25             ALL              ALL              ALL             NONE
      35            NONE              ALL              ALL             NONE
```

Remarks:

1. See `OLOAD`, `THERMAL`, `SPCFORCES`, and `MPCFORCES` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for command format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `5` in the Model Initialization File or on the Nastran command line.

Figure A-77. Database Subcase Vector Output Set Definition (Heat Transfer Analysis).

```
                           S U B C A S E   E L E M E N T   O U T P U T   S E T   D E F I N I T I O N

   SUBCASE          ELEMENT           ELEMENT           ELEMENT           ELEMENT
     ID              FORCE            STRESS            STRAIN        STRAIN ENERGY
     101              ALL               ALL              NONE              ALL
     104              ALL               ALL              NONE              123
     107              334               456              NONE              ALL
```

Remarks:

1. See `FORCE`, `STRESS`, `STRAIN`, and `ESE` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for command format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `5` in the Model Initialization File or on the Nastran command line.

Figure A-78. Database Subcase Element Output Set Definition.

```
                       S U B C A S E   E L E M E N T   O U T P U T   S E T   D E F I N I T I O N

   SUBCASE          ELEMENT          ELEMENT
     ID               FLUX           GRADIENT
      10               ALL             ALL
      20               105             105
      30               ALL             ALL
```

Remarks:

1. See `FLUX` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for command format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `5` in the Model Initialization File or on the Nastran command line.

Figure A-79. Database Subcase Element Output Set Definition (Heat Transfer Analysis).

```
                              S U B C A S E   G R I D   O U T P U T   S E T   D E F I N I T I O N

   SUBCASE        GRID POINT      GRID POINT       GRID POINT    GRID POINT STRESS
     ID             FORCE           STRESS           STRAIN        DISCONTINUITY
     101             ALL              ALL             NONE              ALL
     104              20             NONE             NONE               34
     107             ALL              ALL             NONE              ALL
```

Remarks:

1. See `GPFORCE`, `GPSTRESS`, `GPSTRAIN`, `AND` `GPDISCONT` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for command format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `5` in the Model Initialization File or on the Nastran command line.

Figure A-80. Database Subcase Grid Output Set Definition.

```
                                          S E T   D E F I N I T I O N

    SET                                        ELEMENT/GRID IDS
     ID
      1        ALL
      5           1         THRU          44
      6         101          104         107
      2           1            3           4            6         THRU           18           38           41           44
                 47           50          53           58           59           64           69           84           86
                 88           90         115          117          119          121          131         THRU          159
                161          162         164          165          167          168          170          171          173
                174          176         177          179          180          182          183
      3          79           83          87           91
```

Remarks:

1. See `SET` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for command format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `5` in the Model Initialization File or on the Nastran command line.

Figure A-81. Database Set Definition.

```
                                        V O L U M E   D E F I N I T I O N

   VOLUME      ELEMENT SET     COORDINATE
     ID            ID              ID
      20            2               2
      30            5         MATERIAL
      40            9             GRID
```

Remarks:

1. See `VOLUME` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for command format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `5` in the Model Initialization File or on the Nastran command line.

Figure A-82. Database Volume Definition.

```
                                                   S U R F A C E   D E F I N I T I O N

    SURFACE    ELEMENT SET     COORDINATE       X-AXIS       NORMAL
      ID           ID              ID
       10           5                0             X            Z
       20           7               30             R            Z
       30           6             GRID             Z            X
```

Remarks:

1. See `SURFACE` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for command format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `5` in the Model Initialization File or on the Nastran command line.

Figure A-83. Database Surface Definition.

```
                 S P R I N G   E L E M E N T   S T R E S S   R E C O V E R Y   P R O P E R T Y   D E F I N I T I O N

   PROPERTY          S
      ID
       70         0.100E+01
       71         0.100E+01
       72         0.100E+01
       73         0.200E+01
```

Remarks:

1. See `CELAS1` and `CELAS2` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `4` or `6` - `7` in the Model Initialization File or on the Nastran command line.

Figure A-84. Database Spring Element Stress Recovery Property Definition.

```
                         R O D   E L E M E N T   S T R E S S   R E C O V E R Y   P R O P E R T Y   D E F I N I T I O N

   PROPERTY           C
      ID
       70         0.000E+00
       80         0.450E+00
       90         0.850E+00
      100         0.150E+01
```

Remarks:

1. See `PROD` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `4` or `6` - `7` in the Model Initialization File or on the Nastran command line.

Figure A-85. Database Rod Element Stress Recovery Property Definition.

```
                    B A R   E L E M E N T   A X I S   O U T P U T   L O C A T I O N   D E F I N I T I O N

ELEMENT           X1           X2           X3           X4           X5           X6           X7           X8           X9
  ID
   61        0.1000E+00   0.2000E+00   0.3000E+00   0.4000E+00   0.5000E+00   0.6000E+00   0.7000E+00   0.8000E+00   0.9000E+00
   62        0.2000E+00   0.4000E+00   0.6000E+00   0.8000E+00
```

Remarks:

1. See `CBARAO` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `4` or `6` - `7` in the Model Initialization File or on the Nastran command line.

Figure A-86. Database Bar Element Axis Output Location Definition.

```
                B A R   E L E M E N T   S T R E S S   R E C O V E R Y   P R O P E R T Y   D E F I N I T I O N

PROPERTY           C1            C2            D1            D2            E1            E2            F1            F2
   ID
    30         0.000E+00     0.000E+00     0.000E+00     0.000E+00     0.000E+00     0.000E+00     0.000E+00     0.000E+00
    40         0.100E+00     0.200E+00     0.000E+00    -0.100E+00     0.100E+00    -0.300E+00     0.600E+00     0.200E+00
    50         0.200E+00     0.400E+00     0.000E+00    -0.200E+00     0.200E+00    -0.400E+00     0.800E+00     0.300E+00
    60         0.000E+00     0.000E+00     0.000E+00     0.000E+00     0.000E+00     0.000E+00     0.000E+00     0.000E+00
```

Remarks:

1. See `PBAR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `4` or `6` - `7` in the Model Initialization File or on the Nastran command line.

Figure A-87. Database Bar Element Stress Recovery Property Definition.

```
                 B E A M   E L E M E N T   S T R E S S   R E C O V E R Y   P R O P E R T Y   D E F I N I T I O N

 PROPERTY  DISTANCE     C1          C2          D1          D2          E1          E2          F1          F2           S1          S2
    ID
    11      0.0000  -0.250E+01  0.500E+00 -0.250E+01 -0.500E+00  0.250E+01 -0.500E+00  0.250E+01  0.500E+00   0.000E+00  0.000E+00
            1.0000  -0.250E+01  0.500E+00 -0.250E+01 -0.500E+00  0.250E+01  0.500E+00  0.250E+01 -0.500E+00
    12      0.0000  -0.250E+01  0.500E+00 -0.250E+01 -0.500E+00  0.250E+01 -0.500E+00  0.250E+01  0.500E+00   0.100E+00  0.200E+00
            1.0000  -0.250E+01  0.500E+00 -0.250E+01 -0.500E+00  0.250E+01  0.500E+00  0.250E+01 -0.500E+00
```

Remarks:

1. See `PBEAM` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `4` or `6` - `7` in the Model Initialization File or on the Nastran command line.

Figure A-88. Database Beam Element Stress Recovery Property Definition.

```
                S H E L L   E L E M E N T   S T R E S S   R E C O V E R Y   P R O P E R T Y   D E F I N I T I O N

  PROPERTY          Z1            Z2
     ID
      40        -0.500E-01    0.150E+00
      50        -0.100E+00    0.100E+00
      90        -0.140E+00    0.140E+00
     100        -0.500E-01    0.500E-01
```

Remarks:

1. See `PSHELL` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `4` or `6` - `7` in the Model Initialization File or on the Nastran command line.

Figure A-89. Database Shell Element Stress Recovery Property Definition.

```
                                R E S U L T   L I M I T S   D E F I N I T I O N

    SET        SUBCASE SET      ELEMENT     OUTPUT SET     OUTPUT SET       RESULT
     ID            ID             TYPE          ID            TYPE          NUMBER
      1             1            SHELL           5          ELEMENT            4
      2             1            SHELL           5          ELEMENT            5
      3             1            SHELL           5          ELEMENT            6
      7             6            SHELL           5             GRID            4
      8             6            SHELL           5             GRID            5
     12             6            SHELL           5             GRID           15
```

Remarks:

1. See `RESULTLIMITS` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for command format.
2. This output is requested by specifying `MODLDATAOUT = ON` and `MODLDATAFORMAT` = `1` - `4` or `6` - `7` in the Model Initialization File or on the Nastran command line.

Figure A-90. Database Result Limits Definition.

```
                                        M O D E L   P A R A M E T E R   D E F I N I T I O N



ELEMENT STIFFNESS MATRIX FORMULATION PARAMETERS

HEX ELEMENT EDGE NODE                             = OFF
HEX ELEMENT INTERNAL NODE                         = ON
HEX ELEMENT REDUCED ORDER INTEGRATION             = ON
QUAD ELEMENT VERTEX ROTATION                      = OFF
QUAD ELEMENT INTERNAL NODE                        = ON
QUAD ELEMENT MEMBRANE REDUCED ORDER INTEGRATION   = ON
QUAD ELEMENT BENDING REDUCED ORDER INTEGRATION    = ON
TRI ELEMENT VERTEX ROTATION                       = OFF
TRI ELEMENT MEMBRANE REDUCED ORDER INTEGRATION    = ON
TRI ELEMENT BENDING REDUCED ORDER INTEGRATION     = ON



ELEMENT LOAD VECTOR FORMULATION PARAMETERS

QUAD ELEMENT EQUIVALENT LOAD VECTOR FORMULATION   = ON
TRI ELEMENT EQUIVALENT LOAD VECTOR FORMULATION    = ON
BAR ELEMENT EQUIVALENT LOAD VECTOR FORMULATION    = ON



MODEL SOLUTION SEQUENCE PARAMETERS

STIFFNESS MATRIX AUTOMATIC SINGLE POINT CONSTRAINT                                 = ON
STIFFNESS MATRIX DIAGONAL RATIO TOLERANCE FOR AUTOMATIC SINGLE POINT CONSTRAINT   = 0.100000E-07
STIFFNESS MATRIX FACTOR DIAGONAL RATIO TOLERANCE FOR MECHANISM DETECTION           = 0.100000E+06
FLOATING POINT PRECISION CONSTANT FOR STIFFNESS MATRIX FACTORIZATION               = 0.222045E-15
MODEL RESULTS FLOATING POINT ZERO TOLERANCE                                        = 0.222045E-15
```

Remarks:

1. See the *Nastran Solver Reference Guide*, Section 5, *Parameters,* for a detailed description of each parameter.
2. This output is requested by specifying `MODLDATAOUT = ON` in the Model Initialization File or on the Nastran command line.

Figure A-91. Database Model Parameter Definition.

```
                                             M O D E L   D A T A B A S E   S I Z E



 MODEL DATABASE SIZE

 SUBCASES                     =        2
 COORDINATE SYSTEM            =        6
 GRID POINTS                  =       93
 ELEMENTS                     =      148
 ELEMENT PROPERTIES           =       19
 MATERIAL PROPERTIES          =       12
 TABLES                       =        8
 SINGLE POINT CONSTRAINTS     =       13
 MULTIPOINT CONSTRAINTS       =      104
 GRID POINT FORCES            =        6
 GRAVITY LOADS                =        2
 GRID POINT TEMPERATURES      =      287
 ELEMENT PRESSURES            =       15
 ELEMENT TEMPERATURES         =        6
 ELEMENT INITIAL DEFORMATIONS =        6
 SETS                         =        6
 SURFACES                     =        1
 VOLUMES                      =        1
```

Figure A-92. Model Database Size.

```
INITIAL PERFORMANCE DATA

MODEL SIZE      =        405 DEGREES OF FREEDOM
MATRIX SIZE     =      82215 WORDS      0.63 MEGA BYTES
SEMIBANDWIDTH   =        405 WORDS
PROFILE DENSITY =      11.01 PERCENT


RESEQUENCED PERFORMANCE DATA

MODEL SIZE      =        405 DEGREES OF FREEDOM
MATRIX SIZE     =      27405 WORDS      0.21 MEGA BYTES
SEMIBANDWIDTH   =         90 WORDS
PROFILE DENSITY =      65.01 PERCENT

MAXIMUM CONNECTIVITY =  26  AT GRID 50
MINIMUM CONNECTIVITY =   1  AT GRID 93
AVERAGE CONNECTIVITY =   9.81
```

Remarks:

1. See `RESEQGRID`, `UNRESEQGRID`, `RESEQSTARTGRID`, and `RESEQGRIDMETHOD` in the *Nastran Solver Reference Guide*, Section 5, *Parameters*.
2. Model size is based on three degrees of freedom for solid elements and six degrees of freedom for all other elements.
3. Matrix size is an upper limit and does not account for sparse matrix storage. In other words, this value includes zero terms that are removed when the matrix is stored on disk.
4. One word equals eight bytes on 32-bit operating systems.
5. This output is not applicable to sparse solutions which do not perform grid point resequencing.

Figure A-93. Grid Point Resequencer Output.

```
ELEMENT GEOMETRY STATISTICS

MAXIMUM HEX ELEMENT FACE WARPING ANGLE  =   0.00 DEGREES ON ELEMENT 28
MAXIMUM HEX ELEMENT FACE SKEW ANGLE     =   0.00 DEGREES ON ELEMENT 28
MAXIMUM HEX ELEMENT FACE INTERIOR ANGLE =  90.00 DEGREES ON ELEMENT 28
MINIMUM HEX ELEMENT FACE INTERIOR ANGLE =  90.00 DEGREES ON ELEMENT 28

MAXIMUM HEX ELEMENT FACE TAPER RATIO =   0.00 ON ELEMENT 28
MAXIMUM HEX ELEMENT ASPECT RATIO     =   2.00 ON ELEMENT 28

MAXIMUM PENT ELEMENT FACE WARPING ANGLE  =   0.00 DEGREES ON ELEMENT 30
MAXIMUM PENT ELEMENT FACE SKEW ANGLE     =  45.00 DEGREES ON ELEMENT 30
MAXIMUM PENT ELEMENT FACE INTERIOR ANGLE =  90.00 DEGREES ON ELEMENT 30
MINIMUM PENT ELEMENT FACE INTERIOR ANGLE =  26.57 DEGREES ON ELEMENT 30

MAXIMUM PENT ELEMENT FACE TAPER RATIO =   0.00 ON ELEMENT 30
MAXIMUM PENT ELEMENT ASPECT RATIO     =   1.15 ON ELEMENT 30

MAXIMUM TET ELEMENT FACE SKEW ANGLE     =  45.00 DEGREES ON ELEMENT 129
MAXIMUM TET ELEMENT FACE INTERIOR ANGLE =  90.00 DEGREES ON ELEMENT 129
MINIMUM TET ELEMENT FACE INTERIOR ANGLE =  26.57 DEGREES ON ELEMENT 129

MAXIMUM TET ELEMENT ASPECT RATIO =   1.61 ON ELEMENT 129

MAXIMUM QUAD ELEMENT WARPING ANGLE  =   0.00 DEGREES ON ELEMENT 20
MAXIMUM QUAD ELEMENT SKEW ANGLE     =   0.00 DEGREES ON ELEMENT 20
MAXIMUM QUAD ELEMENT INTERIOR ANGLE =  90.00 DEGREES ON ELEMENT 134
MINIMUM QUAD ELEMENT INTERIOR ANGLE =  90.00 DEGREES ON ELEMENT 134

MAXIMUM QUAD ELEMENT TAPER RATIO  =   0.00 ON ELEMENT 20
MAXIMUM QUAD ELEMENT ASPECT RATIO =   1.00 ON ELEMENT 20
```

Figure A-94. Element Geometry Statistics.

```
GLOBAL STIFFNESS MATRIX ASSEMBLY STATISTICS

SPARSE MATRIX SIZE =     667276 WORDS        7.6 MEGABYTES
MEMORY ALLOCATED   =    1039263 WORDS        7.9 MEGABYTES

MAXIMUM GLOBAL STIFFNESS MATRIX TERM ZEROED =  9.9494E-16
MINIMUM GLOBAL STIFFNESS MATRIX TERM ZEROED =  0.0000E+00
REDUCTION IN GLOBAL STIFFNESS MATRIX SIZE   =  29.79 PERCENT

ASSEMBLY TIME FOR 4180 ELEMENTS = 9.2 SECONDS
```

Remarks:

1. The output timing format can be changed to hours/minutes/seconds by setting the Model Initialization directive, `SECONDS`, to `OFF`. See the *Nastran Solver Reference Guide*, Section 2, *Initialization,* for more information.
2. One word equals eight bytes on 32-bit operating systems.

Figure A-95. Global Stiffness Matrix Assembly Statistics.

```
DIAGONAL MASS MATRIX FORMULATION USED

ASSEMBLY TIME FOR 64816 ELEMENTS = 62.2 SECONDS
```

Remarks:

1. The output timing format can be changed to hours/minutes/seconds by setting the Model Initialization directive, `SECONDS`, to `OFF`. See the *Nastran Solver Reference Guide*, Section 2, *Initialization,* for more information.
2. The mass matrix formulation method can be controlled using `PARAM`, `COUPMASS`. See the *Nastran Solver Reference Guide*, Section 5, *Parameters,* for more information.

Figure A-96. Global Mass Matrix Assembly Statistics.

```
THE FOLLOWING DEGREES OF FREEDOM HAVE FACTOR DIAGONAL RATIOS GREATER THAN    0.10000E+06
OR HAVE NEGATIVE TERMS ON THE FACTOR DIAGONAL

   GRID ID                COMPONENT          FACTOR DIAGONAL RATIO           MATRIX DIAGONAL
       54                     3                   0.44705E+07                  0.22335E+08

GLOBAL STIFFNESS MATRIX FACTORIZATION STATISTICS

NUMBER OF NEGATIVE TERMS ON FACTOR DIAGONAL = 0
MAXIMUM MATRIX FACTOR DIAGONAL RATIO = 3.322E+02 AT GRID 1903 COMPONENT 1

FACTORED SPARSE MATRIX SIZE =    5304489 WORDS       40.5 MEGABYTES
ADDITIONAL MEMORY ALLOCATED =    7108777 WORDS       54.2 MEGABYTES

FACTORIZATION TIME FOR 5304489 WORDS = 16.4 SECONDS
```

Remarks:

1. Excessive factor diagonal ratios are output during stiffness matrix factorization (decomposition). A value greater than 1.0E+5 indicates a possible mechanism at that degree of freedom and should be investigated.
2. Only factor diagonal ratios greater than the parameter `FACTRATIOTOL/MAXRATIO` (default 1.0E+5) are output. See `FACTRATIOTOL` or `MAXRATIO` in the *Nastran Solver Reference Guide*, Section 5, *Parameters*.
3. For `STATIC` solutions, singularities and negative factor diagonal terms detected during decomposition will result in program termination unless `PARAM`, `SOLUTIONERROR`, `ON` is specified in the Model Input File. Singularities and negative factor diagonal terms indicate a problem with the model, possibly a lack of constraint. See `SOLUTIONERROR` and `FACTDIAG` in the *Nastran Solver Reference Guide*, Section 5, *Parameters*.
4. The output timing format can be changed to hours/minutes/seconds by setting the Model Initialization directive, `SECONDS`, to `OFF`. See the *Nastran Solver Reference Guide*, Section 2, *Initialization,* for more information.

Figure A-97. Global Stiffness Matrix Factorization Statistics.

```
EXTRACTION TIME FOR 20 MODES = 0.8 SECONDS

GLOBAL STIFFNESS MATRIX FACTORIZATION STATISTICS

NUMBER OF NEGATIVE TERMS ON FACTOR DIAGONAL = 20
MAXIMUM MATRIX FACTOR DIAGONAL RATIO = 4.434E+01 AT GRID 33 COMPONENT 3

REORDERING METHOD REQUESTED = AUTO
REORDERING METHOD USED      = VRM1

FACTORED SPARSE MATRIX SIZE =     117589 WORDS        0.9 MEGABYTES
ADDITIONAL MEMORY ALLOCATED =     290889 WORDS        2.2 MEGABYTES

STURM SEQUENCE DATA FOR EIGENVALUE EXTRACTION

TRIAL EIGENVALUE = 1.400684E+10, CYCLES = 1.883606E+04
NUMBER OF EIGENVALUES BELOW THIS VALUE = 20

FACTORIZATION TIME FOR 117589 WORDS = 0.0 SECONDS

REAL EIGENVALUE EXTRACTION STATISTICS

NUMBER OF ITERATION VECTORS USED = 33
NUMBER OF FACTORIZATIONS         = 3
NUMBER OF SUBSPACE ITERATIONS    = 22
NUMBER OF SOLVES                 = 726
NUMBER OF ROOTS FOUND            = 20
```

Remarks:

1. A valid solution is indicated by a minimum orthogonality loss less than 1.0E-6, a minimum error norm less than 1.0E-3, and no missed eigenvalues (Sturm sequence check).
2. Eigenvalue extraction is controlled using the `EIGRL` Bulk Data entry. See the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for more information.
3. The output timing format can be changed to hours/minutes/seconds by setting the Model Initialization directive, `SECONDS`, to `OFF`. See the *Nastran Solver Reference Guide*, Section 2, *Initialization,* for more information.

Figure A-98. Eigenvalue Extraction Statistics.

```
                                                 R E A L   E I G E N V A L U E S

     MODE         EIGENVALUE       RADIANS          CYCLES       GENERALIZED     GENERALIZED     ORTHOGONALITY      ERROR
    NUMBER                                                          MASS          STIFFNESS          LOSS          MEASURE
       1         8.940697E-11    9.455526E-06    1.504894E-06    1.000000E+00    8.940697E-08    0.000000E+00    5.203658E-15
       2         1.087785E-09    3.298158E-05    5.249182E-06    1.000000E+00    1.087785E-06    7.400298E-17    4.724788E-15
       3         1.475215E-09    3.840853E-05    6.112907E-06    1.000000E+00    1.475215E-06    2.020404E-16    2.324607E-15
       4         3.531575E-09    5.942706E-05    9.458110E-06    1.000000E+00    3.531575E-06    2.655955E-19    1.034285E-14
       5         3.717840E-09    6.097409E-05    9.704328E-06    1.000000E+00    3.717840E-06    1.490616E-17    7.200536E-15
       6         5.058944E-09    7.112626E-05    1.132010E-05    1.000000E+00    5.058944E-06    5.076275E-17    1.295316E-14
       7         6.343320E+07    7.964496E+03    1.267589E+03    1.000000E+00    6.343320E+07    2.044111E-16    2.402007E-14
       8         1.548321E+08    1.244315E+04    1.980389E+03    1.000000E+00    1.548321E+08    4.394504E-16    1.309097E-14
       9         4.729638E+08    2.174773E+04    3.461259E+03    1.000000E+00    4.729638E+08    6.144550E-17    2.282439E-15
      10         1.123340E+09    3.351626E+04    5.334279E+03    1.000000E+00    1.123340E+09    1.295926E-16    2.694635E-15

MAXIMUM EIGENVALUE =  1.123340E+09, CYCLES =  5.334279E+03 FOR MODE 10
MINIMUM EIGENVALUE =  8.940697E-11, CYCLES =  1.504894E-06 FOR MODE 1

FIRST FLEXIBLE EIGENVALUE =  6.343320E+07, CYCLES =  1.267589E+03 AT MODE 7

MAXIMUM EIGENVECTOR ORTHOGONALITY LOSS =  4.394504E-16 FOR MODE 8
MINIMUM EIGENVECTOR ORTHOGONALITY LOSS =  2.655955E-19 FOR MODE 4

MAXIMUM EIGENVECTOR ERROR MEASURE =  2.402007E-14 FOR MODE 7
MINIMUM EIGENVECTOR ERROR MEASURE =  2.282439E-15 FOR MODE 9
```

Remarks:

1. An extracted eigenvalue can be considered accurate if its orthogonality loss is less than 1.0E-6 and its error measure is less than 1.0E-3.
2. Eigenvalue extraction is controlled using the `EIGRL` Bulk Data entry. See the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for more information.

Figure A-99. Real Eigenvalue Output.

```
                                   M O D A L   P A R T I C I P A T I O N   F A C T O R S

  MODE           T1             T2             T3             R1             R2             R3
 NUMBER
    1      0.000000E+00  0.178305E-01  0.000000E+00  0.000000E+00  0.000000E+00  0.129354E+00
    2      0.000000E+00  0.988843E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.205835E-01
    3      0.406708E-14 -0.582044E-02  0.000000E+00  0.000000E+00  0.000000E+00 -0.731480E-02
    4      0.713336E-12  0.417804E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.370213E-02
    5     -0.207000E-09 -0.326132E-02  0.000000E+00  0.000000E+00  0.000000E+00 -0.221196E-02
```

Remarks:

1. The modal participation factor for a given mode in a given direction indicates how strongly motion in that direction is represented in the eigenvector.

Figure A-100. Modal Participation Factor Output.

```
                                        R I G I D - B O D Y   E I G E N V A L U E S

       MODE          EIGENVALUE      RIGID-BODY        NORMALIZED
      NUMBER                        STRAIN ENERGY      EIGENVALUE
         1         8.940697E-11    5.761098E-15    1.409466E-18
         2         1.087785E-09    2.614572E-15    1.714851E-17
         3         1.475215E-09    8.321064E-17    2.325620E-17
         4         3.531575E-09    4.633129E-16    5.567393E-17
         5         3.717840E-09    6.075889E-19    5.861031E-17
         6         5.058944E-09    3.894088E-14    7.975231E-17
```

Remarks:

1. Normalized rigid-body eigenvalues are normalized to the first flexible mode.

Figure A-101. Rigid-Body Eigenvalue Output.

```
                                M O D A L   E F F E C T I V E   M A S S

   MODE          T1             T2             T3             R1             R2             R3
  NUMBER
     1      0.000000E+00  0.317925E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.167324E-01
     2      0.000000E+00  0.977810E-04  0.000000E+00  0.000000E+00  0.000000E+00  0.423680E-03
     3      0.000000E+00  0.338775E-04  0.000000E+00  0.000000E+00  0.000000E+00  0.535063E-04
     4      0.000000E+00  0.174560E-04  0.000000E+00  0.000000E+00  0.000000E+00  0.137057E-04
     5      0.000000E+00  0.106362E-04  0.000000E+00  0.000000E+00  0.000000E+00  0.489275E-05


  TOTAL     0.000000E+00  0.477676E-03  0.000000E+00  0.000000E+00  0.000000E+00  0.172282E-01
```

Remarks:

1. If the modal effective masses of all modes are added in any particular direction, the sum should give the total mass of the model, except for mass at restrained degrees of freedom. If the sum is significantly less than the model’s total mass, modes with significant participation in excitation in that direction may not have been extracted.

Figure A-102. Modal Effective Mass Output.

```
                                      R E A L   E I G E N V E C T O R   N U M B E R   1

    GRID      COORDINATE        T1            T2            T3            R1            R2            R3
     ID           ID
       2            0      0.000000E+00  0.147644E+01  0.000000E+00  0.000000E+00  0.000000E+00  0.288181E+01
       3            0      0.000000E+00  0.562172E+01  0.000000E+00  0.000000E+00  0.000000E+00  0.533804E+01
       4            0      0.000000E+00  0.120121E+02  0.000000E+00  0.000000E+00  0.000000E+00  0.737320E+01
       5            0      0.000000E+00  0.202313E+02  0.000000E+00  0.000000E+00  0.000000E+00  0.899831E+01
       6            0      0.000000E+00  0.298786E+02  0.000000E+00  0.000000E+00  0.000000E+00  0.102334E+02
       7            0      0.000000E+00  0.405784E+02  0.000000E+00  0.000000E+00  0.000000E+00  0.111096E+02
       8            0      0.000000E+00  0.519925E+02  0.000000E+00  0.000000E+00  0.000000E+00  0.116705E+02
       9            0      0.000000E+00  0.638333E+02  0.000000E+00  0.000000E+00  0.000000E+00  0.119736E+02
      10            0      0.000000E+00  0.758774E+02  0.000000E+00  0.000000E+00  0.000000E+00  0.120904E+02
      11            0      0.000000E+00  0.879808E+02  0.000000E+00  0.000000E+00  0.000000E+00  0.121077E+02

MAXIMUM DISPLACEMENT MAGNITUDE =  0.879808E+02  AT GRID 11
MAXIMUM ROTATION MAGNITUDE     =  0.121077E+02  AT GRID 11
```

Remarks:

1. This output is requested using the `DISPLACEMENT` Case Control command when running `MODAL` and `BUCKLING` solutions. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. The displacement or global coordinate system for the grid point is indicated in the Coordinate ID column.
3. Components T1, T2, and T3 are translations and R1, R2, and R3 are rotations in the specified coordinate system.

Figure A-103. Real Eigenvector Output.

```
                    G R I D   P O I N T   W E I G H T   O U T P U T


                              REFERENCE POINT = 1

                                     M O
  0.200000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00 -0.200000E+00
  0.000000E+00  0.200000E+00  0.000000E+00  0.000000E+00  0.000000E+00 -0.100000E+01
  0.000000E+00  0.000000E+00  0.200000E+00  0.200000E+00  0.100000E+01  0.000000E+00
  0.000000E+00  0.000000E+00  0.200000E+00  0.266833E+00  0.100000E+01  0.000000E+00
  0.000000E+00  0.000000E+00  0.100000E+01  0.100000E+01  0.666683E+01  0.000000E+00
 -0.200000E+00 -0.100000E+01  0.000000E+00  0.000000E+00  0.000000E+00  0.693333E+01
```

Rigid body mass matrix (MO) relative to the reference point in the basic coordinate system.

```
                                      S
                      0.100000E+01  0.000000E+00  0.000000E+00
                      0.000000E+00  0.100000E+01  0.000000E+00
                      0.000000E+00  0.000000E+00  0.100000E+01
```

Transformation matrix from the basic coordinate system to the principal mass axes.

```
     DIRECTION                                             CENTER OF MASS
 MASS AXIS SYSTEM (S)     MASS                 X             Y             Z
          X           0.200000E+00        0.00000E+00   0.10000E+01   0.00000E+00
          Y           0.200000E+00       -0.50000E+01   0.00000E+00   0.00000E+00
          Z           0.200000E+00       -0.50000E+01   0.10000E+01   0.00000E+00
```

Principal masses and associated center of mass.

```
                                     I(S)
                      0.668333E-01  0.000000E+00  0.000000E+00
                      0.000000E+00  0.166683E+01  0.000000E+00
                      0.000000E+00  0.000000E+00  0.173333E+01
```

Inertia matrix I(S) about the center of mass relative to the principal mass axes.

```
                                     I(Q)
                      0.668333E-01
                                    0.166683E+01
                                                  0.173333E+01
```

Inertia matrix I(Q) about the center of mass relative to the principal inertia axes.

```
                                      Q
                      0.100000E+01  0.000000E+00  0.000000E+00
                      0.000000E+00  0.100000E+01  0.000000E+00
                      0.000000E+00  0.000000E+00  0.100000E+01
```

Transformation matrix [Q] between the S-axes and Q-axes.

Figure A-104. Grid Point Weight Output.

```
                                        P A R T   D E F I N I T I O N

 PROPERTY     MATERIAL       PROPERTY           BOUNDING DIMENSIONS              MASS      VOLUME        GRID      ELEMENTS
    ID           ID            TYPE          D1           D2           D3                                POINTS
      1           1            ROD      4.1900E+02  1.2300E+02  1.0422E+02    3.8330E-01  5.2100E+02       2460         1250
      2           2            ROD      3.8000E+02  1.0900E+02  1.0422E+02    0.0000E+00  1.5135E+01         76           38
      3           1            BAR      4.1850E+02  1.2300E+02  1.0422E+02    1.8817E+00  2.5577E+03       2432         1237
      4           1            BAR      4.1900E+02  1.1181E+02  1.0350E+02    2.6032E+00  3.5384E+03        570          318
      5           3           BEAM      3.8400E+02  1.1800E+02  1.0422E+02    6.2279E+00  8.4653E+03       1428         1394
      6           3           BEAM      4.1850E+02  0.0000E+00  1.0300E+02    8.3522E-01  1.1353E+03        237          236
      7           3           BEAM      3.3600E+02  0.0000E+00  8.8000E+01    2.1555E+00  2.9299E+03        170          168
      8           5           BEAM      2.8400E+02  0.0000E+00  1.0000E+02    6.0881E+00  8.2752E+03        369          380
      9           5           BEAM      0.0000E+00  1.1800E+02  1.0300E+02    3.6288E-01  4.9324E+02         84           82
     10           5           BEAM      4.0400E+02  3.5000E+01  1.0000E+02    2.8093E+00  3.8185E+03        219          209
     11           5           BEAM      3.7200E+02  3.3000E+01  1.0000E+02    1.5244E+00  2.0720E+03        294          280
     12           8           BEAM      4.1700E+02  8.6000E+01  9.8000E+01    9.2676E+00  1.2597E+04        432          412
     13           8           BEAM      3.8422E+02  9.4812E+01  9.5260E+01    2.3988E+00  3.2606E+03        156          154
     14           8           BEAM      3.1200E+02  1.3800E+02  1.0350E+02    2.5100E+00  3.4117E+03        376          368
     15           8           BEAM      0.0000E+00  7.2000E+01  1.0000E+02    6.6507E-01  9.0400E+02        108          104
     16           8           BEAM      3.8400E+02  1.4000E+01  1.0422E+02    2.0857E-01  2.8350E+02        108           90
     17           9          SHELL      4.1700E+02  1.2000E+02  1.0000E+02    2.0614E+01  2.8020E+04      10105         9828
     18           9          SHELL      4.1700E+02  1.2000E+02  1.0000E+02    1.3990E+01  1.9017E+04       3477         3228


  TOTAL                                 4.1900E+02  1.3800E+02  1.0422E+02    7.4526E+01  1.0132E+05      23101        19776
```

Remarks:

1. Part definition output is controlled using `PARAM`, `PARTGEOMOUT`. See the *Nastran Solver Reference Guide*, Section 5, *Parameters,* for more information.
2. Bounding dimensions are in the basic coordinate system.

Figure A-105. Part Definition Output.

```
                                   P A R T   M A S S   P R O P E R T I E S

 PROPERTY    MATERIAL      PROPERTY      MASS             CENTER OF MASS                      MOMENTS OF INERTIA
    ID          ID           TYPE                      X            Y            Z             I1          I2          I3
     1           1            ROD     3.8330E-01   2.1777E+02  5.2844E+01 -5.0382E+01    1.4329E+03  7.0680E+03  6.9394E+03
     2           2            ROD     0.0000E+00   0.0000E+00  0.0000E+00  0.0000E+00    0.0000E+00  0.0000E+00  0.0000E+00
     3           1            BAR     1.8817E+00   2.0635E+02  5.0770E+01 -4.9921E+01    8.3015E+03  3.2565E+04  3.2006E+04
     4           1            BAR     2.6032E+00   1.8845E+02  9.5549E+00 -5.0000E+01    6.2008E+03  3.9415E+04  3.4699E+04
     5           3           BEAM     6.2279E+00   1.9200E+02  6.1000E+01 -5.0000E+01    2.4149E+04  1.0301E+05  9.3331E+04
     6           3           BEAM     8.3522E-01   2.3183E+02  1.2000E+02 -5.0000E+01    2.0532E+03  1.6450E+04  1.4397E+04
     7           3           BEAM     2.1555E+00   1.6800E+02  6.0000E+00 -5.0000E+01    4.1731E+03  2.4458E+04  2.0285E+04
     8           5           BEAM     6.0881E+00   2.1282E+02  1.1700E+02 -5.0000E+01    3.9630E+03  5.0216E+04  4.6253E+04
     9           5           BEAM     3.6288E-01   3.9600E+02  6.1000E+01 -5.0000E+01    1.3840E+03  9.6244E+02  4.2159E+02
    10           5           BEAM     2.8093E+00   1.4082E+02  1.0045E+02 -4.9854E+01    3.1996E+03  5.6223E+04  5.4744E+04
    11           5           BEAM     1.5244E+00   2.4857E+02  6.1786E+01 -5.0000E+01    1.6578E+03  2.1265E+04  2.0370E+04
    12           8           BEAM     9.2676E+00   1.9420E+02  1.2389E+01 -4.9105E+01    1.6590E+04  1.4836E+05  1.5191E+05
    13           8           BEAM     2.3988E+00   1.9893E+02  3.5945E+01 -5.0000E+01    7.6179E+03  3.5160E+04  3.1894E+04
    14           8           BEAM     2.5100E+00   2.2800E+02  5.1008E+01 -5.0000E+01    1.0710E+04  4.2143E+04  3.9410E+04
    15           8           BEAM     6.6507E-01   4.1900E+02  5.9000E+01 -5.0000E+01    1.0348E+03  5.5595E+02  4.7885E+02
    16           8           BEAM     2.0857E-01   1.9200E+02  1.3000E+01 -5.0000E+01    5.7010E+02  3.7701E+03  3.2073E+03
    17           9          SHELL     2.0614E+01   2.3082E+02  6.0000E+01 -5.0000E+01    7.2630E+04  4.0032E+05  3.7723E+05
    18           9          SHELL     1.3990E+01   2.3782E+02  1.8492E+00 -5.0000E+01    1.3021E+04  2.3921E+05  2.2857E+05


  TOTAL                               7.4526E+01   2.1712E+02  4.5330E+01 -4.9883E+01    2.7351E+05  1.3055E+06  1.3353E+06
```

Remarks:

1. Part mass properties output is controlled using `PARAM`, `PARTMASSOUT`. See the *Nastran Solver Reference Guide*, Section 5, *Parameters,* for more information.
2. Center of gravity coordinates are in the basic coordinate system.
3. Moments of inertia are about the center of gravity relative to the principal mass axes.

Figure A-106. Part Mass Properties Output.

```
                              P A R T   E L E M E N T   G E O M E T R Y   S U M M A R Y

PROPERTY     PROPERTY    ASPECT   RECOMMENDED       TAPER   RECOMMENDED        SKEW   RECOMMENDED       WARPING  RECOMMENDED
   ID          TYPE      RATIO       LIMIT          RATIO      LIMIT          ANGLE      LIMIT           ANGLE      LIMIT
    20         QUAD       1.520    100.000          0.500       0.750          0.130      65.000          1.550      45.000
    20          TRI       2.173    100.000          0.770       0.750          1.155      65.000          0.890      45.000
    10          HEX       2.230    100.000          0.000       0.750          0.000      65.000          0.000      45.000
    10         PENT       1.318    100.000          0.130       0.750         36.870      65.000          0.000      45.000
    10          TET       1.412    100.000          0.000       0.750          1.414      80.000          0.000      45.000
```

Remarks:

1. Part element geometry output is controlled using `PARAM`, `ELEMGEOMOUT`. Individual element geometry statistics are also listed by element type and are sorted based on the `ELEMGEOMOUT` setting. See the *Nastran Solver Reference Guide*, Section 5, *Parameters,* for more information.

Figure A-107. Part Element Geometry Summary.

```
                                          G R I D   P O I N T   S I N G U L A R I T Y   T A B L E

    GRID                  STRONGEST                    DIRECTION OF SINGULARITY                      STIFFNESS
     ID                   COMPONENT              T1/R1          T2/R2          T3/R3                   RATIO
      1                       2             0.000000E+00   1.000000E+00   0.000000E+00         0.000000E+00
      1                       3             0.000000E+00   0.000000E+00   1.000000E+00         0.000000E+00
      1                       4             1.000000E+00   0.000000E+00   0.000000E+00         0.000000E+00
      1                       5             0.000000E+00   1.000000E+00   0.000000E+00         0.000000E+00
      1                       6             0.000000E+00   0.000000E+00   1.000000E+00         0.000000E+00
      2                       2             0.000000E+00   1.000000E+00   0.000000E+00         0.000000E+00
      2                       3             0.000000E+00   0.000000E+00   1.000000E+00         0.000000E+00
      2                       4             1.000000E+00   0.000000E+00   0.000000E+00         0.000000E+00
      2                       5             0.000000E+00   1.000000E+00   0.000000E+00         0.000000E+00
      2                       6             0.000000E+00   0.000000E+00   1.000000E+00         0.000000E+00


MAXIMUM STIFFNESS MATRIX DIAGONAL =  1.0000E+04  AT GRID 2 COMPONENT 1
MINIMUM STIFFNESS MATRIX DIAGONAL =  0.0000E+00  AT GRID 2 COMPONENT 6
```

Remarks:

1. This output will be included anytime a stiffness ratio is less than the parameter `STIFFRATIOTOL/EPZERO` (default 1.0E-8). See `STIFFRATIOTOL` or `EPZERO` in the *Nastran Solver Reference Guide*, Section 5, *Parameters*.
2. Automatic stiffness matrix singularity correction can be suppressed by including `PARAM`, `AUTOSPC`, `OFF` in the Model Input File. See `AUTOSPC` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-108. Grid Point Singularity Table.

```
                                   M A S S   M A T R I X   S I N G U L A R I T Y   T A B L E

    GRID                FAILED              MASS
     ID                DIRECTION             RATIO
       1                  6           0.000000E+00
       2                  6           0.000000E+00
       3                  6           0.000000E+00
       4                  6           0.000000E+00
       5                  6           0.000000E+00
       6                  6           0.000000E+00
       7                  6           0.000000E+00
       8                  6           0.000000E+00
       9                  6           0.000000E+00
      10                  6           0.000000E+00


MAXIMUM MASS MATRIX DIAGONAL =  0.8889E-02  AT GRID 15 COMPONENT 3
MINIMUM MASS MATRIX DIAGONAL =  0.0000E+00  AT GRID 18 COMPONENT 6
```

Remarks:

1. This output will be included anytime a mass ratio is less than the parameter `STIFFRATIOTOL/EPZERO` (default 1.0E-8). See `STIFFRATIOTOL` or `EPZERO` in the *Nastran Solver Reference Guide*, Section 5, *Parameters*.
2. Automatic mass matrix singularity correction is enabled by including `PARAM`, `AUTOBPD`, `ON` in the Model Input File. See `AUTOBPD` and `BPDEFDIAG` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-109. Mass Matrix Singularity Table.

```
                                                   L O A D   V E C T O R

     GRID       COORDINATE         T1             T2             T3             R1             R2             R3
      ID            ID
        5            1      0.000000E+00  0.000000E+00 -0.300000E+02  0.000000E+00  0.000000E+00  0.000000E+00
       11            1      0.000000E+00  0.000000E+00 -0.200000E+02  0.000000E+00  0.000000E+00  0.000000E+00
       84            3      0.198030E+00  0.140028E+01  0.141422E+01  0.000000E+00  0.000000E+00  0.000000E+00
       86            3      0.198030E+00  0.140028E+01 -0.141422E+01  0.000000E+00  0.000000E+00  0.000000E+00
      115            2      0.000000E+00  0.000000E+00  0.000000E+00  0.212133E+01  0.212133E+01  0.000000E+00
      121            2      0.000000E+00  0.000000E+00  0.000000E+00 -0.212133E+01  0.212133E+01  0.000000E+00


 MAXIMUM APPLIED FORCE MAGNITUDE  =  0.300000E+02  AT GRID 5
 MAXIMUM APPLIED MOMENT MAGNITUDE =  0.300001E+01  AT GRID 121
```

Remarks:

1. This output is requested using the `OLOAD` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. The displacement or global coordinate system for the grid point is indicated in the Coordinate ID column.
3. Components T1, T2, and T3 are translations and R1, R2, and R3 are rotations in the specified coordinate system.

Figure A-110. Load Vector Output.

```
                                             L O A D   V E C T O R

    GRID          HEAT
     ID           FLOW
      36      3.333333E-02
      38      1.666667E-01
      40      6.666667E-02
      42      1.666667E-01
      44      3.333333E-02
      46      3.333333E-01


 MAXIMUM APPLIED HEAT FLOW =  3.333333E-01  AT GRID 46
```

Remarks:

1. This output is requested using the `OLOAD` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.

Figure A-111. Load Vector Output (Heat Transfer Analysis).

```
                                              D I S P L A C E M E N T   V E C T O R

      GRID      COORDINATE        T1            T2            T3            R1            R2            R3
       ID           ID
        1            1     -0.111078E-04 -0.103480E-05 -0.134126E-04 -0.316660E-05  0.168036E-05 -0.306985E-06
        2            1     -0.111078E-04 -0.118830E-05 -0.142527E-04 -0.316660E-05  0.168036E-05 -0.306985E-06
       10            1     -0.111787E-04 -0.965582E-06 -0.632860E-05  0.249217E-06 -0.468702E-05 -0.697569E-06
       12            1     -0.954637E-05 -0.106435E-05 -0.116879E-04 -0.138968E-05 -0.898107E-05  0.308261E-05
       69            2     -0.262390E-04  0.678990E-05 -0.172376E-05  0.644182E-05 -0.195493E-04 -0.417000E-05
       86            3     -0.784467E-05  0.378310E-04 -0.691947E-05 -0.291616E-05 -0.668634E-05 -0.382609E-04
       88            3     -0.490422E-05 -0.363420E-04  0.636692E-05 -0.932648E-05  0.682732E-05  0.286673E-04
      115            2     -0.271177E-06 -0.200039E-05 -0.954861E-05  0.497369E-04  0.494026E-04 -0.152797E-03


  MAXIMUM DISPLACEMENT MAGNITUDE =  0.392506E-04  AT GRID 86
  MAXIMUM ROTATION MAGNITUDE     =  0.168111E-03  AT GRID 115


  EPSILON       =  0.811677E-14
  STRAIN ENERGY =  0.507373E-03


  SOLUTION TIME FOR 405 DEGREES OF FREEDOM = 0.2 SECONDS
```

Remarks:

1. This output is requested using the `DISPLACEMENT` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. The displacement or global coordinate system for the grid point is indicated in the Coordinate ID column.
3. Components T1, T2, and T3 are translations and R1, R2, and R3 are rotations in the specified coordinate system.
4. Epsilon is a measure of solution accuracy. A solution with an epsilon less than 1.0E-6 can be considered accurate. Values greater than 0.001 indicate excessive numerical ill-conditioning possibly due to a modeling error. Check the model for mechanisms, unconstrained rigid body motion, or unreasonably stiff elements.
5. Strain energy is the work performed by the applied loading.

Figure A-112. Displacement Vector Output.

```
                                      C O M P L E X   D I S P L A C E M E N T   V E C T O R
                                                        (MAGNITUDE/PHASE)

    GRID      COORDINATE        T1            T2            T3            R1            R2            R3
     ID           ID
       6            0      1.337207E-07  1.001223E+00  0.000000E+00  0.000000E+00  0.000000E+00  4.050266E-04
                           1.799997E+02  3.599994E+02  0.000000E+00  0.000000E+00  0.000000E+00  3.594554E+02
       7            0      1.529925E-07  1.001642E+00  0.000000E+00  0.000000E+00  0.000000E+00  4.322463E-04
                           1.799997E+02  3.599991E+02  0.000000E+00  0.000000E+00  0.000000E+00  3.594475E+02
       8            0      1.684971E-07  1.002083E+00  0.000000E+00  0.000000E+00  0.000000E+00  4.479169E-04
                           1.799997E+02  3.599989E+02  0.000000E+00  0.000000E+00  0.000000E+00  3.594411E+02
       9            0      1.798528E-07  1.002535E+00  0.000000E+00  0.000000E+00  0.000000E+00  4.554922E-04
                           1.799997E+02  3.599986E+02  0.000000E+00  0.000000E+00  0.000000E+00  3.594369E+02
      10            0      1.867800E-07  1.002993E+00  0.000000E+00  0.000000E+00  0.000000E+00  4.581617E-04
                           1.799997E+02  3.599984E+02  0.000000E+00  0.000000E+00  0.000000E+00  3.594351E+02
      11            0      1.891082E-07  1.003451E+00  0.000000E+00  0.000000E+00  0.000000E+00  4.585366E-04
                           1.799997E+02  3.599981E+02  0.000000E+00  0.000000E+00  0.000000E+00  3.594348E+02


MAXIMUM DISPLACEMENT MAGNITUDE =  1.003451E+00  AT GRID 11
MAXIMUM ROTATION MAGNITUDE     =  4.585366E-04  AT GRID 11


EPSILON       =  0.811677E-14
STRAIN ENERGY =  0.507373E-03


SOLUTION TIME FOR 405 DEGREES OF FREEDOM = 0.2 SECONDS
```

Remarks:

1. This output is requested using the `DISPLACEMENT` Case Control command in frequency response solutions. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. The REAL or IMAG option requests complex output in rectangular format (real and imaginary). The PHASE option requests complex output in polar format (magnitude and phase) as shown. Phase output is in degrees.
3. Maximums magnitudes reported are determined using MAGNITUDE/PHASE results.

Figure A-113. Complex Displacement Vector Output.

```
                       D I S P L A C E M E N T   V E C T O R   P O W E R   S P E C T R A L   D E N S I T Y

    GRID      COORDINATE        T1            T2            T3            R1            R2            R3
     ID           ID
      1            0      0.000000E+00  9.999961E-05  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      2            0      0.000000E+00  1.047097E-04  0.000000E+00  0.000000E+00  0.000000E+00  2.180307E-07
      3            0      0.000000E+00  1.184294E-04  0.000000E+00  0.000000E+00  0.000000E+00  7.340857E-07
      4            0      0.000000E+00  1.410860E-04  0.000000E+00  0.000000E+00  0.000000E+00  1.371883E-06
      5            0      0.000000E+00  1.729179E-04  0.000000E+00  0.000000E+00  0.000000E+00  2.000010E-06
      6            0      0.000000E+00  2.141029E-04  0.000000E+00  0.000000E+00  0.000000E+00  2.533815E-06
      7            0      0.000000E+00  2.645599E-04  0.000000E+00  0.000000E+00  0.000000E+00  2.932218E-06
      8            0      0.000000E+00  3.239046E-04  0.000000E+00  0.000000E+00  0.000000E+00  3.189857E-06
      9            0      0.000000E+00  3.915265E-04  0.000000E+00  0.000000E+00  0.000000E+00  3.326730E-06
     10            0      0.000000E+00  4.667437E-04  0.000000E+00  0.000000E+00  0.000000E+00  3.377880E-06
     11            0      0.000000E+00  5.489987E-04  0.000000E+00  0.000000E+00  0.000000E+00  3.385214E-06


MAXIMUM DISPLACEMENT MAGNITUDE =  5.489987E-04  AT GRID 11
MAXIMUM ROTATION MAGNITUDE     =  3.385214E-06  AT GRID 11
```

Remarks:

1. This output is requested using the `DISPLACEMENT` Case Control command in random response solutions. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. The PSDF option requests power spectral density function, RMS, and number of positive crossings output. The ATOC option requests autocorrelation function output. RALL requests both PSDF and ATOC output.

Figure A-114. Displacement Vector Power Spectral Density Output.

```
                         D I S P L A C E M E N T   V E C T O R   R E S P O N S E   R M S   V A L U E S

    GRID      COORDINATE        T1             T2             T3             R1             R2             R3
     ID           ID
      1            0      0.000000E+00  8.983649E-02  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
      2            0      0.000000E+00  8.968070E-02  0.000000E+00  0.000000E+00  0.000000E+00  1.287919E-02
      3            0      0.000000E+00  9.173900E-02  0.000000E+00  0.000000E+00  0.000000E+00  2.389674E-02
      4            0      0.000000E+00  1.015759E-01  0.000000E+00  0.000000E+00  0.000000E+00  3.307752E-02
      5            0      0.000000E+00  1.230964E-01  0.000000E+00  0.000000E+00  0.000000E+00  4.046820E-02
      6            0      0.000000E+00  1.558505E-01  0.000000E+00  0.000000E+00  0.000000E+00  4.614519E-02
      7            0      0.000000E+00  1.970648E-01  0.000000E+00  0.000000E+00  0.000000E+00  5.022410E-02
      8            0      0.000000E+00  2.439397E-01  0.000000E+00  0.000000E+00  0.000000E+00  5.287220E-02
      9            0      0.000000E+00  2.942950E-01  0.000000E+00  0.000000E+00  0.000000E+00  5.432289E-02
     10            0      0.000000E+00  3.465664E-01  0.000000E+00  0.000000E+00  0.000000E+00  5.488942E-02
     11            0      0.000000E+00  3.997500E-01  0.000000E+00  0.000000E+00  0.000000E+00  5.497369E-02


MAXIMUM DISPLACEMENT MAGNITUDE =  3.997500E-01  AT GRID 11
MAXIMUM ROTATION MAGNITUDE     =  5.497370E-02  AT GRID 11
```

Remarks:

1. This output is requested using the `DISPLACEMENT` Case Control command in random response solutions. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. The PSDF option requests power spectral density function, RMS, and number of positive crossings output. The ATOC option requests autocorrelation function output. RALL requests both PSDF and ATOC output.

Figure A-115. Displacement Vector RMS Output.

```
                D I S P L A C E M E N T   V E C T O R   N U M B E R   O F   P O S I T I V E   C R O S S I N G S

     GRID      COORDINATE        T1             T2             T3             R1             R2             R3
      ID           ID
        1           0      0.000000E+00  5.502793E+01  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
        2           0      5.096015E-14  5.400281E+01  0.000000E+00  0.000000E+00  0.000000E+00  6.612009E+01
        3           0      1.988516E-13  5.189819E+01  0.000000E+00  0.000000E+00  0.000000E+00  6.632348E+01
        4           0      4.291960E-13  5.143856E+01  0.000000E+00  0.000000E+00  0.000000E+00  6.654465E+01
        5           0      7.194435E-13  5.381554E+01  0.000000E+00  0.000000E+00  0.000000E+00  6.677165E+01
        6           0      1.041181E-12  5.698809E+01  0.000000E+00  0.000000E+00  0.000000E+00  6.698858E+01
        7           0      1.362914E-12  5.953698E+01  0.000000E+00  0.000000E+00  0.000000E+00  6.717817E+01
        8           0      1.653150E-12  6.132775E+01  0.000000E+00  0.000000E+00  0.000000E+00  6.732498E+01
        9           0      1.883479E-12  6.256434E+01  0.000000E+00  0.000000E+00  0.000000E+00  6.741924E+01
       10           0      2.031359E-12  6.343286E+01  0.000000E+00  0.000000E+00  0.000000E+00  6.746150E+01
       11           0      2.082314E-12  6.405878E+01  0.000000E+00  0.000000E+00  0.000000E+00  6.746840E+01


 MAXIMUM DISPLACEMENT MAGNITUDE =  6.405878E+01  AT GRID 11
 MAXIMUM ROTATION MAGNITUDE     =  6.746840E+01  AT GRID 11
```

Remarks:

1. This output is requested using the `DISPLACEMENT` Case Control command in random response solutions. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. The PSDF option requests power spectral density function, RMS, and number of positive crossings output. The ATOC option requests autocorrelation function output. RALL requests both PSDF and ATOC will be output.

Figure A-116. Displacement Vector Number of Positive Crossings Output.

```
                                        T E M P E R A T U R E   V E C T O R

    GRID      TEMPERATURE
     ID
       1      2.250000E+02
       2      2.250000E+02
      19      2.250000E+02
      20      2.250000E+02
      79      2.062500E+02
      80      1.125000E+02


MAXIMUM TEMPERATURE =  2.250000E+02  AT GRID 19


EPSILON       =  1.629364E-16
STRAIN ENERGY =  4.517442E+09


SOLUTION TIME FOR 18 DEGREES OF FREEDOM = 0.8 SECONDS
```

Remarks:

1. This output is requested using the `THERMAL` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. Epsilon is a measure of solution accuracy. A solution with an epsilon less than 1.0E-6 can be considered accurate. Values greater than 0.001 indicate excessive numerical ill-conditioning possibly due to a modeling error. Check the model for mechanisms, unconstrained rigid body motion, or unreasonably stiff elements.
3. Strain energy is the work performed by the applied loading.

Figure A-117. Temperature Vector Output.

```
                                 F O R C E S   O F   S I N G L E - P O I N T   C O N S T R A I N T

     GRID      COORDINATE        T1            T2            T3            R1            R2            R3
      ID           ID
      149            1     -0.619070E+00 -0.458939E+00  0.193836E+01  0.000000E+00  0.000000E+00  0.000000E+00
      152            1     -0.184517E+00 -0.129463E+01  0.464866E+01  0.000000E+00  0.000000E+00  0.000000E+00
      153            1     -0.217605E+00  0.227765E+00  0.939635E+01  0.000000E+00  0.000000E+00  0.000000E+00
      156            1      0.208149E+01  0.665902E-01  0.672048E+01  0.000000E+00  0.000000E+00  0.000000E+00
      157            1      0.116390E+01  0.123305E+01  0.415750E+01  0.000000E+00  0.000000E+00  0.000000E+00
      211            1      0.213363E+01 -0.133050E+01  0.608575E+01  0.000000E+00  0.000000E+00  0.000000E+00


 MAXIMUM SINGLE POINT CONSTRAINT FORCE MAGNITUDE  =  0.940163E+01  AT GRID 153
 MAXIMUM SINGLE POINT CONSTRAINT MOMENT MAGNITUDE =  0.000000E+00  AT GRID 211
```

Remarks:

1. This output is requested using the `SPCFORCES` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. The displacement or global coordinate system for the grid point is indicated in the Coordinate ID column.
3. Components T1, T2, and T3 are translations and R1, R2, and R3 are rotations in the specified coordinate system.

Figure A-118. Single Point Constraint Force Vector Output.

```
                              H E A T   F L O W S   O F   S I N G L E - P O I N T   C O N S T R A I N T

    GRID          HEAT
     ID           FLOW
       1     -1.462241E-01
       2     -2.966635E-01
       3     -1.297938E-01
      37     -1.419115E-01
      43     -2.853445E-01
      49     -1.633632E-01

 MAXIMUM SINGLE POINT CONSTRAINT HEAT FLOW =  2.966635E-01  AT GRID 2
```

Remarks:

1. This output is requested using the `SPCFORCES` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.

Figure A-119. Single Point Constraint Heat Flow Vector Output.

```
                                     F O R C E S   O F   M U L T I P O I N T   C O N S T R A I N T

      GRID       COORDINATE        T1            T2            T3            R1            R2            R3
       ID            ID
         1            0      5.820766E-11  2.762431E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
         2            0      2.910383E-11 -5.524862E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00
         3            0     -1.455192E-10  2.762431E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00


  MAXIMUM MULTIPOINT CONSTRAINT FORCE MAGNITUDE  =  5.524862E+00  AT GRID 2
  MAXIMUM MULTIPOINT CONSTRAINT MOMENT MAGNITUDE =  0.000000E+00  AT GRID 18
```

Remarks:

1. This output is requested using the `MPCFORCES` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. The displacement or global coordinate system for the grid point is indicated in the Coordinate ID column.
3. Components T1, T2, and T3 are translations and R1, R2, and R3 are rotations in the specified coordinate system.

Figure A-120. Multipoint Constraint Force Vector Output.

```
                               H E A T   F L O W S   O F   M U L T I P O I N T   C O N S T R A I N T

    GRID          HEAT
     ID           FLOW
      10     -1.962511E+01
      11      3.925023E+01
      12     -1.962511E+01


 MAXIMUM MULTIPOINT CONSTRAINT HEAT FLOW =  3.925023E+01  AT GRID 11
```

Remarks:

1. This output is requested using the `MPCFORCES` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.

Figure A-121. Multipoint Constraint Heat Flow Vector Output.

```
                                        L O A D   V E C T O R   R E S U L T A N T

   SUBCASE          T1            T2            T3            R1            R2            R3
     ID
      101     -0.582077E-09  0.400001E+01  0.500000E+02 -0.626186E+04  0.185852E+05  0.988000E+05
      102      0.948285E+01  0.533385E+01  0.882526E+02 -0.621339E+04  0.185289E+05  0.988574E+05
      103     -0.797546E+03 -0.424002E+03 -0.505244E-07  0.277984E+02  0.319797E+04  0.483753E+03
      104     -0.831220E-06  0.278032E-02  0.128755E-04 -0.720179E+04  0.237968E+04  0.606118E+03
```

Remarks:

1. Resultant of the applied loads about the reference point specified using `PARAM`, `GRDPNT` (default is the origin of the basic coordinate system).
2. Components T1, T2, and T3 are translations and R1, R2, and R3 are rotations in the basic coordinate system.
3. One line is output for each subcase.

Figure A-122. Load Vector Resultant.

```
                      S I N G L E   P O I N T   C O N T R A I N T   V E C T O R   R E S U L T A N T

SUBCASE          T1            T2            T3            R1            R2            R3
  ID
   101     -0.581845E-09 -0.298959E-05 -0.499629E-07 -0.629791E+04  0.185957E+05  0.987999E+05
   102     -0.579865E-09 -0.299413E-05 -0.501999E-07 -0.626033E+04  0.185031E+05  0.988513E+05
   103     -0.831205E-06  0.278032E-02  0.128755E-04 -0.720179E+07  0.237968E+04  0.606118E+03
   104     -0.831043E-06  0.278027E-02  0.128616E-04 -0.720364E+07  0.237962E+04  0.608919E+03
```

Remarks:

1. Resultant of the single point constraint forces about the reference point specified using `PARAM`, `GRDPNT` (default is the origin of the basic coordinate system).
2. Components T1, T2, and T3 are translations and R1, R2, and R3 are rotations in the basic coordinate system.
3. One line is output for each subcase.

Figure A-123. Single Point Constraint Force Vector Resultant.

```
                                     M A X I M U M   A P P L I E D   L O A D S

 SUBCASE          T1            T2            T3            R1            R2            R3
   ID
    101      0.198030E+00  0.140028E+01  0.300000E+02  0.212133E+01  0.212133E+01  0.000000E+00
    102      0.231861E+02  0.337671E+02  0.197449E+02  0.519829E+02  0.115759E+02  0.197422E+02
    103      0.773470E+06  0.546926E+04  0.553346E+04  0.683066E-01  0.749143E+02  0.773214E+02
    104      0.106066E+06  0.636393E+05  0.299999E+05  0.000000E+00  0.000000E+00  0.000000E+00
```

Remarks:

1. The largest magnitude of applied grid point loads transformed into the basic coordinate system is output. The maximum shown for one component may not be at the same grid point as another component for the same subcase.
2. Components T1, T2, and T3 are translations and R1, R2, and R3 are rotations in the basic coordinate system.
3. One line is output for each subcase.

Figure A-124. Maximum Applied Loads.

```
                               M A X I M U M   D I S P L A C E M E N T S

 SUBCASE          T1            T2            T3            R1            R2            R3
   ID
    101      0.262390E-04  0.378310E-04  0.371446E-04  0.497369E-04  0.494026E-04  0.152797E-03
    102      0.117592E-03  0.280954E-03  0.285502E-03  0.360148E-02  0.626240E-03  0.134828E-02
    103      0.430951E-01  0.623623E-01  0.685955E-01  0.259675E-01  0.595437E-01  0.615254E-01
    104      0.823080E-01  0.789537E-01  0.120970E-01  0.106439E-01  0.566643E-01  0.662530E-01
```

Remarks:

1. The largest magnitude of grid point displacements transformed into the basic coordinate system is output. The maximum shown for one component may not be at the same grid point as another component for the same subcase.
2. Components T1, T2, and T3 are translations and R1, R2, and R3 are rotations in the basic coordinate system.
3. One line is output for each subcase.

Figure A-125. Maximum Displacements.

```
                    M A X I M U M   F O R C E S   O F   S I N G L E   P O I N T   C O N S T R A I N T

SUBCASE           T1             T2             T3             R1             R2             R3
  ID
   101      0.213363E+01  0.205637E+01  0.939635E+01  0.000000E+00  0.000000E+00  0.000000E+00
   102      0.139293E+02  0.781034E+01  0.408182E+02  0.000000E+00  0.000000E+00  0.000000E+00
   103      0.117108E+05  0.217001E+05  0.312012E+05  0.000000E+00  0.000000E+00  0.000000E+00
   104      0.276169E+05  0.524756E+03  0.441806E+04  0.000000E+00  0.000000E+00  0.000000E+00
```

Remarks:

1. The largest magnitude of single point constraint forces transformed into the basic coordinate system is output. The maximum shown for one component may not be at the same grid point as another component for the same subcase.
2. Components T1, T2, and T3 are translations and R1, R2, and R3 are rotations in the basic coordinate system.
3. One line is output for each subcase.

Figure A-126. Maximum Single Point Constraint Forces.

```
                                 N O N L I N E A R   F O R C E S   I N   G A P   E L E M E N T S

   ELEMENT      FORCES IN ELEMENT COORDINATE SYSTEM             DISPLACEMENTS IN ELEMENT COORDINATE SYSTEM               STATUS
     ID          AXIAL-X      SHEAR-Y      SHEAR-Z        AXIAL-U      SHEAR-V      SHEAR-W        SLIP-V       SLIP-W
      20       1.52049E-02  0.00000E+00  0.00000E+00    1.52049E-04 -4.02023E-04  0.00000E+00   -4.02023E-04  0.00000E+00    OPEN
      21       2.00007E+02  9.98480E+00  0.00000E+00    2.00008E-06  9.98480E-07  0.00000E+00    0.00000E+00  0.00000E+00    STICK


MAXIMUM GAP ELEMENT AXIAL FORCE =  2.000075E+02  AT ELEMENT 21
MINIMUM GAP ELEMENT AXIAL FORCE =  1.520487E-02  AT ELEMENT 20
MAXIMUM GAP ELEMENT SHEAR FORCE =  9.984796E+00  AT ELEMENT 21
MINIMUM GAP ELEMENT SHEAR FORCE =  0.000000E+00  AT ELEMENT 20
```

Remarks:

1. This output is requested using the `STRESS` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. Positive axial force is compression. See `CGAP` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for the definition of positive shear.

Figure A-127. Nonlinear Forces in Gap Elements.

```
                              F O R C E S   I N   S P R I N G   E L E M E N T S

 ELEMENT        FORCE
   ID
  1134      -0.166204E+04
  1135      -0.113493E+03
  1136      -0.104514E+02
  2134      -0.407497E+02
  2135      -0.169350E+03
  2136      -0.377195E+03
```

Remarks:

1. This output is requested using the `FORCE` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. Positive element force acts in the negative direction on the first defined degree of freedom.
3. This output is typical of all spring elements (`CELAS1` and `CELAS2`).
4. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-128. Forces in Spring Elements.

```
                                        S T R E S S E S   I N   S P R I N G   E L E M E N T S

   ELEMENT        STRESS
     ID
    1135      -0.113493E+06
    1136      -0.104514E+05
    2134      -0.814994E+05
    2135      -0.338699E+06
    2136      -0.377195E+05
    3135       0.770380E+04


MAXIMUM SPRING ELEMENT STRESS =  0.770380E+04  AT ELEMENT 3135
MINIMUM SPRING ELEMENT STRESS = -0.338699E+06  AT ELEMENT 2135
```

Remarks:

1. This output is requested using the `STRESS` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. Stress output is obtained by multiplying the force in the spring by the stress coefficient (field 9 of the `CELAS2` entry or fields 5 and 9 of the `PELAS` entry). See the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for entry format.
3. This output is typical of all spring elements (`CELAS1` and `CELAS2`).
4. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-129. Stresses in Spring Elements.

```
                                           F O R C E S   I N   R O D   E L E M E N T S

  ELEMENT         AXIAL          TORQUE
    ID            FORCE
     45      -0.555411E+03   0.305070E+02
     46      -0.158292E+03   0.000000E+00
     47       0.268081E+03   0.340150E+01
     48       0.104133E+03   0.000000E+00
     49      -0.200040E+04   0.203403E+01
     50       0.183684E+03   0.000000E+00
```

Remarks:

1. This output is requested using the `FORCE` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. Positive axial force is tension. See `CROD` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for the definition of positive torque.
3. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-130. Forces in Rod Elements.

```
                                              S T R E S S E S   I N   R O D   E L E M E N T S

   ELEMENT        AXIAL         TORSIONAL
     ID           STRESS          STRESS
      52       0.132790E+05   0.435050E+03
      53      -0.910854E+04   0.000000E+00
      54       0.172799E+05   0.503250E+01
      55       0.371227E+04   0.000000E+00
      56      -0.250064E+05   0.372707E+05
     133       0.794536E+05   0.113493E+06


MAXIMUM ROD ELEMENT AXIAL STRESS =  0.794536E+05  AT ELEMENT 133
MINIMUM ROD ELEMENT AXIAL STRESS = -0.250064E+05  AT ELEMENT 56
```

Remarks:

1. This output is requested using the `STRESS` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. Positive stress is tension. See `CROD` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for the definition of positive torque.
3. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-131. Stresses in Rod Elements.

```
                                 F O R C E S   I N   B A R   E L E M E N T S

 ELEMENT        DISTANCE      BENDING MOMENT                SHEAR FORCE              AXIAL          TORQUE
   ID                       PLANE 1       PLANE 2        PLANE 1       PLANE 2       FORCE
    62          0.0000  -0.402176E+04 -0.133193E+03  -0.494325E+04 -0.417755E+03  -0.154780E+04   0.990432E+02
                0.2000  -0.467744E+04 -0.188604E+03  -0.494325E+04 -0.417755E+03  -0.154780E+04   0.990432E+02
                0.4000  -0.533312E+04 -0.244016E+03  -0.494325E+04 -0.417755E+03  -0.154780E+04   0.990432E+02
                0.6000  -0.598880E+04 -0.299427E+03  -0.494325E+04 -0.417755E+03  -0.154780E+04   0.990432E+02
                0.8000  -0.664448E+04 -0.354839E+03  -0.494325E+04 -0.417755E+03  -0.154780E+04   0.990432E+02
                1.0000  -0.743365E+03  0.143865E+03  -0.494325E+04 -0.417755E+03  -0.154780E+04   0.990432E+02
    63          0.0000   0.394935E+04 -0.357347E+03   0.416739E+04 -0.681014E+03   0.434080E+04   0.640372E+02
                1.0000   0.389847E+03  0.224329E+03   0.416739E+04 -0.681014E+03   0.434080E+04   0.640372E+02
```

Remarks:

1. This output is requested using the `FORCE` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. Positive axial force is tension. See `CBAR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for the definition of planes 1 and 2 and for the positive directions of bending moments, shears, axial force and torque.
3. Distance is measured from end-A as a fraction of the elements length (i.e., end-A is at a distance of 0.0 and end-B at a distance of 1.0).
4. Intermediate output points are defined using the `CBARAO` Bulk Data entry. See `CBARAO` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data.*
5. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-132. Forces in Bar Elements.

```
                                      S T R E S S E S   I N   B A R   E L E M E N T S

  ELEMENT      DISTANCE      SX-C          SX-D          SX-E          SX-F          AXIAL          SX-MAX        SX-MIN
    ID

     62          0.0000   0.243792E+07  0.243792E+07  0.487584E+07  0.243792E+07  -0.171977E+05   0.485864E+07  0.242072E+07
                 0.2000   0.328467E+07  0.328467E+07  0.656935E+07  0.328467E+07  -0.171977E+05   0.655215E+07  0.326748E+07
                 0.4000   0.411143E+07  0.411143E+07  0.826285E+07  0.411143E+07  -0.171977E+05   0.824565E+07  0.411423E+07
                 0.6000   0.497818E+07  0.497818E+07  0.995635E+07  0.497818E+07  -0.171977E+05   0.993916E+07  0.496098E+07
                 0.8000   0.582493E+07  0.582493E+07  0.116499E+08  0.582493E+07  -0.171977E+05   0.116327E+08  0.580773E+07
                 0.5000   0.455480E+07  0.455480E+07  0.910960E+07  0.455480E+07  -0.171977E+05   0.909240E+07  0.453760E+07
                 0.6000   0.497818E+07  0.497818E+07  0.995635E+07  0.497818E+07  -0.171977E+05   0.993916E+07  0.496098E+07
                 1.0000  -0.179584E+07 -0.179584E+07 -0.359168E+07 -0.179584E+07  -0.171977E+05  -0.181104E+07 -0.360888E+07

     64          0.0000  -0.566230E+07 -0.566230E+07 -0.113246E+08 -0.566230E+07  -0.216618E+05  -0.568396E+07 -0.113463E+08
                 1.0000   0.482402E+07  0.482402E+07  0.964805E+07  0.482402E+07  -0.216618E+05   0.962639E+07  0.480236E+07


 MAXIMUM BAR ELEMENT TOTAL STRESS =  0.116327E+08  AT ELEMENT 62
 MINIMUM BAR ELEMENT TOTAL STRESS = -0.113463E+08  AT ELEMENT 64
```

Remarks:

1. This output is requested using the `STRESS` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. Positive stress is tension.
3. Distance is measured from end-A as a fraction of the elements length (i.e., end-A is at a distance of 0.0 and end-B at a distance of 1.0).
4. Intermediate output points are defined using the `CBARAO` Bulk Data entry. See `CBARAO` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data.*
5. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-133. Stresses in Bar Elements.

```
              T H E R M A L   G R A D I E N T S   A N D   H E A T   F L U X E S   I N   B A R   E L E M E N T S

    ELEMENT        GRADIENT            FLUX
      ID
        1        5.257847E+01     -1.050446E-02
        2        5.219163E+01     -1.074296E-02
        3        5.706667E+01     -1.210745E-02
        4        6.554758E+01     -1.437293E-02
        5        7.770808E+01     -1.768509E-02
        6        9.405507E+01     -2.224681E-02
        7        1.148997E+02     -2.833560E-02
        8        1.393850E+02     -3.630588E-02
        9        1.675567E+02     -4.655468E-02
       10        1.790111E+02     -5.302624E-02


 MAXIMUM BAR ELEMENT THERMAL GRADIENT MAGNITUDE =  1.790111E+02  AT ELEMENT 10
 MINIMUM BAR ELEMENT THERMAL GRADIENT MAGNITUDE =  5.219163E+01  AT ELEMENT 2
 MAXIMUM BAR ELEMENT HEAT FLUX MAGNITUDE        = -1.050446E-02  AT ELEMENT 1
 MINIMUM BAR ELEMENT HEAT FLUX MAGNITUDE        = -5.302624E-02  AT ELEMENT 10
```

Remarks:

1. This output is requested using the `FLUX` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. This output is typical of line elements (`CROD`, `CBAR`, and `CBEAM`).

Figure A-134. Thermal Gradients and Heat Fluxes in Bar Elements.

```
                                      F O R C E S   I N   S H E A R   E L E M E N T S

ELEMENT       SHEAR-12      SHEAR-23      SHEAR-34      SHEAR-41          KICK-1        KICK-2        KICK-3        KICK-4
  ID
   57       0.568432E-11 -0.617109E-03 -0.625278E-12  0.617109E-03      0.264411E+03  0.264411E+03 -0.264411E+03 -0.264411E+03
   58      -0.290539E+03  0.678092E-03 -0.290540E+03  0.678092E-03     -0.568434E-11 -0.568434E-11  0.000000E+00  0.000000E+00
   59       0.852647E-11  0.105424E-02 -0.625278E-12 -0.105424E-02     -0.451708E+03 -0.451708E+03  0.451708E+03  0.451708E+03
   60      -0.472039E+03  0.110170E-02 -0.472041E+03  0.110170E-02      0.454747E-12 -0.227374E-12  0.511591E-12 -0.170530E-12
 1157       0.568432E-11 -0.617109E-03 -0.625278E-12  0.617109E-03      0.264411E+03  0.264411E+03 -0.264411E+03 -0.264411E+03
 1158      -0.290539E+03  0.678092E-03 -0.290540E+03  0.678092E-03     -0.568434E-11 -0.568434E-11  0.000000E+00  0.000000E+00
```

Remarks:

1. This output is requested using the `FORCE` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. See `CSHEAR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for the definition of element coordinate system and force components.

Figure A-135. Forces in Shear Elements.

```
                                    S T R E S S E S   I N   S H E A R   E L E M E N T S

   ELEMENT       AVERAGE       MINIMUM       MAXIMUM
     ID           SHEAR         SHEAR         SHEAR
      57      -0.284217E-11 -0.123422E-01  0.123422E-01
      58      -0.290539E+04 -0.581080E+04  0.115618E-01
      59      -0.270006E-11 -0.210848E-01  0.210848E-01
      60      -0.472039E+04 -0.944082E+04  0.220339E-01
    1157      -0.284217E-11 -0.123422E-01  0.123422E-01
    1460      -0.472039E+04 -0.944082E+04  0.220339E-01


MAXIMUM SHEAR ELEMENT SHEAR STRESS =  0.220339E-01  AT ELEMENT 1460
MINIMUM SHEAR ELEMENT SHEAR STRESS = -0.944082E+04  AT ELEMENT 1460
```

Remarks:

1. This output is requested using the `STRESS` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. Shear stresses are calculated along the element edges. Average shear is the average of all four edge shear stresses. Minimum shear is the minimum edge shear stress. Maximum shear is the maximum edge shear stress.

Figure A-136. Stresses in Shear Elements.

```
                                    H E A T   F L O W   I N T O   H B D Y   E L E M E N T S

  ELEMENT       APPLIED       CONVECTION     RADIATION        TOTAL
    ID
     12      1.000000E+01   0.000000E+00  -1.000159E+01  -1.585793E-03
     13      1.000000E+01   0.000000E+00  -1.000159E+01  -1.585793E-03
     22      9.123600E-03  -2.518653E-02   0.000000E+00  -1.606293E-02
     23      8.020000E-04  -2.511956E-02   0.000000E+00  -2.433756E-02
     26      0.000000E+00  -0.373852E-01   0.000000E+00  -0.373852E-01
     27      0.000000E+00  -0.348658E-01   0.000000E+00  -0.348658E-01
     32      9.126400E-03   0.000000E+00   0.000000E+00   9.126400E-03
     33      1.125000E-03   0.000000E+00   0.000000E+00   1.125000E-03


MAXIMUM HBDY ELEMENT HEAT FLOW =  9.126400E-03  AT ELEMENT 32
MINIMUM HBDY ELEMENT HEAT FLOW = -0.373852E-01  AT ELEMENT 25
```

Remarks:

1. This output is requested using the `FLUX` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. Applied heat flow is the result of heat flux loads (`QBDY1` and `QBDY2`) applied to hbdy elements (`CHBDYP` and `CHBDYG`). Convection heat flow is the result of free convection loads (`CONV`) applied to hbdy elements. Radiation heat flow is the result of surface radiation loads (`RADBC`) applied to hbdy elements.

Figure A-137. Heat Flow Into Hbdy Elements.

```
                           F O R C E S   I N   Q U A D   E L E M E N T S   O N   S U R F A C E   1

  SURFACE COORDINATE ID = GRID  X-AXIS = X  NORMAL = Z

    ELEMENT     GRID              MEMBRANE  FORCES                             BENDING  MOMENTS                 TRANSVERSE SHEAR FORCES
      ID         ID          FX           FY           FXY              MX           MY           MXY              QX           QY
        1      CENTER     0.43733E+04  0.33983E+02  0.19833E+04   -0.60301E+02 -0.53516E+02 -0.51218E+02    0.95923E+02  0.99433E+02
        2      CENTER     0.28545E+04  0.65454E+03  0.12097E+04   -0.36872E+02  0.28229E+02  0.24996E+02    0.15178E+02 -0.27760E+03
        3      CENTER     0.56781E+04 -0.98814E+02  0.16930E+04   -0.14418E+03 -0.10332E+03 -0.40701E+02    0.39510E+03  0.33240E+03
        4      CENTER     0.57292E+04  0.89260E+03 -0.79976E+03   -0.77539E+02  0.43797E+02 -0.12523E+02    0.52993E+03 -0.68043E+03
        5      CENTER    -0.15600E+04  0.23426E+03 -0.56919E+03   -0.71223E+02 -0.33979E+02  0.59102E+02   -0.73129E+03  0.33729E+03
        6      CENTER    -0.27352E+04 -0.18546E+04  0.27665E+04   -0.77953E+02  0.79705E+01  0.32214E+02   -0.17223E+04 -0.24726E+02
```

Remarks:

1. This output is requested using the `FORCE` Case Control command. Corner force output is requested by using the `FORCE(CORNER)` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. Positive force is tension. Forces may be output in any `SURFACE` coordinate system (`ELEMENT`, `GRID`, `BASIC`, `MATERIAL`, or user defined). In the above example, forces are output in the `GRID` or displacement coordinate system (field 7 on the `GRID` Bulk Data entry). See `SURFACE` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information. See `CQUAD4` and `CQUADR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for the definition of element coordinate system and force components.
3. This output is typical of all shell elements (`CQUAD4`, `CQUADR`, `CTRIA3`, and `CTRIAR`).
4. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-138. Forces in Quad Elements (Without Corner Option).

```
                         F O R C E S   I N   Q U A D   E L E M E N T S   O N   S U R F A C E   1

 SURFACE COORDINATE ID = GRID  X-AXIS = X  NORMAL = Z

   ELEMENT     GRID              MEMBRANE  FORCES                           BENDING  MOMENTS                TRANSVERSE SHEAR FORCES
     ID         ID          FX            FY            FXY             MX            MY            MXY             QX            QY
      1       CENTER    0.43733E+04  0.33983E+02  0.19833E+04   -0.60301E+02 -0.53516E+02 -0.51218E+02    0.95923E+02  0.99433E+02
                   1    0.87432E+04  0.43733E+03  0.19833E+04   -0.14227E+03  0.24104E+02 -0.49262E+02    0.43850E+03 -0.21187E+03
                   2    0.33983E+01  0.43733E+03  0.19833E+04   -0.39003E+02 -0.42231E+02 -0.36555E+02   -0.24665E+03 -0.21187E+03
                   5    0.33983E+01 -0.36936E+03  0.19833E+04    0.21666E+02 -0.11114E+03 -0.53174E+02   -0.24665E+03  0.41274E+03
                   4    0.87432E+04 -0.36936E+03  0.19833E+04   -0.81599E+02 -0.64802E+02 -0.65880E+02    0.43850E+03  0.41274E+03
      2       CENTER    0.28545E+04  0.65454E+03  0.12097E+04   -0.36872E+02  0.28229E+02  0.24996E+02    0.15178E+02 -0.27760E+03
                   2    0.65454E+02 -0.52374E+03  0.12097E+04   -0.31639E+01 -0.38089E+02  0.39670E+02    0.25723E+02  0.16734E+03
                   3    0.56436E+04 -0.52374E+03  0.12097E+04   -0.70881E+02  0.78649E+02  0.32381E+02    0.46327E+01  0.16734E+03
                   6    0.56436E+04  0.18328E+04  0.12097E+04   -0.70581E+02  0.94547E+02  0.10321E+02    0.46327E+01 -0.72253E+03
                   5    0.65454E+02  0.18328E+04  0.12097E+04   -0.28642E+01 -0.22191E+02  0.17610E+02    0.25723E+02 -0.72253E+03
      3       CENTER    0.56781E+04 -0.98814E+02  0.16930E+04   -0.14418E+03 -0.10332E+03 -0.40701E+02    0.39510E+03  0.33240E+03
                   4    0.47731E+04 -0.23889E+03  0.16930E+04   -0.16336E+03 -0.72489E+02 -0.45942E+02   -0.50748E+03  0.63098E+03
                   5    0.65831E+04 -0.23889E+03  0.16930E+04   -0.25348E+03 -0.15914E+03 -0.11501E+03    0.12977E+04  0.63098E+03
                   8    0.65831E+04  0.41257E+02  0.16930E+04   -0.12500E+03 -0.11414E+03 -0.35461E+02    0.12977E+04  0.33815E+02
                   7    0.47731E+04  0.41257E+02  0.16930E+04   -0.34876E+02 -0.47491E+02  0.33607E+02   -0.50748E+03  0.33815E+02
      4       CENTER    0.57292E+04  0.89260E+03 -0.79976E+03   -0.77539E+02  0.43797E+02 -0.12523E+02    0.52993E+03 -0.68043E+03
                   5    0.66822E+04  0.21203E+04 -0.79976E+03   -0.26388E+03 -0.62307E+02  0.11908E+03    0.12543E+04 -0.99386E+03
                   6    0.47762E+04  0.21203E+04 -0.79976E+03   -0.61186E+02  0.10948E+03  0.24303E+02   -0.19448E+03 -0.99386E+03
                   9    0.47762E+04 -0.33509E+03 -0.79976E+03    0.10880E+03  0.14990E+03 -0.14412E+03   -0.19448E+03 -0.36700E+03
                   8    0.66822E+04 -0.33509E+03 -0.79976E+03   -0.93692E+02 -0.21887E+02 -0.49350E+02    0.12543E+04 -0.36700E+03
```

Remarks:

1. This output is requested using the `FORCE(CORNER)` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. Positive force is tension. Forces may be output in any `SURFACE` coordinate system (`ELEMENT`, `GRID`, `BASIC`, `MATERIAL`, or user defined). In the above example, forces are output in the `GRID` or displacement coordinate system (field 7 on the `GRID` Bulk Data entry). See `SURFACE` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information. See `CQUAD4` and `CQUADR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for the definition of element coordinate system and force components.
3. This output is typical of all shell elements (`CQUAD4`, `CQUADR`, `CTRIA3`, and `CTRIAR`).
4. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-139. Forces in Quad Elements (With Corner Option).

```
                        S T R E S S E S   I N   Q U A D   E L E M E N T S   O N   S U R F A C E   55

 SURFACE COORDINATE ID = 0  X-AXIS = X  NORMAL = Z

   ELEMENT     GRID        FIBER       STRESSES IN SURFACE COORDINATE SYSTEM       PRINCIPAL STRESSES (ZERO SHEAR)        HENCKY
     ID         ID        DISTANCE      NORMAL-X     NORMAL-Y     SHEAR-XY         ANGLE        MAJOR        MINOR       VON MISES

       3      CENTER   -0.50000E-01    0.20766E+05 -0.12904E+05  0.64626E+04     10.5004    0.21964E+05 -0.14101E+05    0.31480E+05
                        0.15000E+00    0.89118E+05  0.36076E+05  0.25758E+05     22.0818    0.99568E+05  0.25626E+05    0.89548E+05

                   4   -0.50000E-01    0.12459E+05 -0.10184E+05  0.58415E+04     11.6458    0.11878E+05 -0.11602E+05    0.22095E+05
                        0.15000E+00    0.89904E+05  0.24181E+05  0.27621E+05     20.0241    0.99971E+05  0.14115E+05    0.93714E+05

                   5   -0.50000E-01    0.11845E+05 -0.20454E+05 -0.23443E+04     -3.8921    0.14004E+05 -0.20611E+05    0.30161E+05
                        0.15000E+00    0.11402E+06  0.54991E+05  0.52179E+05     26.4326    0.15995E+06  0.29052E+05    0.14759E+06

                   8   -0.50000E-01    0.29073E+05 -0.15623E+05  0.70837E+04      8.7936    0.30168E+05 -0.16719E+05    0.41159E+05
                        0.15000E+00    0.88332E+05  0.47970E+05  0.23895E+05     24.9082    0.99427E+05  0.36874E+05    0.87059E+05

                   7   -0.50000E-01    0.27687E+05 -0.53535E+04  0.15269E+05     21.3733    0.33663E+05 -0.11329E+05    0.40533E+05
                        0.15000E+00    0.44221E+05  0.17161E+05 -0.66266E+03     -1.4020    0.44237E+05  0.17144E+05    0.38632E+05


MAXIMUM QUAD ELEMENT PRINCIPAL STRESS =  0.995677E+05  AT ELEMENT 3
MINIMUM QUAD ELEMENT PRINCIPAL STRESS = -0.815991E+05  AT ELEMENT 27
MAXIMUM QUAD ELEMENT SHEAR STRESS     =  0.423755E+05  AT ELEMENT 26
MAXIMUM QUAD ELEMENT VON MISES STRESS =  0.895483E+05  AT ELEMENT 3
```

Remarks:

1. This output is requested using the `STRESS(CORNER)` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. This output is typical of all shell elements (`CQUAD4`, `CQUADR`, `CTRIA3`, and `CTRIAR`).
3. Stresses may be output in any `SURFACE` coordinate system (`ELEMENT`, `GRID`, `BASIC`, `MATERIAL`, or user defined). In the above example, stresses are output in the basic coordinate system (`SURFACE COORDINATE ID = 0`). See `SURFACE` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information. See `CQUAD4` and `CQUADR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for the definition of element coordinate system and stress components.
4. The angle of principal stress is in the surface coordinate system.
5. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-140. Stresses in Quad Elements (With Corner Option).

```
                        S T R E S S E S   I N   Q U A D   E L E M E N T S   O N   S U R F A C E   55

 SURFACE COORDINATE ID = 1  X-AXIS = X  NORMAL = Z

   ELEMENT     GRID        FIBER       STRESSES IN SURFACE COORDINATE SYSTEM       PRINCIPAL STRESSES (ZERO SHEAR)        HENCKY
     ID         ID        DISTANCE      NORMAL-X     NORMAL-Y     SHEAR-XY         ANGLE        MAJOR        MINOR       VON MISES

       3     CENTER    -0.50000E-01    0.20766E+05 -0.12904E+05  0.64626E+04     10.5004    0.21964E+05 -0.14101E+05    0.31480E+05
                        0.15000E+00    0.89118E+05  0.36076E+05  0.25758E+05     22.0818    0.99568E+05  0.25626E+05    0.89548E+05

       6     CENTER    -0.50000E-01    0.94820E+04 -0.15198E+05  0.69895E+04     14.7640    0.11324E+05 -0.17040E+05    0.24729E+05
                        0.15000E+00   -0.27474E+05 -0.11419E+05  0.22261E+05     54.9145    0.42179E+04 -0.43111E+05    0.45367E+05

      26     CENTER    -0.50000E-01    0.36784E+04  0.21454E+05 -0.10957E+05    -64.5234    0.26675E+05 -0.15424E+04    0.27479E+05
                        0.15000E+00    0.15550E+05  0.19719E+05 -0.42324E+05    -46.4100    0.60010E+05 -0.24741E+05    0.75485E+05

      27     CENTER    -0.50000E-01    0.92164E+04 -0.30582E+05 -0.10629E+05    -14.0549    0.11877E+05 -0.33243E+05    0.40509E+05
                        0.15000E+00   -0.32980E+05 -0.57552E+05 -0.34193E+05    -35.1177   -0.89327E+04 -0.81599E+05    0.77520E+05


MAXIMUM QUAD ELEMENT PRINCIPAL STRESS =  0.995677E+05  AT ELEMENT 3
MINIMUM QUAD ELEMENT PRINCIPAL STRESS = -0.815991E+05  AT ELEMENT 27
MAXIMUM QUAD ELEMENT SHEAR STRESS     =  0.423755E+05  AT ELEMENT 26
MAXIMUM QUAD ELEMENT VON MISES STRESS =  0.895483E+05  AT ELEMENT 3
```

Remarks:

1. This output is requested using the `STRESS(CENTER)` or `STRESS` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. The default for the last output column can be changed to maximum shear stress by specifying the `STRESS(SHEAR)` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
3. This output is typical of all shell elements (`CQUAD4`, `CQUADR`, `CTRIA3`, and `CTRIAR`).
4. Element stresses may be output in any `SURFACE` coordinate system (`ELEMENT`, `GRID`, `BASIC`, `MATERIAL`, or user defined). In the above example, stresses are output in `ELEMENT` coordinate system. See `SURFACE` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information. See `CQUAD4` and `CQUADR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for the definition of element coordinate system and stress components.
5. The angle of principal stress is in the surface coordinate system.
6. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-141. Stresses in Quad Elements (Without Corner Option).

```
                    S T R A I N S   I N   Q U A D   E L E M E N T S   O N   S U R F A C E  1

SURFACE COORDINATE ID = MATERIAL  X-AXIS = Z  NORMAL = R

 ELEMENT     GRID       FIBER      STRAINS IN SURFACE COORDINATE SYSTEM       PRINCIPAL STRAINS (ZERO SHEAR)        HENCKY
   ID         ID       DISTANCE      NORMAL-X      NORMAL-Y     SHEAR-XY         ANGLE        MAJOR        MINOR        VON MISES

      3      CENTER   -0.50000E-01    0.15755E-02 -0.10700E-02  0.92322E-03      9.6191    0.16537E-02 -0.11482E-02    0.16264E-02
                       0.15000E+00    0.61079E-02  0.19403E-02  0.36797E-02     20.7211    0.68039E-02  0.12443E-02    0.41833E-02

      7      CENTER   -0.50000E-01   -0.12195E-02  0.14501E-02 -0.20897E-02    -70.9733    0.18104E-02 -0.15798E-02    0.19589E-02
                       0.15000E+00   -0.79077E-03  0.31047E-03  0.16344E-02     61.9860    0.74524E-03 -0.12255E-02    0.11490E-02

     26      CENTER   -0.50000E-01    0.19100E-03  0.14247E-02 -0.16278E-02    -63.5792    0.18291E-02 -0.21138E-03    0.12964E-02
                       0.15000E+00    0.11119E-02  0.11554E-02 -0.64869E-02    -45.1919    0.43772E-02 -0.21099E-02    0.38208E-02

     27      CENTER   -0.50000E-01    0.80303E-03 -0.21765E-02 -0.17555E-02    -15.2528    0.10424E-02 -0.24159E-02    0.20484E-02
                       0.15000E+00   -0.19047E-02 -0.39152E-02 -0.53270E-02    -34.6610   -0.63034E-04 -0.57569E-02    0.38171E-02


MAXIMUM QUAD ELEMENT PRINCIPAL STRAIN =  0.680389E-02  AT ELEMENT 3
MINIMUM QUAD ELEMENT PRINCIPAL STRAIN = -0.575687E-02  AT ELEMENT 27
MAXIMUM QUAD ELEMENT SHEAR STRAIN     =  0.648704E-02  AT ELEMENT 26
MAXIMUM QUAD ELEMENT VON MISES STRAIN =  0.418331E-02  AT ELEMENT 3
```

Remarks:

1. This output is requested using the `STRAIN(CENTER)` or `STRAIN` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. The default for the last output column can be changed to maximum shear strain by specifying the `STRAIN(SHEAR)` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
3. This output is typical of all shell elements (`CQUAD4`, `CQUADR`, `CTRIA3`, and `CTRIAR`).
4. Element strains may be output in any `SURFACE` coordinate system (`ELEMENT`, `GRID`, `BASIC`, `MATERIAL`, or user defined). In the above example, strains are output in `MATERIAL` coordinate system. See `SURFACE` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information. See `CQUAD4` and `CQUADR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for the definition of element coordinate system and strain components.
5. The angle of principal strain is in the surface coordinate system.
6. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-142. Strains in Quad Elements.

```
                  S T R A I N   E N E R G Y   I N   Q U A D   E L E M E N T S   O N   S U R F A C E   1

   ELEMENT     STRAIN ENERGY       PERCENT       STRAIN ENERGY
     ID                             TOTAL           DENSITY
       1       0.233628E+01         0.0154       0.623008E+02
       2       0.106848E+01         0.0071       0.284927E+02
       3       0.340972E+01         0.0225       0.909259E+02
       4       0.319828E+01         0.0211       0.852875E+02
       5       0.182172E+01         0.0120       0.485792E+02
       6       0.442268E+01         0.0292       0.117938E+03
       7       0.218740E+01         0.0145       0.583307E+02
       8       0.489503E+01         0.0324       0.110534E+03
      25       0.426861E+01         0.0282       0.113830E+03
      26       0.258918E+01         0.0171       0.690449E+02
      27       0.348669E+01         0.0231       0.929783E+02


   SUBTOTAL    0.398987E+02         0.2638


MAXIMUM QUAD ELEMENT STRAIN ENERGY DENSITY =  0.110534E+03  AT ELEMENT 8
MINIMUM QUAD ELEMENT STRAIN ENERGY DENSITY =  0.821887E+01  AT ELEMENT 30
```

Remarks:

1. This output is requested using the `ESE` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. This output is typical of all shell elements (`CQUAD4`, `CQUADR`, `CTRIA3`, and `CTRIAR`).
3. Percentages are based on the entire model not the individual surface.
4. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-143. Strain Energy in Quad Elements.

```
 T H E R M A L   G R A D I E N T S   A N D   H E A T   F L U X E S   I N   Q U A D   E L E M E N T S   O N   S U R F A C E   1

 SURFACE COORDINATE ID = ELEMENT  X-AXIS = X  NORMAL = Z

   ELEMENT     GRID      X-GRADIENT   Y-GRADIENT         X-FLUX       Y-FLUX
     ID         ID
      11      CENTER    -7.37064E+02 -3.18014E+03      1.51835E+00  6.55109E+00
      12      CENTER     6.10143E+02 -3.20875E+03     -1.25690E+00  6.61003E+00
      11      CENTER     1.07172E+02 -1.80983E+03     -5.95445E-02  1.00554E+00
      14      CENTER    -2.04972E+02 -1.76666E+03      1.11883E-01  9.81555E-01
      15      CENTER    -1.02514E+02  1.71191E+03      1.19634E-01 -1.99780E+00
      16      CENTER     9.86067E+01  1.71568E+03     -1.15074E-01 -2.00220E+00
      17      CENTER     5.89261E-01  2.28650E+02     -1.21188E-03 -4.71020E-01
      18      CENTER    -8.63659E+00  2.22806E+02      1.77914E-02 -4.58980E-01
      19      CENTER    -1.26669E+02  5.11178E+02      1.47822E-01 -5.96545E-01
      20      CENTER     1.18774E+02  5.17099E+02     -1.38610E-01 -6.03455E-01


MAXIMUM QUAD ELEMENT THERMAL GRADIENT MAGNITUDE =  3.266248E+03  AT ELEMENT 12
MINIMUM QUAD ELEMENT THERMAL GRADIENT MAGNITUDE =  2.229733E+02  AT ELEMENT 18
MAXIMUM QUAD ELEMENT HEAT FLUX MAGNITUDE        =  6.728470E+00  AT ELEMENT 12
MINIMUM QUAD ELEMENT HEAT FLUX MAGNITUDE        =  4.593251E-01  AT ELEMENT 18
```

Remarks:

1. This output is requested using the `FLUX` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. This output is typical of all shell elements (`CQUAD4`, `CQUADR`, `CTRIA3`, and `CTRIAR`).
3. Element stresses may be output in any `SURFACE` coordinate system (`ELEMENT`, `GRID`, `BASIC`, `MATERIAL`, or user defined). In the above example, thermal gradients and heat fluxes are output in `ELEMENT` coordinate system. See `SURFACE` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information. See `CQUAD4` and `CQUADR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for the definition of element coordinate system.

Figure A-144. Thermal Gradients and Heat Fluxes in Quad Elements.

```
                           G R I D   P O I N T   S T R E S S E S   O N   S U R F A C E   2

  SURFACE COORDINATE ID = 0  X-AXIS = X  NORMAL = Z

     GRID          FIBER       STRESSES IN SURFACE COORDINATE SYSTEM       PRINCIPAL STRESSES (ZERO SHEAR)        HENCKY
      ID          DISTANCE      NORMAL-X     NORMAL-Y     SHEAR-XY          ANGLE         MAJOR         MINOR       VON MISES

        1      -0.50000E-01    0.24805E+05  0.31087E+05 -0.27967E+03    -87.4559    0.31099E+05  0.24793E+05    0.28475E+05
                0.15000E+00    0.66722E+05  0.36196E+05 -0.10315E+05    -17.0261    0.69881E+05  0.33037E+05    0.60549E+05

       12      -0.50000E-01   -0.39380E+05 -0.56639E+05 -0.30871E+05    -37.1910   -0.15955E+05 -0.80064E+05    0.73399E+05
                0.15000E+00    0.60103E+04 -0.51155E+05  0.16150E+05     14.7335    0.10257E+05 -0.55402E+05    0.61179E+05

       38      -0.50000E-01    0.49723E+04 -0.17208E+05 -0.63254E+04    -14.8491    0.66493E+04 -0.18885E+05    0.22945E+05
                0.15000E+00    0.20812E+05 -0.30507E+05 -0.41495E+05    -29.1143    0.43940E+05 -0.53635E+05    0.84642E+05

       41      -0.14000E+00    0.33959E+04 -0.53072E+04 -0.74702E+04    -29.8892    0.76895E+04 -0.96009E+04    0.15004E+05
                0.14000E+00   -0.17352E+05 -0.32984E+05 -0.31119E+05    -37.9506    0.69176E+04 -0.57254E+05    0.61008E+05


 MAXIMUM SHELL ELEMENT PRINCIPAL STRESS =  0.698811E+05  AT GRID 1
 MINIMUM SHELL ELEMENT PRINCIPAL STRESS = -0.800640E+05  AT GRID 12
 MAXIMUM SHELL ELEMENT SHEAR STRESS     =  0.487878E+05  AT GRID 38
 MAXIMUM SHELL ELEMENT VON MISES STRESS =  0.846418E+05  AT GRID 38
```

Remarks:

1. This output is requested using the `GPSTRESS` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. Grid point stresses may be output in any `SURFACE` coordinate system (`GRID`, `BASIC`, `MATERIAL`, or user defined) except `ELEMENT`. In the above example, stresses are output in the basic coordinate system (`SURFACE COORDINATE ID = 0`). See `SURFACE` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information. See `CQUAD4` and `CQUADR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for the definition of element coordinate system and stress components.
3. The direction cosines of the principal stresses are with respect to the `SURFACE` coordinate system.
4. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-145. Surface Grid Point Stresses.

```
     G R I D   P O I N T   T H E R M A L   G R A D I E N T S   A N D   H E A T   F L U X E S   O N   S U R F A C E   1

 SURFACE COORDINATE ID = GRID  X-AXIS = X  NORMAL = Z

    GRID       X-GRADIENT   Y-GRADIENT         X-FLUX       Y-FLUX
     ID
      18      -3.79837E+03 -1.78938E+03      7.82465E+00  3.68612E+00
      19      -3.74115E+03  1.85907E+03      7.70676E+00 -3.82969E+00
      20      -2.61914E+03  3.48481E+01      5.39542E+00 -7.17872E-02
      22      -2.70604E+03 -3.84946E+02      4.31757E+00  5.03433E-01
      23      -2.28393E+03  9.20729E+01      3.23906E+00 -1.20411E-01
      24      -2.69148E+03  5.69092E+02      4.35252E+00 -7.44259E-01
      25       3.46077E+00  1.70603E+02     -5.14846E-01 -1.46940E-01
      26      -1.01182E+02  5.72799E+00     -4.77410E-01 -4.93352E-03
      27       5.04075E+01 -1.59147E+02     -5.43237E-01  1.37073E-01
      28       9.44504E+02  3.44250E+01     -1.19651E+00 -5.55447E-02
      31       4.01729E+02 -3.56035E+01     -5.63093E-01  5.74463E-02
      32       3.38100E+02  9.86796E+00     -5.04472E-01 -1.59219E-02


MAXIMUM SHELL ELEMENT THERMAL GRADIENT MAGNITUDE =  4.198750E+03  AT GRID 18
MINIMUM SHELL ELEMENT THERMAL GRADIENT MAGNITUDE =  1.015437E+02  AT GRID 26
MAXIMUM SHELL ELEMENT HEAT FLUX MAGNITUDE        =  8.649425E+00  AT GRID 18
MINIMUM SHELL ELEMENT HEAT FLUX MAGNITUDE        =  4.774356E-01  AT GRID 26
```

Remarks:

1. This output is requested using the `GPFLUX` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. Thermal gradients and heat fluxes may be output in any `SURFACE` coordinate system (`GRID`, `BASIC`, `MATERIAL`, or user defined) except `ELEMENT`. In the above example, thermal gradients and heat fluxes are output in the `GRID` or displacement coordinate system (field 7 on the `GRID` Bulk Data entry). See `SURFACE` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information. See `CQUAD4` and `CQUADR` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for the definition of element coordinate system.

Figure A-146. Surface Grid Point Thermal Gradients and Heat Fluxes.

```
               S T R E S S E S   I N   C O M P O S I T E   Q U A D   E L E M E N T S   O N   S U R F A C E   0

 SURFACE COORDINATE ID = MATERIAL  X-AXIS = X  NORMAL = Z

   ELEMENT   PLY  STRESSES IN FIBER AND MATRIX DIRECTIONS  INTER-LAMINAR STRESSES     PRINCIPAL STRESSES (ZERO SHEAR)     HENCKY
     ID       ID      NORMAL-1     NORMAL-2     SHEAR-12   SHEAR XZ-MAT SHEAR YZ-MAT   ANGLE     MAJOR        MINOR       VON MISES

       1       1   1.12722E+04  9.56493E+02  2.13307E+01   2.46547E+02  1.61686E+02    0.12  1.12722E+04  9.56449E+02   1.08257E+04
               2   1.99977E+03  9.64334E+01 -1.17085E+03   2.62642E+02  1.78571E+02  -25.45  2.55693E+03 -4.60728E+02   2.81571E+03
               3  -3.42086E+02 -1.28842E+03 -8.15739E+02   2.30154E+02  1.34206E+02  -29.94  1.27783E+02 -1.75829E+03   1.82554E+03
               4  -2.31106E+03 -1.27838E+03  4.05529E+03   1.91004E+02  9.59349E+01   48.63  2.29331E+03 -5.88275E+03   7.30459E+03
               5  -1.92859E+04 -2.17245E+03  4.52332E+02   2.68667E+01  2.39332E+01   88.49 -2.16050E+03 -1.92979E+04   1.83134E+04

       2       1   4.18298E+03  3.23883E+02 -2.22389E+02   1.31625E+02 -3.97567E+01   -3.29  4.19575E+03  3.11110E+02   4.04917E+03
               2   1.70473E+02  7.01896E+01 -4.58916E+02   1.40218E+02 -4.39083E+01  -41.88  5.81978E+02 -3.41316E+02   8.08600E+02
               3  -2.83655E+02 -4.94233E+02 -4.03160E+02   1.22873E+02 -3.29995E+01  -37.68  2.77382E+01 -8.05626E+02   8.19847E+02
               4  -4.04614E+02 -5.42444E+02  1.16599E+03   1.01973E+02 -2.35892E+01   43.31  6.94497E+02 -1.64155E+03   2.07776E+03
               5  -7.29650E+03 -1.06455E+03  3.91265E+02   1.43435E+01 -5.88487E+00   86.42 -1.04008E+03 -7.32096E+03   6.86031E+03


MAXIMUM QUAD ELEMENT PRINCIPAL STRESS          =  1.127220E+04  AT ELEMENT 1
MINIMUM QUAD ELEMENT PRINCIPAL STRESS          = -1.929786E+04  AT ELEMENT 1
MAXIMUM QUAD ELEMENT INTERLAMINAR SHEAR STRESS =  2.626419E+02  AT ELEMENT 1
MINIMUM QUAD ELEMENT INTERLAMINAR SHEAR STRESS = -4.390831E+01  AT ELEMENT 2
```

Remarks:

1. This output is requested using the `STRESS` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. This output is typical of all composite shell elements (`CQUAD4`, `CQUADR`, `CQUAD8`, `CTRIA3`, `CTRIAR`, and `CTRIA6`).
3. Direct inplane stresses are always output in the ply coordinate system (fiber and matrix direction). Interlaminar stresses are always output in the material coordinate system. The angle of principal stress is in the ply coordinate system.
4. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-147. Stresses in Composite Quad Elements.

```
       F A I L U R E   I N D E X E S   F O R   C O M P O S I T E   Q U A D   E L E M E N T S   O N   S U R F A C E   1

   ELEMENT   FAILURE   PLY   MATRIX FAILURE  FAILURE    FIBER FAILURE   FAILURE     BOND FAILURE   FAILURE   ELEMENT FAILURE
     ID      THEORY    ID        INDEX        MODE          INDEX        MODE          INDEX        MODE         INDEX
                                                                                    1.38941E-01
       1      MCT        1    2.61597E-04     FILL       3.42999E-04     FILL
                                                                                    1.48204E-01
              MCT        2    2.35525E-04     FILL       3.04525E-04     FILL
                                                                                    1.38941E-01
              MCT        3    6.36116E-04     FILL       3.40863E-04     FILL
                                                                                    6.48393E-02
              MCT        4    1.74382E-03     WARP       1.28142E-03     FILL                                  1.48204E-01

       2      LARC02     1    5.13293E+03    TENSION     2.29915E+00    TENSION
                                                                                    9.20752E-01
              LARC02     2    1.25951E+02    TENSION     2.36968E+00    TENSION
                                                                                    1.35193E+00
              LARC02     3    3.02465E+01  COMPRESSION   1.85447E+00  COMPRESSION
                                                                                    9.20752E-01
              LARC02     4    3.45537E+02  COMPRESSION   0.00000E+00                                           5.13293E+03   FAILED

MAXIMUM QUAD ELEMENT PLY FAILURE INDEX   =  5.132932E+03  AT ELEMENT 2
MAXIMUM QUAD ELEMENT BOND FAILURE INDEX  =  1.351931E+00  AT ELEMENT 2
```

Remarks:

1. This output is available when composite stress/strain is output and allowable stresses/strains are supplied on the material property (`MAT1`, `MAT2`, or `MAT8`) and `PCOMP` entries that were used to define the laminate. See the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for more information.
2. This output is typical of all composite shell elements (`CQUAD4`, `CQUADR`, `CQUAD8`, `CTRIA3`, `CTRIAR`, and `CTRIA6`).
3. Indexes greater than 1.0 indicate a failure in that ply using the failure criteria specified for the laminate (field 6 on the `PCOMP` Bulk Data entry).

Figure A-148. Failure Indexes in Composite Quad Elements.

```
       S T R E N G T H   R A T I O S   F O R   C O M P O S I T E   Q U A D   E L E M E N T S   O N   S U R F A C E   1

   ELEMENT   FAILURE   PLY  MATRIX STRENGTH  FAILURE    FIBER STRENGTH  FAILURE    BOND STRENGTH   FAILURE  ELEMENT STRENGTH
     ID      THEORY    ID        RATIO        MODE          RATIO         MODE          RATIO          MODE        RATIO

       1      MCT        1    2.34561E+00     WARP       4.16259E+00     FILL
                                                                                    1.81011E+00
              MCT        2    4.32664E+00     FILL       1.11014E+01     WARP
                                                                                    1.63026E+00
              MCT        3    4.54929E+00     FILL       8.64142E+00     WARP
                                                                                    1.81011E+00
              MCT        4    2.47623E+00     WARP       2.92297E+00     FILL                                  2.60945E+00

       2      LARC02     1    3.49234E-02    TENSION     7.21478E-01    TENSION
                                                                                    0.81011E+00
              LARC02     2    2.22249E-01    TENSION     1.09530E+00    TENSION
                                                                                    0.63026E+00
              LARC02     3    3.47694E-01  COMPRESSION   1.00000E+10
                                                                                    0.79011E+00
              LARC02     4    1.25142E-01  COMPRESSION   1.00000E+10                                           3.49234E-02   FAILED

MINIMUM QUAD ELEMENT PLY STRENGTH RATIO  =  3.492343E-02  AT ELEMENT 2
MINIMUM QUAD ELEMENT BOND STRENGTH RATIO =  0.790112E+00  AT ELEMENT 2
```

Remarks:

1. This output is available when composite stress/strain is output, allowable stresses/strains are supplied on the material property (`MAT1`, `MAT2`, or `MAT8`) and `PCOMP` entries that were used to define the laminate, and `PARAM`, `STRENGTHRATIO` is set to `ON`. See the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for more information.
2. This output is typical of all composite shell elements (`CQUAD4`, `CQUADR`, `CQUAD8`, `CTRIA3`, `CTRIAR`, and `CTRIA6`).
3. Strength ratios less than 1.0 indicate a failure in that ply using the failure criteria specified for the laminate (field 6 on the `PCOMP` Bulk Data entry).

Figure A-149. Strength Ratios in Composite Quad Elements.

```
                S T R E S S E S   I N   C O M P O S I T E   H E X   E L E M E N T S   I N   V O L U M E   0

 VOLUME COORDINATE ID = MATERIAL

   ELEMENT   PLY  STRESSES IN FIBER AND MATRIX DIRECTIONS         INTER-LAMINAR STRESSES            HENCKY
     ID      ID      NORMAL-1     NORMAL-2     SHEAR-12      NORMAL-3  SHEAR XZ-MAT SHEAR YZ-MAT    VON MISES

       7       1   1.05192E+04  8.31146E+02 -3.00309E+00  -3.17737E+00  9.15680E+01 -1.23555E+01   1.01323E+04
               2   1.79817E+03  9.18431E+01  1.10030E+03  -3.17737E+00  1.70297E+02 -2.29786E+01   2.60962E+03
               3  -3.10869E+02 -1.20708E+03  7.53370E+02  -3.17737E+00  1.19470E+02 -1.61203E+01   1.70968E+03
               4  -2.12105E+03 -1.19571E+03 -3.80401E+03  -3.17737E+00  1.50122E+02 -2.02563E+01   6.84590E+03
               5  -1.80386E+04 -1.97198E+03 -4.29770E+02   0.00000E+00  0.00000E+00  0.00000E+00   1.71527E+04

      10       1   1.12656E+04  8.94771E+02 -2.62804E+01  -3.16878E+00  1.80170E+02 -1.02792E+01   1.08526E+04
               2   1.99202E+03  9.28781E+01  1.17421E+03  -3.16878E+00  3.35078E+02 -1.91171E+01   2.88072E+03
               3  -3.28080E+02 -1.29486E+03  8.16504E+02  -3.16878E+00  2.35069E+02 -1.34113E+01   1.87946E+03
               4  -2.29524E+03 -1.27970E+03 -4.09224E+03  -3.16878E+00  2.95381E+02 -1.68523E+01   7.38074E+03
               5  -1.93370E+04 -2.10840E+03 -4.49207E+02   0.00000E+00  0.00000E+00  0.00000E+00   1.83902E+04


MAXIMUM HEX ELEMENT PRINCIPAL STRESS           =  1.126871E+04  AT ELEMENT 10
MINIMUM HEX ELEMENT PRINCIPAL STRESS           = -1.934984E+04  AT ELEMENT 10
MAXIMUM HEX ELEMENT INTERLAMINAR SHEAR STRESS =  3.350775E+02  AT ELEMENT 10
MINIMUM HEX ELEMENT INTERLAMINAR SHEAR STRESS = -2.297857E+01  AT ELEMENT 7
```

Remarks:

1. This output is requested using the `STRESS` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. This output is typical of all composite solid elements (`CHEX` and `CPENT`).
3. Direct inplane stresses are always output in the ply coordinate system (fiber and matrix direction). Interlaminar stresses are always output in the material coordinate system. The angle of principal stress is in the ply coordinate system.
4. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-150. Stresses in Composite Hex Elements.

```
                      N O N L I N E A R   S T R E S S E S   I N   Q U A D   E L E M E N T S

  ELEMENT       FIBER    STRESSES/STRAINS IN ELEMENT COORDINATE SYSTEM  EQUIVALENT    EFF. STRAIN
    ID         DISTANCE       NORMAL-X      NORMAL-Y      SHEAR-XY         STRESS    PLASTIC/NLELAST

      6     -2.50000E-02    3.56575E+03   1.57291E+02   1.63680E-02    3.48977E+03    3.12880E-05
                            2.80718E-04  -5.26699E-09   1.78774E-09
             2.50000E-02   -3.56417E+03  -1.54222E+02  -4.78810E-02    3.48961E+03    3.12544E-05
                           -2.80832E-04  -5.26699E-09  -8.49647E-09

      8     -2.50000E-02    2.06319E+03   5.04709E-03   5.11093E-02    2.06321E+03    0.00000E+00
                            1.47378E-04   5.04709E-10   7.04141E-09
             2.50000E-02   -2.06268E+03   5.04709E-03  -6.14061E-02    2.06266E+03    0.00000E+00
                           -1.47327E-04   5.04709E-10  -9.06078E-09

      9     -2.50000E-02    1.23783E+03   7.16163E-03   2.45897E-02    1.23786E+03    0.00000E+00
                            8.84251E-05   7.16163E-10   2.93989E-09
             2.50000E-02   -1.23722E+03   7.16160E-03  -4.46422E-02    1.23719E+03    0.00000E+00
                           -8.83640E-05   7.16161E-10  -6.95039E-09

     10     -2.50000E-02    4.12795E+02  -4.92269E-03   1.44631E-01    4.12835E+02    0.00000E+00
                            2.94948E-05  -4.92270E-10   2.25656E-08
             2.50000E-02   -4.12113E+02  -4.92271E-03  -7.79897E-02    4.12094E+02    0.00000E+00
                           -2.94286E-05  -4.92271E-10  -9.23736E-09


MAXIMUM QUAD ELEMENT EQUIVALENT STRESS =  3.489773E+03  AT ELEMENT 6
MINIMUM QUAD ELEMENT EQUIVALENT STRESS =  4.120938E+02  AT ELEMENT 10
MAXIMUM QUAD ELEMENT EFFECTIVE STRAIN  =  3.128798E-05  AT ELEMENT 6
MINIMUM QUAD ELEMENT EFFECTIVE STRAIN  =  0.000000E+00  AT ELEMENT 10
```

Remarks:

1. This output is requested using the `STRESS` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. This output is typical of all shell elements (`CQUAD4`, `CQUADR`, `CTRIA3`, and `CTRIAR`).

Figure A-151. Nonlinear Stresses in Quad Elements.

```
                         S T R E S S E S   I N   H E X   E L E M E N T S   I N   V O L U M E   1

  VOLUME COORDINATE ID = 2

    ELEMENT     GRID      STRESSES IN VOLUME COORDINATE SYSTEM          PRINCIPAL STRESSES               MEAN         HENCKY
      ID         ID            NORMAL            SHEAR              MAGNITUDE        DIRECTION COSINES      PRESSURE     VON MISES

       73      CENTER       X-0.11770E+04    XY-0.34153E+03      A-0.78385E+04    LX 0.39 -0.58  0.71     0.29557E+04    0.77686E+04
                            Y-0.18485E+04    YZ 0.22953E+04      B-0.20096E+04    LY-0.31 -0.81 -0.49
                            Z-0.56417E+04    ZX-0.30439E+04      C 0.98096E+03    LZ 0.87 -0.03 -0.50

       74      CENTER       X 0.40723E+04    XY-0.92118E+02      A 0.30633E+04    LX 0.97 -0.11  0.22    -0.10455E+05    0.19976E+05
                            Y 0.46117E+04    YZ 0.12483E+04      B 0.45563E+04    LY-0.12 -0.99 -0.06
                            Z 0.22678E+05    ZX-0.43955E+04      C 0.23744E+05    LZ 0.22  0.04 -0.97

       75      CENTER       X-0.32919E+04    XY 0.24347E+03      A-0.18187E+05    LX 0.23 -0.38  0.89     0.79953E+04    0.15329E+05
                            Y-0.34114E+04    YZ-0.96393E+03      B-0.35516E+04    LY 0.06  0.92  0.38
                            Z-0.17281E+05    ZX-0.35591E+04      C-0.22477E+04    LZ 0.97  0.04 -0.24

       76      CENTER       X 0.21270E+04    XY-0.72932E+02      A 0.73529E+03    LX 0.92 -0.20  0.33    -0.54293E+04    0.11003E+05
                            Y 0.29855E+04    YZ 0.12527E+04      B 0.28944E+04    LY-0.16 -0.98 -0.12
                            Z 0.11175E+05    ZX-0.37197E+04      C 0.12658E+05    LZ 0.35  0.06 -0.94


 MAXIMUM HEX ELEMENT PRINCIPAL STRESS =  0.237443E+05  AT ELEMENT 74
 MINIMUM HEX ELEMENT PRINCIPAL STRESS = -0.181867E+05  AT ELEMENT 75
 MAXIMUM HEX ELEMENT SHEAR STRESS     =  0.941695E+04  AT ELEMENT 74
 MAXIMUM HEX ELEMENT VON MISES STRESS =  0.199764E+05  AT ELEMENT 74
```

Remarks:

1. This output is requested using the `STRESS` Case Control command. Corner stress output is requested by using the `STRESS(CORNER)` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. This output is typical of all solid elements (`CHEXA`, `CPENTA`, and `CTETRA`).
3. Stresses may be output in any `VOLUME` coordinate system (`ELEMENT`, `GRID`, `BASIC`, `MATERIAL`, or user defined). In the above example, stresses are output in the basic coordinate system (`VOLUME COORDINATE ID = 2`). See `VOLUME` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
4. The direction cosines of the principal stress are with respect to the `VOLUME` coordinate system.
5. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-152. Stresses in Hex Elements (Without Corner Option).

```
                    S T R A I N S   I N   H E X   E L E M E N T S   I N   V O L U M E   1

  VOLUME COORDINATE ID = 0

    ELEMENT     GRID      STRAINS IN VOLUME COORDINATE SYSTEM           PRINCIPAL STRAINS                 MEAN         HENCKY
      ID         ID          NORMAL              SHEAR            MAGNITUDE       DIRECTION COSINES        PRESSURE     VON MISES

       73      CENTER       X 0.87006E-04    XY-0.88799E-04      A-0.11968E-02    LX 0.47 -0.60  0.65     0.11823E-03    0.67328E-03
                            Y 0.25711E-04    YZ 0.59677E-03      B-0.37454E-04    LY-0.36 -0.80 -0.48
                            Z-0.46740E-03    ZX-0.79143E-03      C 0.87959E-03    LZ 0.81  0.00 -0.59

       74      CENTER       X-0.41152E-03    XY-0.23956E-04      A-0.88462E-03    LX 0.91 -0.21  0.36    -0.41819E-03    0.17311E-02
                            Y-0.34114E-03    YZ 0.32456E-03      B-0.35660E-03    LY-0.19 -0.98 -0.11
                            Z 0.20072E-02    ZX-0.11428E-02      C 0.24958E-02    LZ 0.38  0.03 -0.92

       75      CENTER       X 0.29164E-03    XY 0.63303E-04      A-0.19344E-02    LX 0.38 -0.34  0.86     0.31981E-03    0.11285E-02
                            Y 0.27584E-03    YZ-0.25062E-03      B 0.24071E-03    LY 0.09  0.94  0.33
                            Z-0.15269E-02    ZX-0.92536E-03      C 0.73423E-03    LZ 0.92  0.05 -0.39

       76      CENTER       X-0.21211E-03    XY-0.18962E-04      A-0.78744E-03    LX 0.84 -0.27  0.47    -0.21717E-03    0.95363E-03
                            Y-0.10052E-03    YZ 0.32571E-03      B-0.12104E-03    LY-0.22 -0.96 -0.17
                            Z 0.96417E-03    ZX-0.96711E-03      C 0.15600E-02    LZ 0.50  0.04 -0.86


 MAXIMUM HEX ELEMENT PRINCIPAL STRAIN =  0.249578E-02  AT ELEMENT 74
 MINIMUM HEX ELEMENT PRINCIPAL STRAIN = -0.193438E-02  AT ELEMENT 75
 MAXIMUM HEX ELEMENT SHEAR STRAIN     =  0.122420E-02  AT ELEMENT 74
 MAXIMUM HEX ELEMENT VON MISES STRAIN =  0.173129E-02  AT ELEMENT 74
```

Remarks:

1. This output is requested using the `STRAIN` Case Control command. Corner stress output is requested by using the `STRAIN(CORNER)` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. This output is typical of all solid elements (`CHEXA`, `CPENTA`, and `CTETRA`).
3. Strains may be output in any `VOLUME` coordinate system (`ELEMENT`, `GRID`, `BASIC`, `MATERIAL`, or user defined). In the above example, strains are output in the basic coordinate system (`VOLUME COORDINATE ID = 0`). See `VOLUME` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
4. The direction cosines of the principal strains are with respect to the `VOLUME` coordinate system.
5. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-153. Strains in Hex Elements (Without Corner Option).

```
                    S T R A I N   E N E R G Y   I N   H E X   E L E M E N T S   I N   V O L U M E   200

      ELEMENT     STRAIN ENERGY       PERCENT       STRAIN ENERGY
        ID                             TOTAL           DENSITY
         69       0.597955E+00         0.0040       0.478363E+01
         70       0.261125E+01         0.0173       0.208899E+02
         71       0.153824E+01         0.0102       0.123059E+02
         72       0.598673E+00         0.0040       0.478937E+01
         73       0.779929E+00         0.0052       0.623943E+01
         74       0.347546E+01         0.0230       0.278037E+02
         75       0.220775E+01         0.0146       0.176620E+02
         76       0.129356E+01         0.0086       0.103485E+02


      SUBTOTAL    0.111028E+02         0.0866


   MAXIMUM HEX ELEMENT STRAIN ENERGY DENSITY =  0.278037E+02  AT ELEMENT 74
   MINIMUM HEX ELEMENT STRAIN ENERGY DENSITY =  0.478363E+01  AT ELEMENT 69
```

Remarks:

1. This output is requested using the `ESE` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. This output is typical of all solid elements (`CHEXA`, `CPENTA`, and `CTETRA`).
3. Percentages are based on the entire model not the individual volume.
4. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-154. Strain Energy in Hex Elements.

```
  T H E R M A L   G R A D I E N T S   A N D   H E A T   F L U X E S   I N   H E X   E L E M E N T S   I N   V O L U M E   1

VOLUME COORDINATE ID = ELEMENT

  ELEMENT     GRID      X-GRADIENT   Y-GRADIENT   Z-GRADIENT         X-FLUX       Y-FLUX       Z-FLUX
    ID         ID
     1      CENTER     2.76897E+04 -1.79641E+01  3.97245E+01     -5.76777E+00  3.74192E-03 -8.27462E-03
     2      CENTER     2.79017E+04 -1.18061E+02 -5.92445E+01     -5.81191E+00  2.45920E-02  1.23406E-02
     3      CENTER     2.84411E+04 -1.55285E+02  1.97697E+02     -5.92432E+00  3.23458E-02 -4.11804E-02
     4      CENTER     2.78650E+04  2.58710E+02 -7.33032E+02     -5.80429E+00 -5.38893E-02  1.52691E-01
     5      CENTER     2.67981E+04 -1.11993E+03  2.73993E+03     -5.58204E+00  2.33281E-01 -5.70728E-01
     6      CENTER     2.76716E+04  1.97119E+01 -4.91855E+01     -5.76400E+00 -4.10599E-03  1.02453E-02
     7      CENTER     2.77816E+04 -1.56552E+02 -2.27186E+01     -5.78691E+00  3.26098E-02  4.73229E-03
     8      CENTER     2.85712E+04 -9.94829E+01  1.40234E+02     -5.95119E+00  2.07223E-02 -2.92107E-02
     9      CENTER     2.78649E+04 -2.53870E+02 -5.39261E+02     -5.80425E+00  5.28812E-02  1.12328E-01
    10      CENTER     2.67305E+04  9.89251E+02  2.02086E+03     -5.56796E+00 -2.06061E-01 -4.20945E-01


MAXIMUM HEX ELEMENT THERMAL GRADIENT MAGNITUDE =  2.857176E+04  AT ELEMENT 8
MINIMUM HEX ELEMENT THERMAL GRADIENT MAGNITUDE =  2.682501E+04  AT ELEMENT 10
MAXIMUM HEX ELEMENT HEAT FLUX MAGNITUDE        =  5.951498E+00  AT ELEMENT 8
MINIMUM HEX ELEMENT HEAT FLUX MAGNITUDE        =  5.587649E+00  AT ELEMENT 10
```

Remarks:

1. This output is requested using the `FLUX` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. This output is typical of all solid elements (`CHEXA`, `CPENTA`, and `CTETRA`).
3. Element stresses may be output in any `VOLUME` coordinate system (`ELEMENT`, `GRID`, `BASIC`, `MATERIAL`, or user defined). In the above example, thermal gradients and heat fluxes are output in `ELEMENT` coordinate system. See `VOLUME` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information. See `CHEXA` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for the definition of element coordinate system.

Figure A-155. Thermal Gradients and Heat Fluxes in Hex Elements.

```
                              G R I D   P O I N T   S T R E S S E S   I N   V O L U M E  14

 VOLUME COORDINATE ID = 10

    GRID      STRESSES IN VOLUME COORDINATE SYSTEM          PRINCIPAL STRESSES                  MEAN         HENCKY
     ID           NORMAL             SHEAR            MAGNITUDE        DIRECTION COSINES        PRESSURE       VON MISES

       2        X 1.29593E+04    XY 1.91679E-08      A-4.97633E+00    LX 0.02  0.00  1.00    -4.31977E+03    1.29668E+04
                Y 0.00000E+00    YZ-9.64888E-11      B 5.09317E-11    LY 0.00  1.00  0.00
                Z 8.22112E-10    ZX 2.53997E+02      C 1.29643E+04    LZ-1.00  0.00  0.02

       3        X 1.29593E+04    XY 1.99081E-08      A-4.97633E+00    LX 0.02  0.00  1.00    -4.31977E+03    1.29668E+04
                Y 0.00000E+00    YZ 8.30516E-09      B-1.81899E-10    LY 0.00  0.00  0.00
                Z-1.21144E-10    ZX 2.53997E+02      C 1.29643E+04    LZ-1.00  1.00  0.02

      49        X-1.29593E+04    XY-2.06680E-08      A-1.29643E+04    LX 1.00  0.00  0.02     4.31977E+03    1.29668E+04
                Y 0.00000E+00    YZ 8.31785E-09      B-1.63709E-10    LY 0.00  0.00  0.00
                Z-1.63469E-10    ZX 2.53997E+02      C 4.97633E+00    LZ-0.02  1.00  1.00

      50        X-2.51948E+03    XY-6.15041E-08      A-2.53146E+03    LX 1.00  0.00  0.07     8.41605E+02    2.53479E+03
                Y 3.29563E-09    YZ 2.78953E-09      B 3.28487E-09    LY 0.00  1.00  0.00
                Z-5.33971E+00    ZX 1.73997E+02      C 6.64506E+00    LZ-0.07  0.00  1.00


MAXIMUM SOLID ELEMENT PRINCIPAL STRESS =  1.296428E+04  AT GRID 3
MINIMUM SOLID ELEMENT PRINCIPAL STRESS = -1.296428E+04  AT GRID 49
MAXIMUM SOLID ELEMENT SHEAR STRESS     =  6.112592E+03  AT GRID 49
MAXIMUM SOLID ELEMENT VON MISES STRESS =  1.296677E+04  AT GRID 49
```

Remarks:

1. This output is requested using the `GPSTRESS` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. Grid point stresses may be output in any `VOLUME` coordinate system (`GRID`, `BASIC`, `MATERIAL`, or user defined) except `ELEMENT`. In the above example stresses are output in a user defined `VOLUME` coordinate system (`VOLUME COORDINATE ID = 10`). See `VOLUME` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
3. The direction cosines of the principal stresses are with respect to the `VOLUME` coordinate system.
4. Prestress results can be excluded from the above output by including `PARAM`, `ADDPRESTRESS`, `OFF` in the Model Input File (`PRESTRESS STATIC` and `PRESTRESS MODAL` solutions only). See `ADDPRESTRESS` in the *Nastran Solver Reference Guide*, Section 5, *Parameters.*

Figure A-156. Volume Grid Point Stresses.

```
       G R I D   P O I N T   T H E R M A L   G R A D I E N T S   A N D   H E A T   F L U X E S   I N   V O L U M E   1

   VOLUME COORDINATE ID = GRID

      GRID       X-GRADIENT   Y-GRADIENT   Z-GRADIENT         X-FLUX       Y-FLUX       Z-FLUX
       ID
        40       2.79953E+04 -1.78471E+02  1.25428E+03     -5.83143E+00  3.71754E-02 -2.61266E-01
        41       2.78796E+04  8.92464E+02 -4.68607E+03     -5.80732E+00 -1.85900E-01  9.76108E-01
        42       2.82252E+04 -3.86822E+03  1.75250E+04     -5.87931E+00  8.05751E-01 -3.65046E+00
        43       2.76755E+04  7.50000E+00  2.20000E+02     -5.76480E+00 -1.56225E-03 -4.58260E-02
        44       2.77687E+04 -1.46432E+01 -7.09963E+01     -5.78423E+00  3.05017E-03  1.47885E-02
        45       2.81607E+04 -2.56317E+02  6.45286E+01     -5.86587E+00  5.33908E-02 -1.34411E-02
        46       2.82644E+04 -4.19699E+00 -1.87611E+02     -5.88748E+00  8.74233E-04  3.90795E-02
        47       2.69572E+04  2.84141E+01  6.87277E+02     -5.61517E+00 -5.91865E-03 -1.43160E-01
        48       2.58449E+04 -2.30997E+02 -2.56648E+03     -5.38348E+00  4.81166E-02  5.34597E-01
        49       2.76798E+04  3.00000E+00 -5.60000E+02     -5.76571E+00 -6.24900E-04  1.16648E-01
        50       2.76858E+04  1.16864E+01  2.14254E+02     -5.76695E+00 -2.43427E-03 -4.46292E-02
        51       2.82003E+04 -3.28687E+02 -2.98661E+02     -5.87412E+00  6.84656E-02  6.22111E-02


  MAXIMUM SOLID ELEMENT THERMAL GRADIENT MAGNITUDE =  3.344773E+04  AT GRID 42
  MINIMUM SOLID ELEMENT THERMAL GRADIENT MAGNITUDE =  2.597300E+04  AT GRID 48
  MAXIMUM SOLID ELEMENT HEAT FLUX MAGNITUDE        =  6.967163E+00  AT GRID 42
  MINIMUM SOLID ELEMENT HEAT FLUX MAGNITUDE        =  5.410176E+00  AT GRID 48
```

Remarks:

1. This output is requested using the `GPFLUX` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. Thermal gradients and heat fluxes may be output in any `VOLUME` coordinate system (`GRID`, `BASIC`, `MATERIAL`, or user defined) except `ELEMENT`. In the above example, thermal gradients and heat fluxes are output in the `GRID` or displacement coordinate system (field 7 on the `GRID` Bulk Data entry). See `VOLUME` in the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information. See `CHEXA` in the *Nastran Solver Reference Guide*, Section 4, *Bulk Data,* for the definition of element coordinate system.

Figure A-157. Volume Grid Point Thermal Gradients and Heat Fluxes.

```
                                    N O N L I N E A R   S T R E S S E S   I N   H E X   E L E M E N T S

  ELEMENT                      STRESSES/STRAINS IN ELEMENT COORDINATE SYSTEM                           EQUIVALENT     EFF. STRAIN
    ID        NORMAL-X      NORMAL-Y      NORMAL-Z      SHEAR-XY      SHEAR-YZ      SHEAR-ZX          STRESS     PLASTIC/NLELAST

      31      4.85653E+01   4.85773E+01  -1.84711E+03   4.85990E+01  -1.99415E+02  -1.99431E+02    1.89217E+03    5.46869E-03
              1.40362E-04   1.40396E-04  -5.33844E-03   2.80918E-04  -1.15268E-03  -1.15278E-03

      32      4.85653E+01   4.85773E+01  -1.84711E+03   4.85990E+01  -1.99415E+02  -1.99431E+02    1.89217E+03    5.46869E-03
              1.40362E-04   1.40396E-04  -5.33844E-03   2.80918E-04  -1.15268E-03  -1.15278E-03


MAXIMUM HEX ELEMENT EQUIVALENT STRESS =  1.892174E+03  AT ELEMENT 32
MINIMUM HEX ELEMENT EQUIVALENT STRESS =  1.892174E+03  AT ELEMENT 32
MAXIMUM HEX ELEMENT EFFECTIVE STRAIN  =  5.468695E-03  AT ELEMENT 32
MINIMUM HEX ELEMENT EFFECTIVE STRAIN  =  5.468695E-03  AT ELEMENT 32
```

Remarks:

1. This output is requested using the `STRESS` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. This output is typical of all solid elements (`CHEXA`, `CPENTA`, and `CTETRA`).

Figure A-158. Nonlinear Stresses in Hex Elements.

```
                    C O M P L E X   S T R E S S E S   I N   H E X   E L E M E N T S   I N   V O L U M E   0
                                                      (MAGNITUDE/PHASE)

  VOLUME COORDINATE ID = ELEMENT

    ELEMENT     GRID                            STRESSES IN VOLUME COORDINATE SYSTEM
      ID         ID           NORMAL-X     NORMAL-Y     NORMAL-Z     SHEAR-XY     SHEAR-YZ     SHEAR-ZX

       39      CENTER        1.17697E+03  4.19053E-01  6.52153E+00  4.99829E+02  7.00869E-03  3.63761E-01
                             3.59145E+02  5.77118E+01  3.44029E+00  3.58664E+02  1.78626E+02  3.59109E+02

       40      CENTER        3.57107E+02  4.53223E+01  4.67287E+01  2.95935E+02  2.04734E-02  1.65107E-01
                             3.59350E+02  1.79239E+02  1.79182E+02  3.59122E+02  1.79183E+02  1.78814E+02

       41      CENTER        8.18110E+03  1.11817E+03  1.95108E+02  5.82602E+02  4.65842E+01  7.82209E+00
                             1.77772E+02  1.77734E+02  1.77610E+02  3.57170E+02  3.57760E+02  1.77737E+02

       42      CENTER        7.11986E+03  3.15651E+02  6.39725E+01  5.90102E+02  1.24033E+01  5.44894E+00
                             1.77844E+02  3.57731E+02  3.57622E+02  3.57214E+02  1.77760E+02  3.57768E+02


 MAXIMUM HEX ELEMENT PRINCIPAL STRESS =  8.285501E+03  AT ELEMENT 41
 MINIMUM HEX ELEMENT PRINCIPAL STRESS = -8.233173E+03  AT ELEMENT 41
 MAXIMUM HEX ELEMENT SHEAR STRESS     =  6.743822E+03  AT ELEMENT 41
 MAXIMUM HEX ELEMENT VON MISES STRESS =  1.430581E+04  AT ELEMENT 41
```

Remarks:

1. This output is requested using the `STRESS` Case Control command in frequency response solutions. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. This output is typical of all solid elements (`CHEXA`, `CPENTA`, and `CTETRA`).
3. The REAL or IMAG option requests complex output in rectangular format (real and imaginary). The PHASE option requests complex output in polar format (magnitude and phase) as shown. Phase output is in degrees.
4. Maximums results reported are determined using MAGNITUDE/PHASE results.

Figure A-159. Complex Stresses in Hex Elements.

```
                      E L E M E N T   R E S U L T   L I M I T S    (S U B C A S E   S E A R C H)

     SET     OUTPUT SET    RESULT     SUBCASE     STEP        ELEMENT    ELEMENT         LIMIT              RESULT             RESULT
      ID         ID        NUMBER       ID         ID          TYPE         ID           TYPE                TYPE

       1          1          22          1          0           SHELL      5644      MAXIMUM            MAX VON MISES-1/2      1.100129E+03
                                         2          0           SHELL      4316      MINIMUM            MAX VON MISES-1/2      1.875387E-03
                                         1          0           SHELL      5644      MAXIMUM ABSOLUTE   MAX VON MISES-1/2      1.100129E+03
                                         2          0           SHELL      4316      MINIMUM ABSOLUTE   MAX VON MISES-1/2      1.875387E-03

       2          1          23          1          0           SHELL      5644      MAXIMUM            MAX SHEAR-1/2          5.777335E+02
                                         2          0           SHELL      4315      MINIMUM            MAX SHEAR-1/2          9.067865E-04
                                         1          0           SHELL      5644      MAXIMUM ABSOLUTE   MAX SHEAR-1/2          5.777335E+02
                                         2          0           SHELL      4315      MINIMUM ABSOLUTE   MAX SHEAR-1/2          9.067865E-04

       3          1          24          1          0           SHELL     13871      MAXIMUM            MAX PRINCIPAL-1/2      5.938117E+02
                                         1          0           SHELL      5900      MINIMUM            MAX PRINCIPAL-1/2     -2.230217E-01
                                         1          0           SHELL     13871      MAXIMUM ABSOLUTE   MAX PRINCIPAL-1/2      5.938117E+02
                                         2          0           SHELL      4307      MINIMUM ABSOLUTE   MAX PRINCIPAL-1/2     -1.294128E-04

       4          1          25          2          0           SHELL      7860      MAXIMUM            MIN PRINCIPAL-1/2      3.737456E+01
                                         1          0           SHELL      5644      MINIMUM            MIN PRINCIPAL-1/2     -1.034851E+03
                                         1          0           SHELL      5644      MAXIMUM ABSOLUTE   MIN PRINCIPAL-1/2     -1.034851E+03
                                         2          0           SHELL      4343      MINIMUM ABSOLUTE   MIN PRINCIPAL-1/2     -1.425817E-06
```

Remarks:

1. This output is requested using the `RESULTLIMITS` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. For each result limits set the subcase search identifies limits in all specified subcases for all specified elements. Only subcases and elements identified on the `RESULTLIMITS` Case Control command will be included in the search.
3. RESULT TYPE and RESULT NUMBER are defined in the *Nastran Solver Reference Guide*, Appendix A, *Results Neutral File Format*.

Figure A-160. Element Result Limits (Subcase Search).

```
                        E L E M E N T   R E S U L T   L I M I T S    (G L O B A L   S E A R C H)

RESULT     SET      SUBCASE    STEP       ELEMENT    ELEMENT         LIMIT             RESULT               RESULT
NUMBER      ID        ID        ID         TYPE        ID            TYPE               TYPE

  22          1         1         0         SHELL     5644      MAXIMUM           MAX VON MISES-1/2      1.100129E+03
              1         2         0         SHELL     4316      MINIMUM           MAX VON MISES-1/2      1.875387E-03
              1         1         0         SHELL     5644      MAXIMUM ABSOLUTE  MAX VON MISES-1/2      1.100129E+03
              1         2         0         SHELL     4316      MINIMUM ABSOLUTE  MAX VON MISES-1/2      1.875387E-03

  23          2         1         0         SHELL     5644      MAXIMUM           MAX SHEAR-1/2          5.777335E+02
              2         2         0         SHELL     4315      MINIMUM           MAX SHEAR-1/2          9.067865E-04
              2         1         0         SHELL     5644      MAXIMUM ABSOLUTE  MAX SHEAR-1/2          5.777335E+02
              2         2         0         SHELL     4315      MINIMUM ABSOLUTE  MAX SHEAR-1/2          9.067865E-04

  24          3         1         0         SHELL    13871      MAXIMUM           MAX PRINCIPAL-1/2      5.938117E+02
              3         1         0         SHELL     5900      MINIMUM           MAX PRINCIPAL-1/2     -2.230217E-01
              3         1         0         SHELL    13871      MAXIMUM ABSOLUTE  MAX PRINCIPAL-1/2      5.938117E+02
              3         2         0         SHELL     4307      MINIMUM ABSOLUTE  MAX PRINCIPAL-1/2     -1.294128E-04

  25          4         2         0         SHELL     7860      MAXIMUM           MIN PRINCIPAL-1/2      3.737456E+01
              4         1         0         SHELL     5644      MINIMUM           MIN PRINCIPAL-1/2     -1.034851E+03
              4         1         0         SHELL     5644      MAXIMUM ABSOLUTE  MIN PRINCIPAL-1/2     -1.034851E+03
              4         2         0         SHELL     4343      MINIMUM ABSOLUTE  MIN PRINCIPAL-1/2     -1.425817E-06
```

Remarks:

1. This output is requested using the `RESULTLIMITS` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. For all result limits sets the global search identifies limits for each RESULT NUMBER specified.
3. RESULT TYPE and RESULT NUMBER are defined in the *Nastran Solver Reference Guide*, Appendix A, *Results Neutral File Format*.

Figure A-161. Element Result Limits (Global Search).

```
                    G R I D   P O I N T   R E S U L T   L I M I T S   (S U B C A S E   S E A R C H)

  SET    OUTPUT SET   RESULT    SUBCASE    STEP      ELEMENT     GRID          LIMIT             RESULT              RESULT
  ID        ID        NUMBER      ID        ID        TYPE        ID           TYPE               TYPE

    5         1         22         1         0         SHELL      313      MAXIMUM           MAX VON MISES-1/2      1.064701E+03
                                   2         0         SHELL       88      MINIMUM           MAX VON MISES-1/2      2.498522E-03
                                   1         0         SHELL      313      MAXIMUM ABSOLUTE  MAX VON MISES-1/2      1.064701E+03
                                   2         0         SHELL       88      MINIMUM ABSOLUTE  MAX VON MISES-1/2      2.498522E-03

    6         1         23         1         0         SHELL      313      MAXIMUM           MAX SHEAR-1/2          5.405927E+02
                                   2         0         SHELL      136      MINIMUM           MAX SHEAR-1/2          6.349614E-04
                                   1         0         SHELL      313      MAXIMUM ABSOLUTE  MAX SHEAR-1/2          5.405927E+02
                                   2         0         SHELL      136      MINIMUM ABSOLUTE  MAX SHEAR-1/2          6.349614E-04

    7         1         24         1         0         SHELL      753      MAXIMUM           MAX PRINCIPAL-1/2      6.182511E+02
                                   1         0         SHELL      308      MINIMUM           MAX PRINCIPAL-1/2     -8.994402E+01
                                   1         0         SHELL      753      MAXIMUM ABSOLUTE  MAX PRINCIPAL-1/2      6.182511E+02
                                   2         0         SHELL      178      MINIMUM ABSOLUTE  MAX PRINCIPAL-1/2     -6.550865E-05

    8         1         25         2         0         SHELL      984      MAXIMUM           MIN PRINCIPAL-1/2      9.573911E+01
                                   1         0         SHELL      313      MINIMUM           MIN PRINCIPAL-1/2     -1.047412E+03
                                   1         0         SHELL      313      MAXIMUM ABSOLUTE  MIN PRINCIPAL-1/2     -1.047412E+03
                                   2         0         SHELL        3      MINIMUM ABSOLUTE  MIN PRINCIPAL-1/2     -2.633753E-06
```

Remarks:

1. This output is requested using the `RESULTLIMITS` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. The subcase search identifies limits for each result limits set in all specified subcases for all specified grid points. Only subcases and grid points identified on the `RESULTLIMITS` Case Control command will be included in the search.
3. RESULT TYPE and RESULT NUMBER are defined in the *Nastran Solver Reference Guide*, Appendix A, *Results Neutral File Format*.

Figure A-162. Grid Point Result Limits (Subcase Search).

```
                     G R I D   P O I N T   R E S U L T   L I M I T S    (G L O B A L   S E A R C H)

RESULT     SET      SUBCASE    STEP       ELEMENT     GRID           LIMIT             RESULT               RESULT
NUMBER      ID        ID        ID         TYPE        ID            TYPE               TYPE

  22          5         1         0         SHELL      313      MAXIMUM           MAX VON MISES-1/2      1.064701E+03
              5         2         0         SHELL       88      MINIMUM           MAX VON MISES-1/2      2.498522E-03
              5         1         0         SHELL      313      MAXIMUM ABSOLUTE  MAX VON MISES-1/2      1.064701E+03
              5         2         0         SHELL       88      MINIMUM ABSOLUTE  MAX VON MISES-1/2      2.498522E-03

  23          6         1         0         SHELL      313      MAXIMUM           MAX SHEAR-1/2          5.405927E+02
              6         2         0         SHELL      136      MINIMUM           MAX SHEAR-1/2          6.349614E-04
              6         1         0         SHELL      313      MAXIMUM ABSOLUTE  MAX SHEAR-1/2          5.405927E+02
              6         2         0         SHELL      136      MINIMUM ABSOLUTE  MAX SHEAR-1/2          6.349614E-04

  24          7         1         0         SHELL      753      MAXIMUM           MAX PRINCIPAL-1/2      6.182511E+02
              7         1         0         SHELL      308      MINIMUM           MAX PRINCIPAL-1/2     -8.994402E+01
              7         1         0         SHELL      753      MAXIMUM ABSOLUTE  MAX PRINCIPAL-1/2      6.182511E+02
              7         2         0         SHELL      178      MINIMUM ABSOLUTE  MAX PRINCIPAL-1/2     -6.550865E-05

  25          8         2         0         SHELL      984      MAXIMUM           MIN PRINCIPAL-1/2      9.573911E+01
              8         1         0         SHELL      313      MINIMUM           MIN PRINCIPAL-1/2     -1.047412E+03
              8         1         0         SHELL      313      MAXIMUM ABSOLUTE  MIN PRINCIPAL-1/2     -1.047412E+03
              8         2         0         SHELL        3      MINIMUM ABSOLUTE  MIN PRINCIPAL-1/2     -2.633753E-06
```

Remarks:

1. This output is requested using the `RESULTLIMITS` Case Control command. See the *Nastran Solver Reference Guide*, Section 3, *Case Control,* for more information.
2. The global search identifies limits for all result limits sets for each RESULT NUMBER specified.
3. RESULT TYPE and RESULT NUMBER are defined in the *Nastran Solver Reference Guide*, Appendix A, *Results Neutral File Format*.

Figure A-163. Grid Point Result Limits (Global Search).

```
MODEL ANALYSIS TIME SUMMARY

TOTAL CPU TIME = 0.9 SECONDS
WALLCLOCK TIME = 1.0 SECONDS

EXECUTION TERMINATED NORMALLY

TOTAL WARNINGS     = 0
TOTAL FATAL ERRORS = 0
```

Remarks:

1. Always check the Model Results Output File when a warning or fatal error is encountered. System errors such as I/O or memory management errors are written to the System File (see Section 7, *Error Messages,* for more information).
2. The output timing format can be changed to hours/minutes/seconds by setting the Model Initialization directive, `SECONDS`, to `OFF`. See the *Nastran Solver Reference Guide*, Section 2, *Initialization,* for more information.

Figure A-164. Execution Summary.

APPENDIX B - LIMITS

Models in Autodesk Inventor Nastran are generally only limited by available disk space. There is, however, one limit that may affect your ability to run very large models. It is the maximum global matrix size (stored non-zero matrix terms) and it cannot be greater than 2,147,483,647. For very large models which may exceed this limit the following settings are recommended:

```
DECOMPMETHOD = PCGLSS
EXTRACTMETHOD = LANCZOS
SPARSEITERMODE = 3
```

These settings will avoid assembling global matrixes and will handle most operations at the element level. The following table lists other size limitations:

Description	**Limit**	**Entry/Command**
Output Sets	1000	`SET` Command
DDAM Data	20	`DDAMDAT` Entry
X-Y Plot	10,000	`XYDATA` Command
Modal Sets	100	`MODESET` Command
Viscoelastic Material Coefficients	120	`TABVE` Entry (number of series terms)
2-D Layered Shell Element Plies	1000	`PCOMP` Entry (number of plies)
3-D Layered Solid Element Plies	500	`PCOMP` Entry (number of plies)
Superelements	10,000	N/A

APPENDIX C - REFERENCES

1. Bathe, Klas-Jürgen, *Finite Element Procedures in Engineering Analysis.* Prentice-Hall Inc., 1982.
2. Belsheim, R. O. and O'Hara, G. J., *Shock Design of Shipboard Equipment, Part I, Dynamic Design Analysis Method*, NAVSHIPS 250-430, May 1962.
3. Cifuentes, Arturo O., *Using MSC/NASTRAN: Statics and Dynamics*. New York, NY: Springer-Verlag, Inc., 1989.
4. Cook, Robert D., Malkus, David S., and Plesha, Michael E., *Concepts and Applications of Finite Element Analysis*. New York, NY: John Wiley and Sons, 1989.
5. Dávila, Carlos G., Jaunky, Navin, and Goswami, Sanjib, *Failure Criteria for FRP Laminates in Plane Stress*, NASA Langley Research Center, December 2001.
6. Hurty, Walter C. and Rubenstein, Moshe F., *Dynamics of Structures.* Prentice-Hall Inc., 1964.
7. Incropera, Frank P. and DeWitt, David P., *Fundamentals of Heat and Mass Transfer, Second Edition.* John Wiley & Sons, Inc., 1985.
8. Jones, R. T., *Mechanics of Composite Materials.* McGraw-Hill, 1975.
9. Ley, Robert P., Weichuan, Lin, and Mbanefo, Uy, *Facesheet Wrinkling in Sandwich Structures*, NASA-CR-1999-208994, 1999.
10. MacNeal, Richard H., *Finite Elements: Their Design and Performance.* New York, NY: Marcel Dekker, Inc., 1994.
11. MacNeal, R. H. (ed), *The NASTRAN Theoretical manual.* December 1972.
12. Puck, A. and Schürmann, H., *Failure Analysis of FRP Laminates by Means of Physically Based Phenomenological Models*, Composites Science and Technology, Volume 58, Issue 7, July 1998.
13. Puck, A., Kopp, J., and Knops, M., *Guidelines for the Determination of the Parameters in Puck's Action Plane Strength Criterion*, Composites Science and Technology, Volume 62, Issue 3, February 2002.
14. Roark, R. J. and Young, W. C., *Formulas for Stress and Strain*, McGraw-Hill, Fifth Edition, 1975.
15. Schaeffer, Harry G., *MSC/NASTRAN Primer: Static and Normal Modes Analysis*. Milford, NH: Wallace Press, Inc., 1984.
16. *Shock Design Criteria for Surface Ships,* Naval Sea Command, NAVSEA 0908-LP-000-3010 Revision 1, September 1995.
17. Timoshenko, S., Young, D. H., Weaver, W. Jr., *Vibration Problems in Engineering*, Fourth Edition, John Wiley & Sons Inc., 1974.
18. Timoshenko, S., *Theory of Elastic Stability*, McGraw-Hill, 1936.
19. Tsai, Stephen W., *Theory of Composites Design*, Think Composites, Ohio, 1992.
20. Mayes J.S. and Hansen A.C., *Multicontinuum Failure Analysis of Composite Structural Laminates*, Mechanics of Composite Materials and Structures**,** 2001; 8(4):249-262.

21. Mayes J.S. and Hansen A.C., *Composite Laminate Failure Analysis Using Multicontinuum Theory*, Composites Science and Technology, 2004; 64(3-4):379-394.
22. Hansen A.C., Kenik D.J., and Nelson E.E., *Multicontinuum Failure Analysis of Composites*, Proceedings of the 17th International Conference of Composites Materials (ICCM-17), Edinburgh, Scotland, 2009.
23. Soden P.D., Hinton M.J., Kaddour A.S., *Lamina Properties, Lay-Up Configurations and Loading Conditions for A Range of Fibre-Reinforced Composite Laminates*, Composites Science and Technology, 1998; 58:7 1011.