

MSC Nastran 2022.2

Verification Guide

HEXAGON

Americas
5161 California Ave. Suite 200
University Research Park
Irvine, CA 92617
Telephone: (714) 540-8900
Email: americas.contact@mscsoftware.com

Europe, Middle East, Africa
Am Moosfeld 13
81829 Munich, Germany
Telephone: (49) 89 431 98 70
Email: europe@mscsoftware.com

Japan
KANDA SQUARE 16F
2-2-1 Kanda Nishikicho, Chiyoda-ku
1-Chome, Shinjuku-Ku
Tokyo 101-0054, Japan
Telephone: (81)(3) 6275 0870
Email: MSCJ.Market@mscsoftware.com

Asia-Pacific
100 Beach Road
#16-05 Shaw Tower
Singapore 189702
Telephone: 65-6272-0082
Email: APAC.Contact@mscsoftware.com

Worldwide Web
www.mscsoftware.com, www.hexagon.com

Support
https://simcompanion.hexagon.com

Disclaimer

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

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

User Documentation: Copyright © 2022 Hexagon AB and/or its subsidiaries. All Rights Reserved.

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

This software may contain certain third-party software that is protected by copyright and licensed from Hexagon suppliers. Additional terms and conditions and/or notices may apply for certain third party software. Such additional third party software terms and conditions and/or notices may be set forth in documentation and/or at http://www.mscsoftware.com/thirdpartysoftware (or successor website designated by Hexagon from time to time).

PCGLSS 8.0, Copyright © 1992-2016, Computational Applications and System Integration Inc. All rights reserved. PCGLSS 8.0 is licensed from Computational Applications and System Integration Inc.

The Hexagon logo, Hexagon, MSC Software logo, MSC, Dytran, Marc, MSC Nastran, Patran, e-Xstream, Digimat, and Simulating Reality are trademarks or registered trademarks of Hexagon AB and/or its subsidiaries in the United States and/or other countries.

NASTRAN is a registered trademark of NASA. FLEXlm and FlexNet Publisher are trademarks or registered trademarks of Flexera Software. All other trademarks are the property of their respective owners.

Use, duplicate, or disclosure by the U.S. Government is subjected to restrictions as set forth in FAR 12.212 (Commercial Computer Software) and DFARS 227.7202 (Commercial Computer Software and Commercial Computer Software Documentation), as applicable.

U.S. Patent 9,361,413

June 15, 2022

NA:V2022.2:Z:Z:Z:DC-VER-PDF

Documentation Feedback

At Hexagon Manufacturing Intelligence, we strive to produce the highest quality documentation and welcome your feedback. If you have comments or suggestions about our documentation, write to us at: documentation-feedback@mscsoftware.com.

Please include the following information with your feedback:

- Document name
- Release/Version number
- Chapter/Section name
- Topic title (for Online Help)
- Brief description of the content (for example, incomplete/incorrect information, grammatical errors, information that requires clarification or more details and so on).
- Your suggestions for correcting/improving documentation

You may also provide your feedback about Hexagon Manufacturing Intelligence documentation by taking a short 5-minute survey at: http://msc-documentation.questionpro.com.

Note:	The above mentioned e-mail address is only for providing documentation specific feedback. If you have any technical problems, issues, or queries, please contact Technical Support.

Contents

MSC Nastran Verification Guide

Preface 1

Section 1: MacNeal-Harder Problems

1.1 Straight Cantilever Beam 9
1.2 Curved Beam 13
1.3 Patch Test 16
1.4 Rectangular Plate 19
1.5 Scordelis-Lo-Roof 27
1.6 Spherical Shell 31
1.7 Thick Wall Cylinder 35
1.8 Twisted Beam 38

Section 2: Original Verification Manual Problems

2.1 Static Analysis of a Pin-Jointed Truss 43
2.2 Heat Transfer from Cooling Fin 46
2.3 Truss Analysis using Thermal Loads 49
2.4 Inertia Load on Pilot During Landing 52
2.5 Cantilever Beam, Statics, Checkpoint 55
2.6 Spring Supported Bar in a Skewed Coordinate System 58
2.7 Punched Output of a Static Solution 60
2.8 Cantilever Beam Constructed of Plates, Stress Contours 63
2.9 Bar with Offsets in a Skewed Coordinate System 66
2.10 Thick Walled Cylinder, Internal Pressure 69
2.11 Cantilevered Cylindrical Shell, Axisymmetric Force 72
2.12 Static Analysis of a Pin-Joined Truss Using Superelements 76
2.13 Beam with Point Masses, Enforced Acceleration 79
2.14 Two Degree of Freedom Modal Analysis 82

2.15 Normal Modes of a Beam Structure using Component Modal Synthesis 84
2.16 Torsional Vibration of a Shaft with Three Disks 88
2.17 Cantilever Beam, Normal Modes, Restart 90
2.18 Normal Modes Analysis with Reduction Techniques 93
2.19 Modes of Fixed Circular Plate with Trapezoidal Ring Element 96
2.20 Two Degree of Freedom Modal Analysis Using Superelements 99
2.21 Modes of Fixed Circular Plate with Superelement Method 101
2.22 Natural Frequencies of a Circular Plate Immersed in Fluid 104
2.23 DDAM Method of Response Spectra 107
2.24 Lateral Buckling of Cantilever Beam 111
2.25 Simple Frame Analysis with Buckling 114
2.26 Beam with Gap/Lift-Off 117
2.27 Cantilever Beam with Plastic Hinge 119
2.28 Complex Eigenvalues of Three-by-Three Matrix 121
2.29 Complex Roots as a Function of Gain 123
2.30 Frequency Response of a Single Degree of Freedom System 125
2.31 Transient Response of a Rocket - Direct Time Integration 128
2.32 Response Spectra of a Single Oscillator 131
2.33 Transient Response with Combined Loading Functions 133
2.34 Direct Transient Response to an Internal Pressure 135
2.35 Complex Eigenvalues of Three-by-Three Matrix, Modal Method 138
2.36 Modal Frequency Response of a Beam Structure 141
2.37 Transient Response of a Beam Modal Method 144
2.38 Deflection of a Circular Plate, Cyclic Symmetry 147
2.39 Normal Modes of a Circular Plate, Cyclic Symmetry 150
2.40 Euler Buckling of a Simply Supported Beam, Cyclic Symmetry 154
2.41 Frequency Response of a Beam-Mass Structure using Dihedral Cyclic Symmetry 157
2.42 Stress Wave Propagation 161
2.43 Impact Analysis 165
2.44 Flutter of a Two-Dimensional Airfoil 168
2.45 Radiation Exchange Between Two Panels 172
2.46 Transient Temperature Distribution in a Slab 174
2.47 Transient Temperature Distribution in a Slab with Convective Effects 177
2.48 Large Deflection of Uniformly Loaded Plate 180
2.49 Large Deflection of a Beam 183



Main Index

Section 3: Rotordynamics Problems

3.1 Flexible Rotor Critical Speed and Mode Shape 189
3.2 Jeffcott Rotor Eigenvalues 194
3.3 Complex Eigenvalue solution for Jeffcott rotor with multiple disks 198
3.4 Complex Eigenvalue Solution for Jeffcott Rotor with Overhung Disk and Anisotropic Bearings 202

Section 4: SOL 200 Optimization Problems

4.1 Optimization of a Three Bar Truss 209
4.2 Design with Constraint on Minimum Frequency 212
4.3 Shape Design of a Culvert 215
4.4 Topology Design of the MBB Beam 218

Section 5: SOL 700 Explicit Dynamics Problems

5.1 Impulsively Loaded Strip 223
5.2 Basic Blast Analysis 226
5.3 JWL Blast Analysis 229
5.4 Taylor Test (a rod impacted against a rigid wall) 232

Section 6: NAFEMS Problems

Linear Elastic

6.1 LE1: Elliptic Membrane 242
6.2 LE2: Cylindrical Shell Patch Test 245
6.3 LE3: Hemisphere-Point Loads 248
6.4 LE5: Z-section Cantilever 250
6.5 LE6: Skew Plate Normal Pressure 253
6.6 LE7: Axisymmetric Cylinder/Sphere-Pressure 255
6.7 LE8: Axisymmetric Shell Pressure 257
6.8 LE9: Axisymmetric Branched Shell-Pressure 259
6.9 LE10: Thick Plate Pressure 261
6.10 LE11: Solid Cylinder/ Taper/ Sphere-Temperature 264

Linear Thermo Elastic

6.11 T1: Membrane with Hot Spot 267

Heat Transfer

6.12 T2: One Dimensional Heat Transfer with Radiation 271
6.13 T3: One Dimensional Transient Heat Transfer 273
6.14 T4: Two Dimensional Heat Transfer with Convection 275

Free Vibration

6.15 FV2- Pin-Ended Double Cross: In-Plane Vibration 278
6.16 FV4 - Cantilever with Off-Centre Point Masses 281
6.17 FV5 - Deep Simply-supported Beam 283
6.18 FV12- Free Thin Square Plate 285
6.19 FV15- Clamped Thin Rhombic Plate 288
6.20 FV16- Cantilevered Thin Plate 291
6.21 FV22 - Clamped Thick Rhombic Plate 296
6.22 FV32 - Cantilevered Tapered Membrane 298
6.23 FV41 - Axisymmetric Vibration 301
6.24 FV42 - Thick Hollow Sphere 304
6.25 FV52 - Simply Supported Solid Square Plate 307
6.26 FV73 - Cantilevered Thin Square Plate 309

Forced Vibration

6.27 Test 5 - Deep Simply Supported Beam 313
6.28 Test 5H - Deep Simply Supported Beam Harmonic Forced Response 315
6.29 Test 5P - Deep Simply Supported Beam Periodic Forced Vibration Response 317
6.30 Test 5T-Deep Simply Supported Beam Transient Forced Vibration Response 320
6.31 Test 5R-Deep Simply Supported Beam Random Forced Response 323
6.32 Test 13: Simply Supported Thin Square Plate: Frequency Extraction 327
6.33 Test 13H: Simply Supported Thin Square Plate: Harmonic Forced Vibration 330
6.34 Test 13P: Simply Supported Thin Square Plate: Periodic Forced Vibration 334
6.35 Test 13T: Simply Supported Thin Square Plate: Transient Forced Vibration 338

6.36 Test 13R: Simply Supported Thin Square Plate: Random Forced Vibration 342
6.37 Test 21: Simply Supported Thick Square Plate: Frequency Extraction 345
6.38 Test 21H: Simply Supported Thick Square Plate: Harmonic Forced Vibration 348
6.39 Test 21P: Simply Supported Thick Square Plate: Periodic Forced Vibration 351
6.40 Test 21T: Simply Supported Thick Square Plate: Transient Forced Vibration 355
6.41 Test 21R: Simply Supported Thick Square Plate: Random Forced Vibration 360

Nonlinear

6.42 NL4: Snap-back Under Displacement Control 364
6.43 NL6: Straight Cantilever with Axial End Point Load 369
6.44 NL7: Lee's Frame Buckling Problem 372

Fracture Mechanics

6.45 Test 1.1: Center Cracked Plate in Tension 377
6.46 Test 1.2: Center Cracked Plate with Thermal Load 380
6.47 Test 2.1: Single Edge Cracked Plate in Tension 383
6.48 Test 3: Angle Crack Embedded in a Plate 386
6.49 Test 4: Cracks at a Hole in a Plate 389
6.50 Test 5: Axisymmetric Crack in a Bar 392
6.51 Test 6: Compact Tension Specimen 395
6.52 Test 7: T-Joint Weld Attachment 398
6.53 Test 8: V-Notch Specimen in Tension 401

Creep

6.54 Test 1A: 2-D Plane Stress - Uniaxial Load, Secondary Creep 405
6.55 Test 1B: 2-D Plane Stress - Uniaxial Displacement, Secondary Creep 409
6.56 Test 2A: 2-D Plane Stress - Biaxial Load, Secondary Creep 413
6.57 Test 2B: 2-D Plane Stress - Biaxial Displacement, Secondary Creep 416

6.58 Test 3B: 2-D Plane Stress - Biaxial (-ve) Displacement, Secondary Creep 420
6.59 Test 8A: 2D Plane Stress - Uniaxial Load, Primary Creep 424
6.60 Test 8B: 2-D Plane Stress - Uniaxial Displacement, Primary Creep 427
6.61 Test 8C: 2D Plane Stress - Stepped Load Primary Creep 430
6.62 Test 9A: 2-D Plane Stress - Biaxial Load, Primary Creep 433
6.63 Test 9B: 2-D Plane Stress - Biaxial Displacement, Primary Creep 436
6.64 Test 9C: 2-D Plane Stress - Biaxial Stepped Load, Primary Creep 439

Composite

6.65 R0031(1): Laminated Strip under Three-point Bending 443
6.66 R0031(2): Wrapped Thick Cylinder under Pressure and Thermal Loading 445
6.67 R0031 (3): Three-layer Sandwich Shell under Normal Pressure Loading 448

Contact

6.68 R0094 (1): Benchmark 1: 2D Cylinder Roller Contact 451
6.69 R0094 (2): Benchmark 2: 3D Punch (Rounded Edges) 454
6.70 R0094 (3): Benchmark 3: 3D Sheet Metal Forming 457
6.71 R0094 (4): Benchmark 4: 3D Loaded Pin 461
6.72 R0094 (5): Benchmark 5: 3D Steel Roller on Rubber 465

Geometric Nonlinear

6.73 3DNLG1: Elastic Large Deflection Response of A Z-Shaped Cantilever under End Load 470
6.74 3DNLG2: Elastic Large Deflection Response of a Pear-Shaped Cylinder under End Shortening 473
6.75 3DNLG6: Buckling of a Flat Plate When Subjected To In-Plane Shear 476
6.76 3DNLG7: Elastic Large Deflection Response of a Hinged Spherical Shell Under Pressure Loading 479
6.77 3DNLG9: Large Elastic Deflection of a Pinched Hemispherical Shell 482
6.78 3DNLG10: Elastic-Plastic Behavior of a Stiffened Cylindrical Panel under Compressive End Load 485

Preface

- About this Guide 2
- List of MSC Nastran Guides 3
- Technical Support 4
- Training and Internet Resources 4

About this Guide

The last update of the Verification Manual for MSC Nastran was in 1985. In the years since then, many features have been added to the program, but the manual has not been updated. This version is an initial update to bring the manual more up to date. The intent is for this to be a 'living document', with additional problems added with each new release of MSC Nastran.

This manual not only contains the original problems from the Verification Manual, but it also contains many additional problems.

The requirement for a problem to be included in this manual is that there must exist either a theoretical solution, a solution accepted in published literature, or a published set of test results. MSC tests over 29,000 problems in MSC Nastran daily during the development process and this manual contains a small subset of those problems.

A section named “MacNeal-Harder Problems” is added. This section contains the set of test problems proposed by Dr. Richard MacNeal and Bob Harder in 1985. These are problems intended to test how well the elements perform in varying situations which are known to be difficult for finite elements to handle properly.

Three sections on newer features in MSC Nastran are added. These are Rotordynamics, SOL 200 Optimization, and SOL 700 Explicit Dynamics. It turns out that it is difficult to find documented examples in literature with theoretical results or experimental results and a complete description of the structure/model used, so these sections each have a small set of problems. Many more examples can be found in the associated User Guides.

The final new section is called NAFEMS. This section contains many of the benchmark problems defined by NAFEMS (https://www.nafems.org/).

A standard format is followed for all problems:

- **Problem Description**: a short description of the problem being solved.
- **Solution Number**: the solution used in MSC Nastran to run the problem
- **Features Used**: a short list of features used in solving the problem
- **Reference**: the reference used as a baseline for the problem
- **Modeling Techniques Used**: a short description of the problem and properties/features used
- **Illustrations of the Model**: one or more illustrations of the structure and/or finite element model
- **Model File Name and Location**: the files are delivered with the documentation in a directory called 'tpl' (test problem library). This provides the name(s) of the file(s) used and the directory where they can be found.
- **Results**: a short presentation (usually in tabular form) of the results
- **Conclusion**: a summary of the findings

List of MSC Nastran Guides

A list of some of the MSC Nastran guides is as follows:

Installation and Release Guides
■ Installation and Operations Guide
■ Release Guide
Reference Guides
■ Quick Reference Guide
■ DMAP Programmer's Guide
■ Reference Guide
■ Utilities Guide
■ Getting Started Guide
■ SOL 400 Getting Started Guide
Demonstration Guides
■ Linear Analysis
■ Implicit Nonlinear (SOL 400)
■ Explicit Nonlinear (SOL 700)
■ MSC Nastran Verification Guide
User's Guides
■ Automated Component Modal Synthesis (ACMS)
■ Access Manual
■ Aeroelastic Analysis
■ Design Sensitivity and Optimization
■ DEMATD
■ Dynamic Analysis
■ Embedded Fatigue
■ Embedded Vibration Fatigue
■ Explicit Nonlinear (SOL 700)
■ High Performance Computing
■ Linear Static Analysis
■ Nonlinear (SOL 400)
■ Numerical Methods
■ Rotordynamics

- Superelements and Modules
- Thermal Analysis
- User Defined Services

You may find any of these documents from Hexagon at:

https://simcompanion.hexagon.com/customers/s/article/MSC-Nastran-Support-Home-Page

Technical Support

For technical support phone numbers and contact information, please visit:
https://simcompanion.hexagon.com/customers/s/article/support-contact-information-kb8019304

Support Center (http://simcompanion.hexagon.com)

The SimCompanion link above gives you access to the wealth of resources for Hexagon products. Here you will find product and support contact information, product documentations, knowledge base articles, product error list, knowledge base articles and SimAcademy Webinars. It is a searchable database which allows you to find articles relevant to your inquiry. Valid MSC customer entitlement and login is required to access the database and documents. It is a single sign-on that gives you access to product documentation for complete list of products from Hexagon, allows you to manage your support cases, and participate in our discussion forums.

Training and Internet Resources

Hexagon corporate site has information on the latest events, products and services for the CAD/CAE/CAM marketplace.

http://simcompanion.hexagon.com

The SimCompanion link above gives you access to the wealth of resources for Hexagon products. Here you will find product and support contact information, product documentations, knowledge base articles, product error list, knowledge base articles and SimAcademy Webinars. It is a searchable database which allows you to find articles relevant to your inquiry. Valid MSC customer entitlement and login is required to access the database and documents. It is a single sign-on that gives you access to product documentation for complete list of products from Hexagon, allows you to manage your support cases, and participate in our discussion forums.

http://www.mscsoftware.com/msc-training

The link above will point you to schedule and description of seminars. Following courses are recommended for beginning Nastran users.

NAS120 - Linear Static Analysis using MSC Nastran and Patran

This seminar introduces basic finite element analysis techniques for linear static, normal modes, and buckling analysis of structures using MSC Nastran and Patran. MSC Nastran data structure, the element library, modeling practices, model validation, and guidelines for efficient solutions are discussed and illustrated with examples and workshops. Patran will be an integral part of the examples and workshops and will be used to

generate and verify illustrative MSC Nastran models, manage analysis submission requests, and visualize results. This seminar provides the foundation required for intermediate and advanced MSC Nastran applications.

Section 1: MacNeal-Harder Problems



Main Index



Main Index

1.1 Straight Cantilever Beam

Problem Description

This problem demonstrates the use of different elements to model a cantilevered beam with a rectangular cross section subjected to unit forces at free end. The solution is compared with the analytical solution.

Solution Number

SOL 101

Features Used

- Linear Statics.

Reference

1. Macneal., R.H., Harder, R.L., 1985. A proposed standard set of problems to test finite element accuracy. Finite Elem. Anal. Des. 1, 3-20.

Modeling Techniques Used

The model is analyzed using different types of elements such as HEXA, PENTA, QUAD, QUADR and TRIA elements. The model details are given below:

Material Properties

The material is linear elastic with a Young's modulus, $E = 1.0 \times 10^7$ and Poisson's ratio, $\nu = 0.30$.

Geometrical properties

Length = 6.0

Width = 0.2

Depth = 0.1

Loading

A force of one unit applied at the free end.

Boundary Condition

The displacements $u_x = u_y = u_z = 0.0$ at one end of the beam as shown in Figure 1.1-1.

Illustrations of the Model

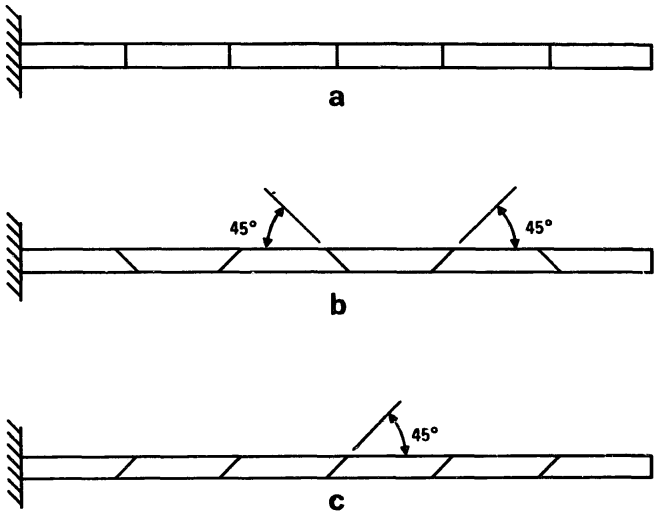


Figure 1.1-1 Straight cantilever beam. (a) Regular shape elements (b) Trapezoidal shape elements (c) Parallelogram shape elements

Model File Name and Location

Model File Name	Description	Location
macneal013	Straight cantilever beam, HEXA8 elements	tpl\verifman\macneal
macneal014	Straight cantilever beam, HEXA20 elements	tpl\verifman\macneal
macneal015	Straight cantilever beam, PENTA elements	tpl\verifman\macneal
macneal016	Straight cantilever beam, QUAD4 elements	tpl\verifman\macneal
macneal017	Straight cantilever beam, QUAD8 elements	tpl\verifman\macneal
macneal018	Straight cantilever beam, QUADR elements	tpl\verifman\macneal
macneal019	Straight cantilever beam, TRIA elements	tpl\verifman\macneal

Results

Rectangular Elements

Tip Loading Direction	Tip Displacement in Direction of Load							
	Theoretical [1]	HEXA8	HEXA20	PENTA	QUAD4	QUAD8	QUADR	TRIAR
Extension	3.0×10^{-5}	2.9×10^{-5}	3.0×10^{-5}	3.0×10^{-5}	3.0×10^{-5}	3.0×10^{-5}	3.0×10^{-5}	3.0×10^{-5}
In plane shear	0.1081	0.10599	0.10543	0.10542	0.10732	0.10667	0.10732	0.02559
Out-of-plane shear	0.4321	0.42374	0.41901	0.41678	0.42649	0.42837	0.42542	0.42114
Twist	0.03208	0.02917	0.02922	0.02963	0.03059	0.03046	0.03036	0.04238

Trapezoidal Elements

Tip Loading Direction	Tip Displacement in Direction of Load							
	Theoretical [1]	HEXA8	HEXA20	PENTA	QUAD4	QUAD8	QUADR	TRIAR
Extension	3.0×10^{-5}	2.9×10^{-5}	2.9×10^{-5}	2.9×10^{-5}	2.9×10^{-5}	2.9×10^{-5}	3.0×10^{-5}	2.9×10^{-5}
In plane shear	0.1081	0.00463	0.09701	0.04221	0.00569	0.10222	0.10449	0.02405
Out-of-plane shear	0.4321	0.12793	0.41534	0.26323	0.42389	0.43117	0.41735	0.41813
Twist	0.03208	0.02849	0.02930	0.28351	0.03085	0.03642	0.02951	0.04094

Parallelogram Elements

Tip Loading Direction	Tip Displacement in Direction of Load							
	Theoretical [1]	HEXA8	HEXA20	PENTA	QUAD4	QUAD8	QUADR	TRIAR
Extension	3.0×10^{-5}	2.9×10^{-5}	2.9×10^{-5}	2.9×10^{-5}	2.9×10^{-5}	2.9×10^{-5}	3.0×10^{-5}	2.9×10^{-5}
In plane shear	0.1081	0.02924	0.10575	0.08601	0.06834	0.10758	0.01046	0.03742
Out-of-plane shear	0.4321	0.24852	0.41492	0.37517	0.42874	0.42552	0.42163	0.42051
Twist	0.03208	0.026731	0.02923	0.028937	0.30652	0.03577	0.02820	0.043161

Conclusion

From the above observations, the rectangular element results from MSC Nastran are in better agreement with analytical results than trapezoidal and parallelogram elements.

1.2 Curved Beam

Problem Description

This problem demonstrates the use of different elements to model a 90-degree section of a circular beam. The beam is analyzed with two types of loading conditions:

a. loaded in its plane in a radial direction on its free end, and

b. loaded in out of plane direction on its free end.

The solution is compared to the analytical solution.

Solution Number

SOL 101

Features Used

- Linear Statics.

Reference

1. Macneal., R.H., Harder, R.L., 1985. A proposed standard set of problems to test finite element accuracy. Finite Elem. Anal. Des. 1, 3-20.

Modeling Techniques Used

The model is analyzed using different types of elements such as HEXA, PENTA, QUAD, QUADR and TRIAR elements. The model details are given below:

Material Properties

The material is linear elastic with a Young's modulus, $E = 1.0 \times 10^7$ and Poisson's ratio $\nu = 0.25$.

Geometrical properties

Inner radius = 4.12

Outer radius=4.32

Thickness=0.1

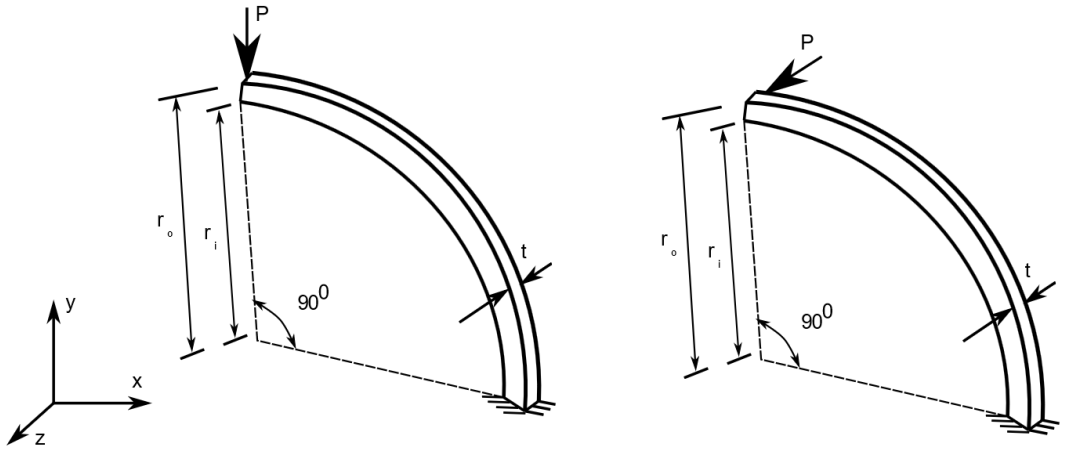
Loading

A force of one unit applied at the tip shown in Figure 1.2-1.

Boundary Condition

The displacements $u_x = u_y = u_z = 0.0$ at one end of the beam as shown in Figure 1.2-1.

Illustrations of the Model



a. Vertical Loading b. Out-of-plane Loading

Figure 1.2-1 Curved Beam

Model File Name and Location

Model File Name	Description	Location
macneal001	Curved beam, HEXA8 elements	tpl\verifman\macneal
macneal002	Curved beam, PENTA elements	tpl\verifman\macneal
macneal003	Curved beam, QUAD4 elements	tpl\verifman\macneal
macneal004	Curved beam, QUAD8 elements	tpl\verifman\macneal
macneal005	Curved beam, QUADR elements	tpl\verifman\macneal
macneal006	Curved beam, TRIAR elements	tpl\verifman\macneal

Results

Tip Loading Direction	Tip Displacement in Direction of Load						
	Theoretical [1]	HEXA8	PENTA	QUAD4	QUAD8	QUADR	TRIAR
In plane (vertical)	0.08734	0.07688	0.08191	0.077517	0.08797	0.08826	0.017425
Out-of-plane	0.5022	0.41940	0.46258	0.48018	0.47972	0.47069	0.500471

Conclusion

The results from MSC Nastran are in a good agreement with analytical results.

1.3 Patch Test

Problem Description

This problem contains basic patch test cases for Nastran elements. The solution is compared to the analytical solution.

Solution Number

SOL 101

Features Used

- Linear Statics.

Reference

1. Macneal., R.H., Harder, R.L., 1985. A proposed standard set of problems to test finite element accuracy. Finite Elem. Anal. Des. 1, 3-20.

Modeling Techniques Used

For the patch test, elements such as HEXA, QUAD, QUADR and TRIAR are used. The model details are given below:

Material Properties

The material is elastic with a Young's modulus, $E = 1.0 \times 10^7$ and Poisson's ratio $\nu = 0.25$.

Geometrical properties

Membrane Plate

thickness=0.001

length=0.24

width=0.12

Solid Cube

A cube of unit length is used as shown in Figure 1.3-1.

Boundary Condition

Displacement boundary conditions at all exterior nodes for constant stress loading are given below:

Membrane Plate

$u = 10^{-3} (x+y/2)$

$v = 10^{-3} (y+x/2)$

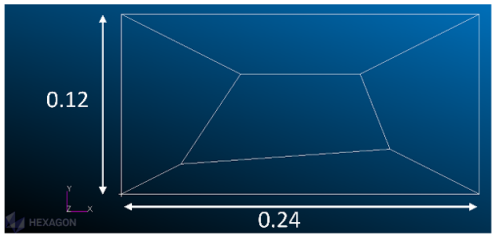
Solid Cube

$u = 10^{-3} (2x+y+z)/2$

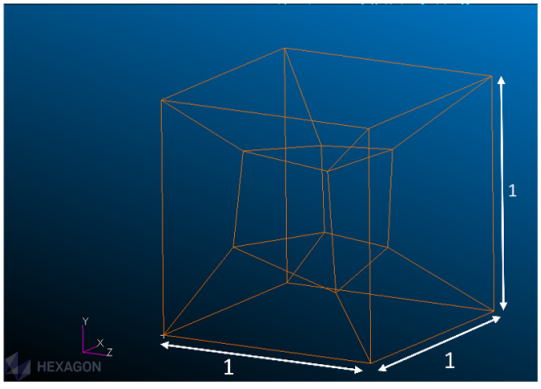
$v = 10^{-3} (x+2y+z)/2$

$w = 10^{-3} (x+y+2z)/2$

Illustrations of the Model



a. plates



b. solids

Figure 1.3-1 Patch Test Models

Model File Name and Location

Model File Name	Description	Location
macneal026	Patchtest, HEXA8 elements	tpl\verifman\macneal
macneal027	Patchtest, HEXA20 elements	tpl\verifman\macneal
macneal028	Patchtest, HEXA2R elements	tpl\verifman\macneal
macneal029	Patchtest, QUAD4 elements	tpl\verifman\macneal
macneal030	Patchtest, QUAD8 elements	tpl\verifman\macneal
macneal031	Patchtest, QUADR elements	tpl\verifman\macneal
macneal032	Patchtest, TRIAR elements	tpl\verifman\macneal

Results

Theoretical Results

Membrane Plate Patch Test

$\sigma_x = \sigma_y = 1333$, $\tau_{xy} = 400$

Solid Patch Test

$\sigma_x = \sigma_y = \sigma_z = 2000$, $\tau_{xy} = \tau_{yz} = \tau_{zx} = 400$

Maximum Error in Stress							
	HEXA8	HEXA20	HEXA20R	QUAD4	QUAD8	QUADR	TRIAR
Constant stress loading	0	0	0	0	15%	0	0

Conclusion

All elements show exact solutions except for the QUAD8 which differ from the analytical solution by about 15%.

1.4 Rectangular Plate

Problem Description

This problem demonstrates the use of different elements to model a rectangular plate. The plate is analyzed with two types of loading conditions:

a. uniform pressure, and

b. concentrated central load.

The solution is compared to the analytical solution.

Solution Number

SOL 101

Features Used

- Linear Statics.

Reference

1. Macneal., R.H., Harder, R.L., 1985. A proposed standard set of problems to test finite element accuracy. Finite Elem. Anal. Des. 1, 3-20.

Modeling Techniques Used

The model is analyzed using different types of elements such as HEXA, PENTA, QUAD, QUADR and TRIAR elements. The model details are given below:

Material Properties

The material is elastic with a Young's modulus, $E = 1.7472 \times 10^7$ and Poisson's ratio $\nu = 0.3$.

Geometrical properties

Two types of plate geometry are used for the analysis.

1. Square plate: a=2.0
2. Rectangular plate: a=2.0, b=10.0
3. Thickness= 0.0001(Shell element) and 0.01 (Solid element).

Loading

Two types of loadings are used for the analysis

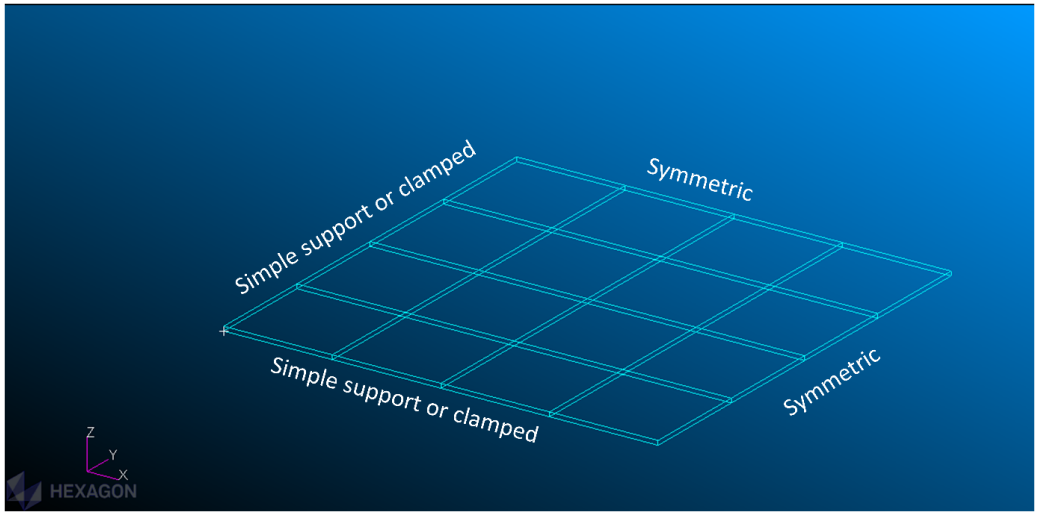
1. Uniform pressure= 10^{-4}
2. Central load = 4.0×10^{-4}

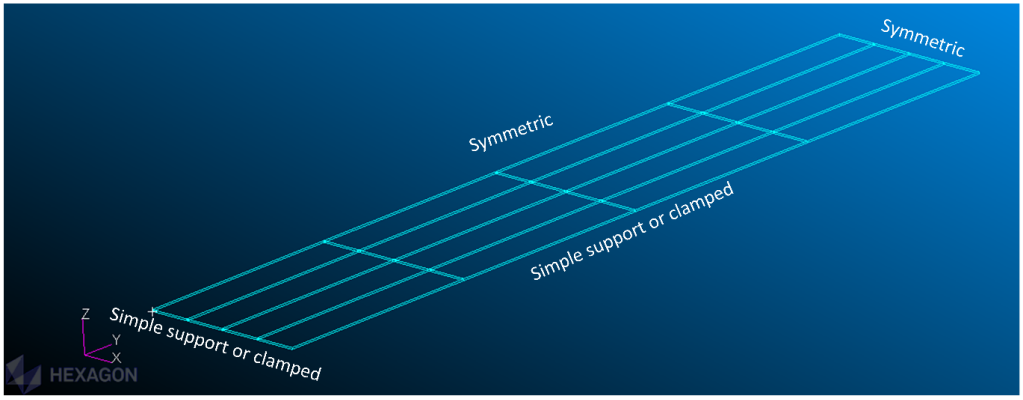
Boundary Condition

The model is analyzed using two types of boundary conditions

1. Simply supported
2. Clamped

Illustrations of the Model





(b)

Figure 1.4-1 (a) Plate with Aspect Ratio 1.0 (b) Plate with Aspect Ratio 5.0

Model File Name and Location

Model File Name	Description	Location
macneal033	Rectangular Plate, HEX8 element, AR*=1, N*=2	tpl\verifman\macneal
macneal034	Rectangular Plate, HEX8 element, AR=1, N=4	tpl\verifman\macneal
macneal035	Rectangular Plate, HEX8 element, AR=1, N=6	tpl\verifman\macneal
macneal036	Rectangular Plate, HEX8 element, AR=1, N=8	tpl\verifman\macneal
macneal037	Rectangular Plate, HEX8 element, AR=5, N=2	tpl\verifman\macneal
macneal038	Rectangular Plate, HEX8 element, AR=5, N=4	tpl\verifman\macneal
macneal039	Rectangular Plate, HEX8 element, AR=5, N=6	tpl\verifman\macneal
macneal040	Rectangular Plate, HEX8 element, AR=5, N=8	tpl\verifman\macneal
macneal041	Rectangular Plate, PENTA element, AR=1, N=2	tpl\verifman\macneal
macneal042	Rectangular Plate, PENTA element, AR=1, N=4	tpl\verifman\macneal
macneal043	Rectangular Plate, PENTA element, AR=1, N=6	tpl\verifman\macneal
macneal044	Rectangular Plate, PENTA element, AR=1, N=8	tpl\verifman\macneal
macneal045	Rectangular Plate, PENTA element, AR=5, N=2	tpl\verifman\macneal
macneal046	Rectangular Plate, PENTA element, AR=5, N=4	tpl\verifman\macneal
macneal047	Rectangular Plate, PENTA element, AR=5, N=6	tpl\verifman\macneal
macneal048	Rectangular Plate, PENTA element, AR=5, N=8	tpl\verifman\macneal
macneal049	Rectangular Plate, QUAD4 element, AR=1, N=2	tpl\verifman\macneal
macneal050	Rectangular Plate, QUAD4 element, AR=1, N=4	tpl\verifman\macneal
macneal051	Rectangular Plate, QUAD4 element, AR=1, N=6	tpl\verifman\macneal
macneal052	Rectangular Plate, QUAD4 element, AR=1, N=8	tpl\verifman\macneal
macneal053	Rectangular Plate, QUAD4 element, AR=5, N=2	tpl\verifman\macneal
macneal054	Rectangular Plate, QUAD4 element, AR=5, N=4	tpl\verifman\macneal
macneal055	Rectangular Plate, QUAD4 element, AR=5, N=6	tpl\verifman\macneal
macneal056	Rectangular Plate, QUAD4 element, AR=5, N=8	tpl\verifman\macneal
macneal057	Rectangular Plate, QUAD8 element, AR=1, N=2	tpl\verifman\macneal
macneal058	Rectangular Plate, QUAD8 element, AR=1, N=4	tpl\verifman\macneal
macneal059	Rectangular Plate, QUAD8 element, AR=1, N=6	tpl\verifman\macneal
macneal060	Rectangular Plate, QUAD8 element, AR=1, N=8	tpl\verifman\macneal

Model File Name	Description	Location
macneal061	Rectangular Plate, QUAD8 element, AR=5, N=2	tpl\verifman\macneal
macneal062	Rectangular Plate, QUAD8 element, AR=5, N=4	tpl\verifman\macneal
macneal063	Rectangular Plate, QUAD8 element, AR=5, N=6	tpl\verifman\macneal
macneal064	Rectangular Plate, QUAD8 element, AR=5, N=8	tpl\verifman\macneal
macneal065	Rectangular Plate, QUADR element, AR=1, N=2	tpl\verifman\macneal
macneal066	Rectangular Plate, QUADR element, AR=1, N=4	tpl\verifman\macneal
macneal067	Rectangular Plate, QUADR element, AR=1, N=6	tpl\verifman\macneal
macneal068	Rectangular Plate, QUADR element, AR=1, N=8	tpl\verifman\macneal
macneal069	Rectangular Plate, QUADR element, AR=5, N=2	tpl\verifman\macneal
macneal070	Rectangular Plate, QUADR element, AR=5, N=4	tpl\verifman\macneal
macneal071	Rectangular Plate, QUADR element, AR=5, N=6	tpl\verifman\macneal
macneal072	Rectangular Plate, QUADR element, AR=5, N=8	tpl\verifman\macneal
macneal073	Rectangular Plate, TRIAR element, AR=1, N=2	tpl\verifman\macneal
macneal074	Rectangular Plate, TRIAR element, AR=1, N=4	tpl\verifman\macneal
macneal075	Rectangular Plate, TRIAR element, AR=1, N=6	tpl\verifman\macneal
macneal076	Rectangular Plate, TRIAR element, AR=1, N=8	tpl\verifman\macneal
macneal077	Rectangular Plate, TRIAR element, AR=5, N=2	tpl\verifman\macneal
macneal078	Rectangular Plate, TRIAR element, AR=5, N=4	tpl\verifman\macneal
macneal079	Rectangular Plate, TRIAR element, AR=5, N=6	tpl\verifman\macneal
macneal080	Rectangular Plate, TRIAR element, AR=5, N=8	tpl\verifman\macneal

*AR = aspect ratio, N= Number of node spaces per edge of model.

Results

1. Simply Supported and Uniform Load

a. Aspect Ratio 1.0

Lateral Deflection at the Center							
N	Theoretical [1]	HEXA8	PENTA	QUAD4	QUAD8	QUADR	TRIAR
2	4.062(shell)	4.01E-06	2.200 E-06	4.146	3.764	3.986	3.860
4	4.062E-06 (solid)	4.05E-06	3.203 E-06	4.117	4.064	4.078	4.037
6		4.05E-06	3.625 E-06	4.093	4.059	4.076	4.062
8		4.06E-06	3.828 E-06	4.081	4.061	4.071	4.068

b. Aspect Ratio 5.0

Lateral Deflection at the Center							
N	Theoretical [1]	HEXA8	PENTA	QUAD4	QUAD8	QUADR	TRIAR
2	12.97(shell)	12.3E-06	12.70E-06	13.04	15.86	13.64	12.873
4	12.97E-06 (solid)	12.6E-06	11.96 E-06	12.84	13.01	12.85	13.039
6		12.8E-06	13.43 E-06	12.91	12.97	12.92	13.018
8		12.9E-06	13.78 E-06	12.94	12.97	12.94	13.004

2. Simply Supported and Concentrated Load

a. Aspect Ratio 1.0

Lateral Deflection at the Center							
N	Theoretical [1]	HEXA8	PENTA	QUAD4	QUAD8	QUADR	TRIAR
2	11.60(shell)	11.68E-06	7.375 E-06	12.35	10.98	11.80	10.59
4	11.60E-06 (solid)	11.59E-06	8.171 E-06	11.94	11.50	11.78	11.29
6		11.59E-06	9.614 E-06	11.79	11.56	11.71	11.46
8		11.59E-06	10.39 E-06	11.72	11.58	11.68	11.52

b. Aspect Ratio 5.0

Lateral Deflection at the Center							
N	Theoretical [1]	HEXA8	PENTA	QUAD4	QUAD8	QUADR	TRIAR
2	16.96(shell)	15.31E-06	12.12 E-06	16.25	15.86	13.75	12.35
4	16.96 E-06 (solid)	16.04E-06	6.338 E-06	16.65	13.01	15.80	16.00
6		16.49E-06	7.634 E-06	16.97	12.97	16.50	16.47
8		16.67E-06	8.601 E-06	17.07	12.97	16.77	16.65

3. Clamped Supports and Uniform Load

a. Aspect Ratio 1.0

Lateral Deflection at the Center							
N	Theoretical [1]	HEXA8	PENTA	QUAD4	QUAD8	QUADR	TRIAR
2	1.26 (shell)	1.239E-06	2.07 E-08	1.360	1.458	1.269	1.203
4	1.26 E-06 (solid)	1.258 E-06	1.74 E-07	1.323	1.225	1.300	1.255
6		1.262 E-06	6.88 E-07	1.298	1.258	1.288	1.262
8		1.263E-06	9.69 E-07	1.286	1.263	1.280	1.263

b. Aspect Ratio 5.0

Lateral Deflection at the Center							
N	Theoretical [1]	HEXA8	PENTA	QUAD4	QUAD8	QUADR	TRIAR
2	2.56 (shell)	2.897E-06	7.64 E-08	3.406	3.866	3.364	2.695
4	2.56 E-06 (solid)	2.501E-06	7.26 E-07	2.653	3.112	2.600	2.708
6		2.532E-06	1.97 E-06	2.604	2.466	2.603	2.662
8		2.563E-06	2.46 E-06	2.603	2.648	2.603	2.639

4. Clamped Supports and Concentrated Load

a. Aspect Ratio 1.0

Lateral Deflection at the Center							
N	Theoretical [1]	HEXA8	PENTA	QUAD4	QUAD8	QUADR	TRIAR
2	5.60 (shell)	4.960 E-06	8.76E-08	5.612	6.030	5.234	4.354
4	5.60E-06 (solid)	5.447 E-06	5.74 E-07	5.793	5.429	5.662	5.199
6		5.538 E-06	2.69 E-06	5.740	5.560	5.674	5.407
8		5.571 E-06	4.04 E-06	5.701	5.588	5.662	5.488

b. Aspect Ratio 5.0

Lateral Deflection at the Center							
N	Theoretical [1]	HEXA8	PENTA	QUAD4	QUAD8	QUADR	TRIAR
2	7.23(shell)	2.322E-06	7.77 E-08	3.823	3.918	3.753	3.236
4	7.23E-06 (solid)	6.145E-06	6.30 E-07	6.692	5.451	6.241	5.658
6		6.705E-06	1.73 E-06	7.105	6.741	6.798	6.405
8		6.920E-06	2.42 E-06	7.242	7.046	7.031	6.708

Conclusion

The results from MSC Nastran are in a good agreement with analytical results.

1.5 Scordelis-Lo-Roof

Problem Description

This problem demonstrates the deformation of thin curved membrane loaded under its own weight. The computed maximum Z displacement is compared to the theoretical solution.

Solution Number

SOL 101

Features Used

- Linear Statics.

Reference

1. Macneal., R.H., Harder, R.L., 1985. A proposed standard set of problems to test finite element accuracy. Finite Elem. Anal. Des. 1, 3-20.

Modeling Techniques Used

The model is analyzed using different types of elements such as HEXA, PENTA, QUAD, QUADR and TRIAR elements. The model details are given below:

Material Properties

The material is linear elastic with a Young's modulus, $E = 1.0 \times 10^7$ and Poisson's ratio, $\nu = 0.25$.

Geometrical properties

Radius = 25.0

length=50.

Thickness=0.25

Angle = 40°

Loading

A force of 90.0 per unit area in the - Z direction is applied on the surface.

Boundary Condition

The displacements boundary conditions are applied as shown in Figure 1.5-1.

Illustrations of the Model

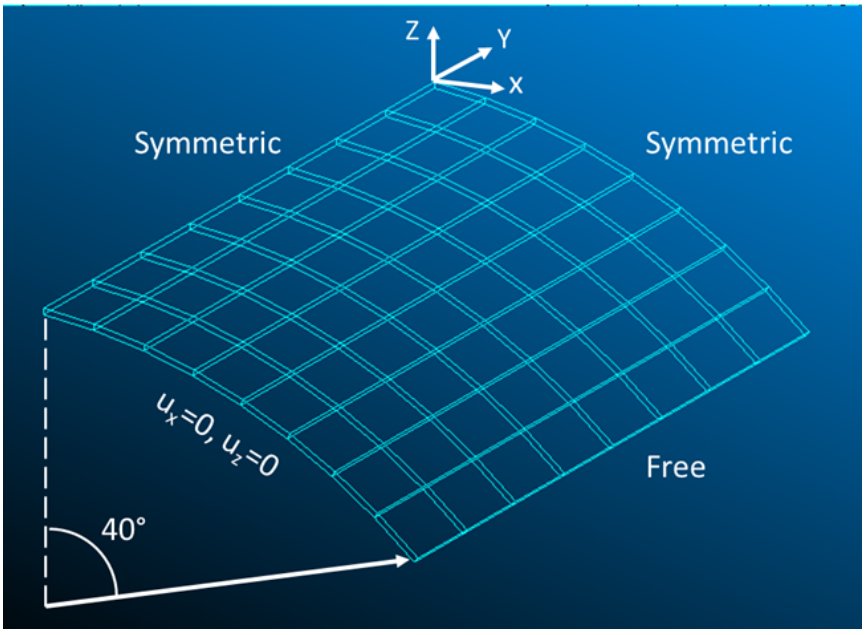


Figure 1.5-1 Scordelis-Lo-Roof

Model File Name and Location

Model File Name	Description	Location
macneal081	Scordolis-lo-roof, HEXA8 element, N*=2	tpl\verifman\macneal
macneal082	Scordolis-lo-roof, HEXA8 element, N=4	tpl\verifman\macneal
macneal083	Scordolis-lo-roof, HEXA8 element, N=6	tpl\verifman\macneal
macneal084	Scordolis-lo-roof, HEXA8 element, N=8	tpl\verifman\macneal
macneal085	Scordolis-lo-roof, PENTA element, N=2	tpl\verifman\macneal
macneal086	Scordolis-lo-roof, PENTA element, N=4	tpl\verifman\macneal
macneal087	Scordolis-lo-roof, PENTA element, N=6	tpl\verifman\macneal
macneal088	Scordolis-lo-roof, PENTA element, N=8	tpl\verifman\macneal
macneal089	Scordolis-lo-roof, QUAD4 element, N=2	tpl\verifman\macneal
macneal090	Scordolis-lo-roof, QUAD4 element, N=4	tpl\verifman\macneal
macneal081	Scordolis-lo-roof, QUAD4 element, N=6	tpl\verifman\macneal
macneal082	Scordolis-lo-roof, QUAD4 element, N=8	tpl\verifman\macneal
macneal083	Scordolis-lo-roof, QUAD4 element, N=2	tpl\verifman\macneal
macneal084	Scordolis-lo-roof, QUAD8 element, N=4	tpl\verifman\macneal
macneal085	Scordolis-lo-roof, QUAD8 element, N=6	tpl\verifman\macneal
macneal086	Scordolis-lo-roof, QUAD8 element, N=8	tpl\verifman\macneal
macneal087	Scordolis-lo-roof, QUAD8 element, N=2	tpl\verifman\macneal
macneal088	Scordolis-lo-roof, QUAD8 element, N=4	tpl\verifman\macneal
macneal089	Scordolis-lo-roof, QUADR element, N=6	tpl\verifman\macneal
macneal100	Scordolis-lo-roof, QUADR element, N=8	tpl\verifman\macneal
macneal101	Scordolis-lo-roof, QUADR element, N=2	tpl\verifman\macneal
macneal102	Scordolis-lo-roof, QUADR element, N=4	tpl\verifman\macneal
macneal103	Scordolis-lo-roof, QUADR element, N=6	tpl\verifman\macneal
macneal104	Scordolis-lo-roof, TRIA element, N=8	tpl\verifman\macneal
macneal105	Scordolis-lo-roof, TRIA element, N=2	tpl\verifman\macneal
macneal106	Scordolis-lo-roof, TRIA element, N=4	tpl\verifman\macneal
macneal107	Scordolis-lo-roof, TRIA element, N=6	tpl\verifman\macneal
macneal108	Scordolis-lo-roof, TRIA element, N=8	tpl\verifman\macneal

* where N: Number of node spaces per edge of mode.

Results

Lateral Deflection at the Center							
N*		HEXA8	PENTA	QUAD4	QUAD8	QUADR	TRIAR
2	0.3024	0.4556	0.0443	0.4868	0.3707	0.4730	0.3029
4		0.3550	0.1082	0.3658	0.3413	0.3643	0.2995
6		0.3497	0.2000	0.3534	0.3474	0.3538	0.3229
8		0.3470	0.2735	0.3496	0.3463	0.3504	0.3329
10				0.3478	0.3454	0.3487	0.3377

* where N: Number of node spaces per edge of model

Conclusion

Except for PENTA elements, all other element results from MSC Nastran are in good agreement with the theoretical solution.

1.6 Spherical Shell

Problem Description

This problem demonstrates the use of different elements to model a thin hemispherical shell subjected to concentrated loads. The solution is compared to the theoretical solution [1].

Solution Number

SOL 101

Features Used

- Linear Statics.

Reference

1. Macneal., R.H., Harder, R.L., 1985. A proposed standard set of problems to test finite element accuracy. Finite Elem. Anal. Des. 1, 3-20.

Modeling Techniques Used

The model is analyzed using different types of elements such as HEXA, PENTA, QUAD, QUADR and TRIAR elements. The model details are given below:

Material Properties

The material is linear elastic with a Young's modulus, $E = 6.825 \times 10^7$ and a Poisson's ratio, $\nu = 0.3$.

Geometrical properties

Radius = 10.0

Thickness = 0.04

Loading

1.0-unit load is applied at quadrant.

Boundary Condition

The displacements boundary conditions are applied as shown in Figure 1.6-1.

Illustrations of the Model

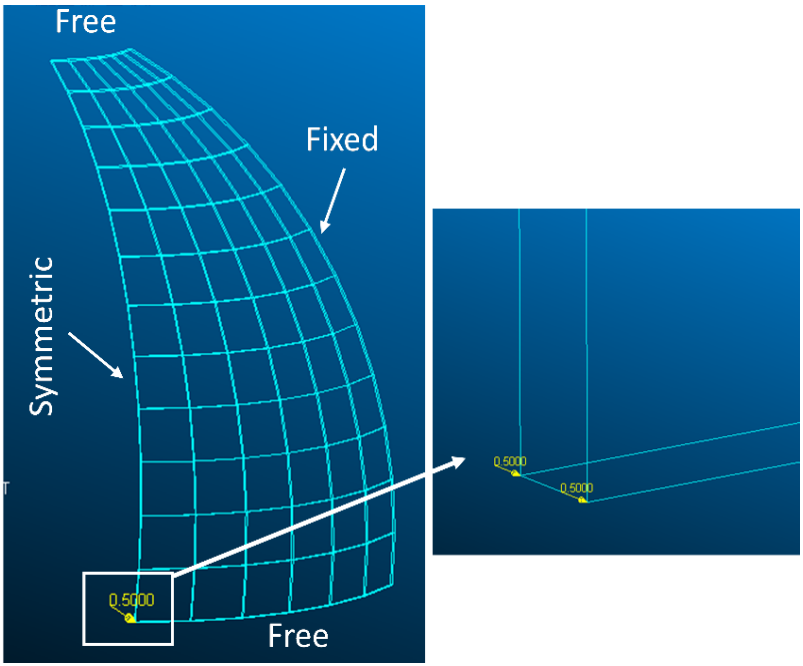


Figure 1.6-1 Spherical shell modeled with eight hexahedral elements

Model File Name and Location

Model File Name	Description	Location
macneal135	Spherical Shell, HEXA8 elements, N*=4	tpl\verifman\macneal
macneal136	Spherical Shell, HEXA8 elements, N=8	tpl\verifman\macneal
macneal137	Spherical Shell, HEXA8 elements, N=12	tpl\verifman\macneal
macneal138	Spherical Shell, PENTA elements, N=4	tpl\verifman\macneal
macneal109	Spherical Shell, PENTA elements, N=8	tpl\verifman\macneal
macneal110	Spherical Shell, PENTA elements, N=12	tpl\verifman\macneal
macneal111	Spherical Shell, QUAD4 elements, N=2	tpl\verifman\macneal
macneal112	Spherical Shell, QUAD4 elements, N=4	tpl\verifman\macneal
macneal113	Spherical Shell, QUAD4 elements, N=6	tpl\verifman\macneal
macneal114	Spherical Shell, QUAD4 elements, N=8	tpl\verifman\macneal
macneal115	Spherical Shell, QUAD4 elements, N=10	tpl\verifman\macneal
macneal116	Spherical Shell, QUAD4 elements, N=12	tpl\verifman\macneal
macneal117	Spherical Shell, QUAD8 elements, N=2	tpl\verifman\macneal
macneal118	Spherical Shell, QUAD8 elements, N=4	tpl\verifman\macneal
macneal119	Spherical Shell, QUAD8 elements, N=6	tpl\verifman\macneal
macneal120	Spherical Shell, QUAD8 elements, N=8	tpl\verifman\macneal
macneal121	Spherical Shell, QUAD8 elements, N=10	tpl\verifman\macneal
macneal122	Spherical Shell, QUAD8 elements, N=12	tpl\verifman\macneal
macneal123	Spherical Shell, QUADR elements, N=2	tpl\verifman\macneal
macneal124	Spherical Shell, QUADR elements, N=4	tpl\verifman\macneal
macneal125	Spherical Shell, QUADR elements, N=6	tpl\verifman\macneal
macneal126	Spherical Shell, QUADR elements, N=8	tpl\verifman\macneal
macneal127	Spherical Shell, QUADR elements, N=10	tpl\verifman\macneal
macneal128	Spherical Shell, QUADR elements, N=12	tpl\verifman\macneal
macneal129	Spherical Shell, TRIAR elements, N=2	tpl\verifman\macneal
macneal130	Spherical Shell, TRIAR elements, N=4	tpl\verifman\macneal

Model File Name	Description	Location
macneal131	Spherical Shell, TRIAR elements, N=6	tpl\verifman\macneal
macneal132	Spherical Shell, TRIAR elements, N=8	tpl\verifman\macneal
macneal133	Spherical Shell, TRIAR elements, N=10	tpl\verifman\macneal
macneal134	Spherical Shell, TRIAR elements, N=12	tpl\verifman\macneal

* where N: Number of node spaces per edge of model

Results

Lateral Deflection at the Center							
N		HEXA8	PENTA	QUAD4	QUAD8	QUADR	TRIAR
2	0.0940			0.09373	0.00230	0.03863	0.07960
4		0.00384	0.00070	0.09716	0.01139	0.04019	0.10591
6				0.09581	0.04644	0.06845	0.09543
8		0.06978	0.00379	0.09492	0.07739	0.08401	0.10850
10				0.09441	0.08975	0.08943	0.10482
12		0.08994	0.01446	0.09413	0.09323	0.09140	0.10966

Conclusion

Except for PENTA and CTRIAR elements, the results from MSC Nastran are in a good agreement with theoretical results.

1.7 Thick Wall Cylinder

Problem Description

This problem demonstrates the use of different elements to model a thick-walled cylinder subjected to internal pressure. Nearly incompressible material is used. The solution is compared to the analytical solution.

Solution Number

SOL 101

Features Used

- Linear Statics.

Reference

1. Macneal., R.H., Harder, R.L., 1985. A proposed standard set of problems to test finite element accuracy. Finite Elem. Anal. Des. 1, 3-20.

Modeling Techniques Used

The model is analyzed using different types of elements such as HEXA, PENTA, QUAD, QUADR and TRIAR elements. The model details are given below:

Material Properties

The material is linear elastic with a Young's modulus, $E = 1.0 \times 10^7$ and Poisson's ratio $\nu = 0.49$, 0.499 and 0.4999.

Geometrical properties

Inner radius = 3.0

Outer radius=9.0

Thickness=1

Loading

A unit pressure is applied at inner radius as shown in Figure 1.7-1.

Boundary Condition

$u_y = u_z = R_x = R_y = R_z = 0.0$ for all the nodes.

Illustrations of the Model

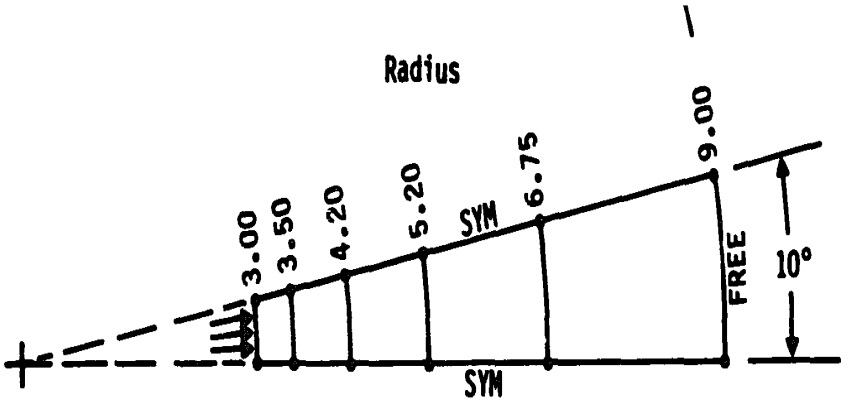


Figure 1.7-1 Thick-walled Cylinder

Model File Name and Location

Model File Name	Description	Location
macneal020	Thick Cylinder, HEXA8 elements	tpl\verifman\macneal
macneal021	Thick Cylinder, PENTA elements	tpl\verifman\macneal
macneal022	Thick Cylinder, QUAD4 elements	tpl\verifman\macneal
macneal023	Thick Cylinder, QUAD8 elements	tpl\verifman\macneal
macneal024	Thick Cylinder, QUADR elements	tpl\verifman\macneal
macneal025	Thick Cylinder, TRIAR elements	tpl\verifman\macneal

Results

Radial displace at inside radius							
Poisson's ratio	Theoretical [1]	HEXA8	PENTA	QUAD4	QUAD8	QUADR	TRIAR
0.49	0.00504	0.00497	0.00503	0.00497	0.00503	0.00496	0.00419
0.499	0.00506	0.00498	0.00504	0.00498	0.00504	0.00498	0.00404
0.4999	0.00506	0.00499	0.00504	0.00499	0.00489	0.00498	0.00402

Conclusion

Except CTRIAR elements, the results from MSC Nastran are in a good agreement with analytical results.

1.8 Twisted Beam

Problem Description

This problem demonstrates the use of different elements to model a twisted beam with a rectangular cross section subjected to unit forces at free end. The solution is compared with the analytical solution.

Solution Number

SOL 101

Features Used

- Linear Statics.

Reference

1. Macneal., R.H., Harder, R.L., 1985. A proposed standard set of problems to test finite element accuracy. Finite Elem. Anal. Des. 1, 3-20.

Modeling Techniques Used

The model is analyzed using different types of elements such as HEXA, PENTA, QUAD, QUADR and TRIAR elements. The model details are given below:

Material Properties

The material is linear elastic with a Young's modulus, $E = 29.0 \times 10^6$ and Poisson's ratio $\nu = 0.3$.

Geometrical properties

Length = 12.0

Width = 1.1

Depth = 0.32

Twist= 90° (root to tip)

Loading

A force of one unit applied at the free end.

Boundary Condition

The displacements $u_x = u_y = u_z = 0.0$ at one end of the beam as shown in Figure 1.8-1.

Illustrations of the Model

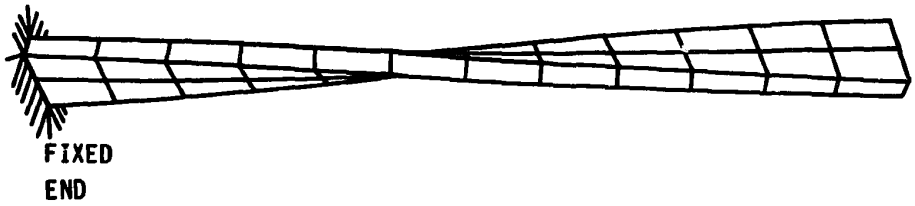


Figure 1.8-1 Twisted Beam

Model File Name and Location

Model File Name	Description	Location
macneal007	Twisted Beam, HEXA8 elements	tpl\verifman\macneal
macneal008	Twisted Beam, PENTA elements	tpl\verifman\macneal
macneal009	Twisted Beam, QUAD4 elements	tpl\verifman\macneal
macneal010	Twisted Beam, QUAD8 elements	tpl\verifman\macneal
macneal011	Twisted Beam, QUADR elements	tpl\verifman\macneal
macneal012	Twisted Beam, TRIA elements	tpl\verifman\macneal

Results

Tip loading direction	Radial displace at inside radius						
	Theoretical [1]	HEXA8	PENTA	QUAD4	QUAD8	QUADR	TRIAR
In plane (vertical)	0.005424	0.00539	0.00537	0.00539	0.00541	0.00538	0.00531
Out-of-plane	0.001754	0.00173	0.00174	0.00173	0.00175	0.00173	0.00164

Conclusion

The results from MSC Nastran are in a good agreement with analytical results.

Section 2: Original Verification Manual Problems



Main Index

2.1 Static Analysis of a Pin-Jointed Truss

Problem Description

A truss consisting of two pin-jointed axial elements is loaded vertically. Find the vertical deflection of the end and the internal forces and stresses in the elements.

Solution Number

SOL 101

Features Used

- Static Analysis

Reference

1. Egor P. Popov, Introduction to Mechanics of Solids (Englewood Cliffs: Prentice-Hall, Inc. 1968), P. 499

Modeling Techniques Used

- The pin jointed truss is modeled using rods (CONROD). The model details are given below:

  Engineering Data

E	=	3.0E+7 lb/in^2
A_1	=	0.15 in^2
A_2	=	0.25 in^2
I	=	60. in^4
P	=	500. lb

Illustrations of the Model

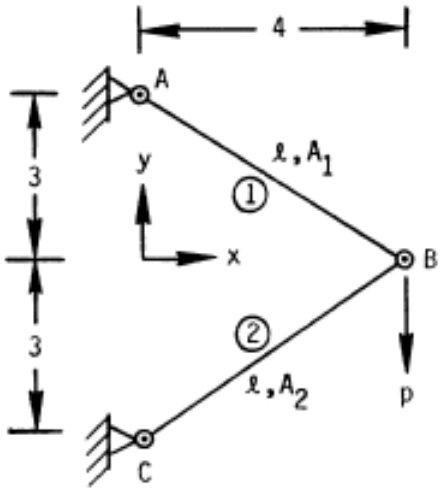


Figure 2.1-1 Pin Jointed Truss

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10101.dat	v2401.dat	Static analysis of pin jointed truss.	tpl\verifman\oldverif\

Results

Theoretical Solution

At point B:	$U_y = -0.0444$ in
Force in Element 1:	$P_1 = +2500.$ lb (tension)
Force in Element 2:	$P_2 = -2500.$ lb (compression)
Stress in Element 1:	$\sigma_1 = P_1/A_1 = +16667.$ lb/in^2
Stress in Element 2:	$\sigma_2 = P_2/A_2 = -10000.$ lb/in^2

MSC Nastran Solution

At point B:	$U_y = -0.0444$ in
Force in Rod 1:	+2500. lb (tension)
Force in Element 2:	-2500. lb (compression)
Stress in Element 1:	+16667. lb/in^2
Stress in Element 2:	-10000. lb/in^2

Comparison of Results

	Solution Summary			
			Element Stress (lb/in^2)	
	Displacement (in)	Element Force (lb)	Rod 1	Rod 2
Theory	-0.0444	±2500.	+16667.	-10000.
MSC Nastran	-0.0444	±2500.	+16667.	-10000.
% Difference	0%	0%	0%	0%

Conclusion

The results from MSC Nastran are comparable with theoretical results.

2.2 Heat Transfer from Cooling Fin

Problem Description

A fin of circular cross section is maintained at 250° at one end. The fin is extended into the air. The air temperature is 70°. Assuming a material conductivity of 132 Btu/hr-ft-°F. Determine the steady-state temperature distribution along the rod.

Solution Number

SOL 400

Features Used

- Steady heat transfer

Reference

1. Alan J. Chapman, Heat transfer, 3rd ed. (New York: MacMillan Publishing Co., Inc., 1974) P. 76, Ex. 3.8

Modeling Techniques Used

- The cooling fin is modeled using CBEAM element. The model details are given below:

  Engineering Data

L	=	1 ft
D	=	0.04167 ft
A	=	0.001365 ft^2
$I_{11}=I_{22}$	=	1.48E-7 ft^4
K	=	132 Btu/hr-ft-°F
H	=	1.6 Btu/hr-ft-°F
T_{base}	=	250°F
T_{fluid}	=	70°F

Illustrations of the Model

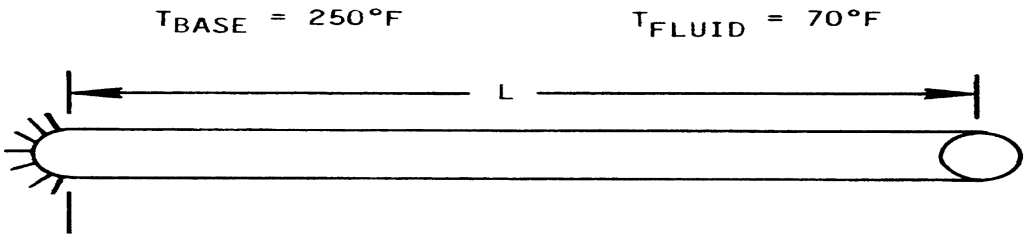


Figure 2.2-1 Cooling Fin

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v40001.dat	v2402.dat	Heat transfer from cooling fin	tpl\verifman\oldverif\

Results

Theoretical Solution

$$T = (T_B - T_F)\left(\frac{\cosh(mL - x)}{\cosh mL}\right) + T_F$$

where $m = \sqrt{\frac{4h}{kd}}$ (for cylindrical rod)

$$m = \sqrt{\frac{4 \times 1.6}{0.63 \times \frac{1}{2 \times 12}}}$$

$$m = 1.079\frac{1}{ft}$$

X/L	T (^{0}F)
0.25	217.0243
0.50	196.0559
0.75	183.7156
1.00	179.7002

Result comparison of theoretical solution and MSC Nastran results are as follows:

	T (^{0}F)			
X/L	0.250	0.500	0.750	1.000
Theory	217.62	196.06	183.72	179.7
MSC Nastran	217.8	196.338	184.054	180.056
Difference	0.083%	0.142%	0.182%	0.198%

Conclusion

The results from MSC Nastran are comparable with theoretical results.

2.3 Truss Analysis using Thermal Loads

Problem Description

A pin joined truss is loaded with a force at one end of the component elements is heated. Considering thermal effects, find the displacements of the truss joints and the forces and stresses in the axial elements.

Solution Number

SOL 101

Features Used

- Static analysis with thermal loads

Reference

1. J. S. Przemieniecki, Theory of Matrix Structural Analysis (New York: McGraw- Hill Book Company, 1968), P. 155

Modeling Techniques Used

- The Pin joined truss assembly is modeled using CONROD elements. The details of the model is as follows.

  Engineering Data

E	=	1.0E+7 lb/in^2
α	=	1.0E-6/^{0}F
A_1	=	1.0 in^2 (Elements 1, 3, 5, 6)
A_2	=	0.7071068 in^2 (Elements 2 and 4)
ΔT	=	+100.^{0}F (Elements 3 only)
F	=	1000. lb
I	=	20. in^4

Illustrations of the Model

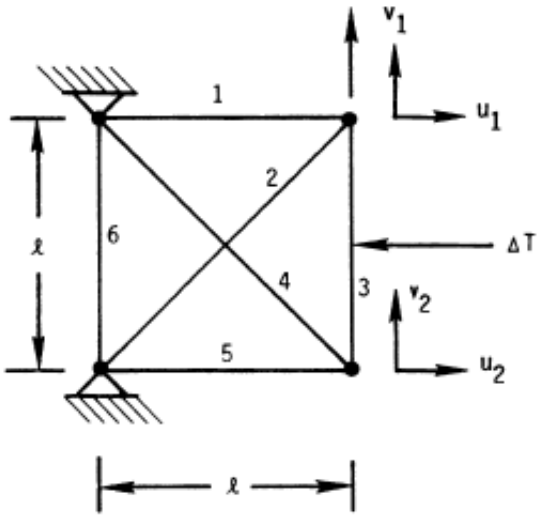


Figure 2.3-1 Truss with thermal loads

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10103.dat	v2403.dat	Truss analysis with thermal loads	tpl\verifman\oldverif\

Results

Displacement (in)				
	u1	v1	u2	v2
Theory	-1.272727E-03	6.363636E-03	7.272727E-04	3.636364E-03
MSC Nastran	-1.272727E-03	6.363636E-03	7.272727E-04	3.636364E-03
Difference	0.00%	0.00%	0.00%	0.00%

Forces (lb)						
	p1	p2	p3	p4	p5	p6
Theory	-636.36	899.95	363.64	-514.26	363.64	0.00
MSC Nastran	-636.36	899.95	363.64	-514.26	363.64	0.00
Difference	0.00%	0.00%	0.00%	0.00%	0.00%	0.00%

Stresses (lb/in^2)						
	σ_1	σ_2	σ_3	σ_4	σ_5	σ_6
Theory	-636.36	1272.73	363.64	-727.27	363.64	0.00
MSC Nastran	-636.36	1272.73	363.64	-727.27	363.64	0.00
Difference	0.00%	0.00%	0.00%	0.00%	0.00%	0.00%

Conclusion

The results from MSC Nastran are comparable with theoretical results.

2.4 Inertia Load on Pilot During Landing

Problem Description

Assuming an aircraft is making a level landing with the nose gear slightly off the ground, find the resulting inertia loads on the pilot due to the main gear landing loads.

Solution Number

SOL 101

Features Used

- Inertia Relief Analysis

Reference

1. E. F. Bruhn, Analysis and Design of flight Vehicle Structures (Cincinnati: Tri- state Offset Company, 1965) P. A4.11

Modeling Techniques Used

- The aircraft and pilot assembly is modeled using beam (CBEAM) and concentrated mass element (CONM2). The model details are given below:

  Engineering Data

  Aircraft: Weight = 100,000 lb, pitching mass moment of inertia = 40,000,000 lb.sec^{2}in

  Pilot: Weight = 180.0 lb

  Forces (at main landing gear): P_z = 300,000 lb (Vertical), P_x = 100,000 lb (aft)

Illustrations of the Model

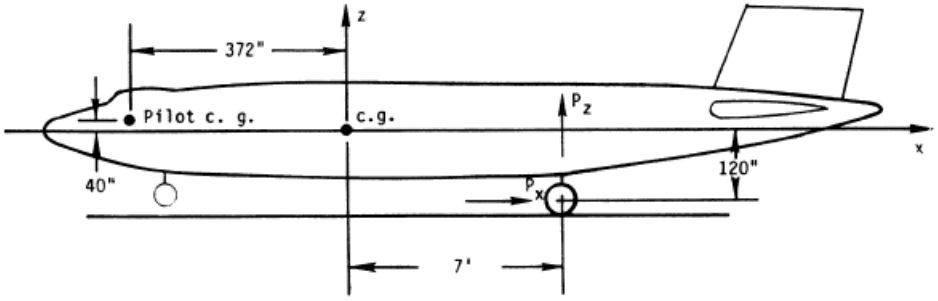


Figure 2.4-1 Aircraft with Pilot

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10104.dat	v2404.dat	Inertia load on Pilot during Landing	tpl\verifman\oldverif\

Results

Theoretical Solution

Loads on Pilot: F_x = 162.7 lb F_z = 378.84 lb

MSC Nastran Solution

The user must put all mass items into consistent units.

Aircraft

M_a = 100,000 lb ($\frac{1}{g}$ = 0.002588 sec^2/in)

M_a = (258.8 lb-sec^2/in)

Pilot

M_p = 180 lb ($\frac{1}{g}$ = 0.002588 sec^2/in)

M_p = 0.4658 (lb-sec^2/in)

Also the aircraft must be adjusted for the pilot.

$$\sum M_a = 258.8 - 0.4658$$

$$\sum M_a = 258.3342 \quad (\text{lb-sec}^2/\text{in})$$

The MSC Nastran results are:

F_x = 162.62 lb

F_z = 378.46 lb

Result comparison of theoretical solution and MSC Nastran results are as follows:

	Load on Pilot (lb)	
	F_x	F_z
Theory	162.7	378.84
MSC Nastran	162.6	378.46
Difference	-0.06%	-0.10%

Conclusion

The results from MSC Nastran are comparable with theoretical results.

2.5 Cantilever Beam, Statics, Checkpoint

Problem Description

A simple cantilever beam is loaded with a vertical shear load at the free end. The solution results are saved (checkpoint) for a restart in a modal solution (v10305r).

Solution Number

SOL 101

Features Used

- Static Analysis

Reference

1. R.J. Roark and W. C. Young, Formulas for stress and strain (New York: McGraw-Hill Book Co. 1975) P. 96

Modeling Techniques Used

- The cantilever beam is modeled using beam (CBEAM) element. The model details are given below:

Engineering Data

E	=	1.0E+7 lb/in^2
ν	=	0.3
l	=	30.0 in
h	=	1.0 in
w	=	0.75 in
t	=	0.10 in
A	=	0.310 in^2
I_1	=	0.0390 in^4
I_2	=	0.0241 in^4
J	=	0.0631 in^4

Mass	=	1.0 lb-sec^2/in
η	=	1.0 in/sec^2

Illustrations of the Model

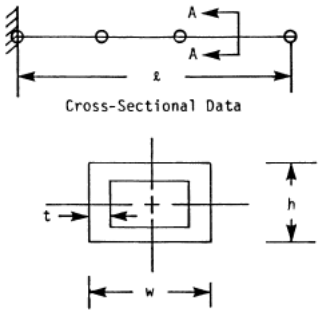


Figure 2.5-1 Cantilever Beam

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10105d.dat	v2405n.dat	Cantilever beam, statics, checkpoint	tpl\verifman\oldverif\
v10305r.dat	v0305r.dat		tpl\verifman\oldverif\

Results

Theoretical Solution

1. Displacement at free end:

$$\delta = \frac{Fl^3}{3EI}$$

$$\delta = \frac{100(30)^3}{3(1.0\times10^7)0.0390}$$

$$\delta = 2.3077 \text{ in}$$

2. Moment at fixed end:

M = FL

M = 100(30)

M = 3000 lb-in

3. Stress at extreme fibre locations:

$$\sigma = \frac{MC}{I}$$

$$\delta = \frac{3000(\pm 0.50)}{(0.0390)}$$

$$\delta = \pm 38462 \text{ lb/in}^2$$

Result comparison of theoretical solution and MSC Nastran results are as follows:

	Displacement at free end (in)	Moment at Fixed end (lb-in)	Maximum Stress (lb/in^2)
	δ	M	σ_{max}
Theory	2.307700	3000.00	38462.00
MSC Nastran	2.307692	3000.00	38462.00
Difference	0.00%	0.00%	0.00%

Conclusion

The results from MSC Nastran are comparable with theoretical results.

2.6 Spring Supported Bar in a Skewed Coordinate System

Problem Description

A coordinate system exists that is rotated 45° with respect to a basic frame of reference in the x-y plane. A bar located in this system is simply supported by elastic supports. A load is applied to the bar at an intermediate point. Calculate the reaction forces in the supports.

Solution Number

SOL 101

Features Used

- Static Analysis

Reference

1. Joseph Shigley, Mechanical Engineering Design (New York: McGraw-Hill Inc., 1977) P. 642

Modeling Techniques Used

- The spring supported bar is modeled using bar element (CBAR) and spring element (CELAS2). The model details are given below:

  Engineering Data

θ	=	45^0
a	=	1 in
b	=	2 in
k	=	1.0E+4 lb/in
F	=	600.0 lb
E	=	3.0E+7 psi
A	=	0.0625 in^2
I	=	3.255E-4 in^4

Illustrations of the Model

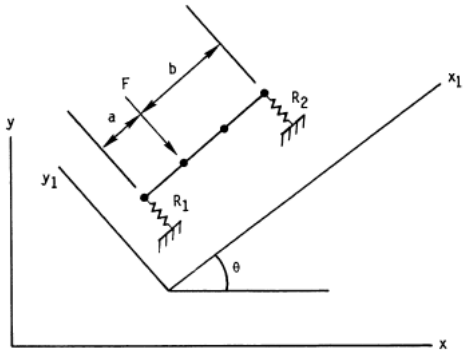


Figure 2.6-1 Spring supported bar

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10106.dat	V2406.dat	Spring supported bar in skewed system	tpl\verifman\oldverif\

Results

Result comparison of theoretical solution and MSC Nastran results are as follows:

	Reaction Forces / Spring Forces (lb)	
	R_1	R_2
Theory	400	200
MSC Nastran	400	200
Difference	0.00%	0.00%

Conclusion

The results from MSC Nastran are comparable with theoretical results.

2.7 Punched Output of a Static Solution

Problem Description

A simple structurally intermediate problem (see schematic below) is analyzed using MSC NASTRAN with the resulting displacement and stress output punched.

Solution Number

SOL 101

Features Used

- Static Analysis

Modeling Techniques Used

- The model details are given below:

Engineering Data

E	=	1.0E+7 lb/in^2
ν	=	0.3
l	=	1.0 in
M	=	1000 in-lb
A	=	1.0 in^2
$I_1=I_2$	=	1.0E-4 in^4

Illustrations of the Model

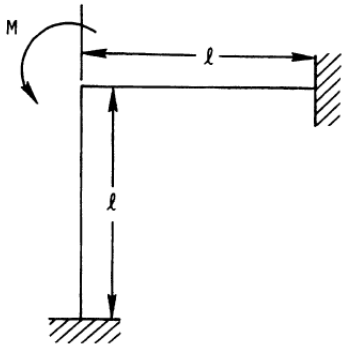


Figure 2.7-1 Spring supported bar

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10107.dat	V2407.dat	Punched output of static solution	tpl\verifman\oldverif\

Results

Punched output of static solution by MSC NASTRAN is as follows.

```
$TITLE   =SIMPLE STRUCTURAL INDETERMINATE PROBLEM 1                V10          1
$SUBTITLE=DEMONSTRATE PUNCHED OUTPUT                                            2
$LABEL   =OUTPUT DISPLACEMENTS AND ELEMENT STRESSES                             3
$DISPLACEMENTS                                                                  4
$REAL OUTPUT                                                                    5
$SUBCASE ID =           1                                                       6
        3       G     -7.491909E-05     -7.491909E-05      0.000000E+00       7
-CONT-                 0.000000E+00      0.000000E+00       1.254046E-01       8
$TITLE   =SIMPLE STRUCTURAL INDETERMINATE PROBLEM 1                V10          9
$SUBTITLE=DEMONSTRATE PUNCHED OUTPUT                                           10
$LABEL   =OUTPUT DISPLACEMENTS AND ELEMENT STRESSES                            11
$ELEMENT STRESSES                                                              12
$REAL OUTPUT                                                                   13
$SUBCASE ID =           1                                                      14
$ELEMENT TYPE =           2  BEAM                                              15
        1                1                0.000000E+00     -2.499401E+05      16
-CONT-                -2.499401E+05     -2.499401E+05     -2.499401E+05       17
-CONT-                -2.499401E+05     -2.499401E+05         1               18
-CONT-                   1                 0              0.000000E+00       19
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       20
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       21
-CONT-                 0.000000E+00      0.000000E+00         0               22
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       23
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       24
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       25
-CONT-                    0              0.000000E+00      0.000000E+00       26
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       27
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       28
-CONT-                 0.000000E+00         0              0.000000E+00       29
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       30
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       31
-CONT-                 0.000000E+00      0.000000E+00         0               32
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       33
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       34
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       35
-CONT-                    0              0.000000E+00      0.000000E+00       36
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       37
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       38
-CONT-                 0.000000E+00         0              0.000000E+00       39
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       40
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       41
-CONT-                 0.000000E+00      0.000000E+00         0               42
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       43
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       44
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       45
-CONT-                    0              0.000000E+00      0.000000E+00       46
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       47
-CONT-                 0.000000E+00      0.000000E+00      0.000000E+00       48
-CONT-                 0.000000E+00         3              1.000000E+00       49
-CONT-                 4.992508E+05      4.992508E+05      4.992508E+05       50
-CONT-                 4.992508E+05      4.992508E+05      4.992508E+05       51
-CONT-                    1                 1                               52
```

Main Index

2.8 Cantilever Beam Constructed of Plates, Stress Contours

Problem Description

A Simple cantilever beam is constructed using plate element and is loaded with vertical shear force at one end. Find the stress at the outside fibers.

Solution Number

SOL 101

Features Used

- Static analysis

Reference

1. Ferdinand L. Singer, Strength of Materials (Harper & Row, 1962) Art. 52, pp. 133.

Modeling Techniques Used

- The cantilever beam is modelled using CQUAD4 element. The model details are given below:

Engineering Data

E	=	10.7E+06 lb/in^2
ν	=	0.3
t	=	0.1 in
l	=	3.0 in
h	=	0.6 in
p	=	120.0 lb

Illustrations of the Model

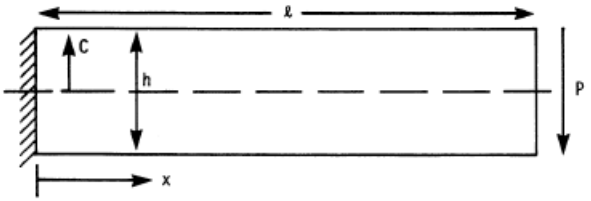


Figure 2.8-1 Cantilever Beam

Model File Name and Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10108.dat	V2408a.dat	Cantilever beam constructed of plates, stress contours	tpl\verifman\oldverif

Results

Theoretical Solution

Moment of Inertia:

$$I = \frac{th^3}{12} = \frac{0.10(0.6)^3}{12} = 1.8\text{E-3 in}^4$$

Bending Moment at fixed end:

$$M = 3.0p = 3.0(120) = 360.0 \text{ in-lb}$$

Bending Stress:

$$S = \frac{MC}{I} = \frac{360.0C}{1.8\text{E-3}} \text{ lb/in}^2$$

C = Fibre distance from neutral axis

MSC Nastran Solution

	Stresses (psi)			
Fibre Distance	**C = 0.0**	**C = 0.10**	**C = 0.20**	**C = 0.30**
Theory	0	20000	40000	60000
MSC Nastran	0	19400	39350	54444
%Difference	0.00%	-3.00%	-1.63%	-9.26%

Conclusion

The results from MSC Nastran are comparable to the theoretical results.

2.9 Bar with Offsets in a Skewed Coordinate System

Problem Description

A coordinate system exists that is rotated 45° with respect to a basic frame of reference in the x-y plane. A bar located in this system is simply supported by elastic supports. The grid points are located along the x1 axis and the bar at an intermediate point. Calculate the reaction forces in the supports.

Solution Number

SOL 101

Features Used

- Static Analysis

Reference

1. Joseph Shigley, Mechanical Engineering Design (New York: McGraw-Hill Inc., 1977) P. 642

Modeling Techniques Used

- The spring bar system is modeled using CELAS and CBAR element, respectively. The model details are given below:

  Engineering Data

θ	=	45^0
a	=	1 in
b	=	2 in
k	=	1.0E+4 lb/in
F	=	600.0 lb
E	=	3.0E+7 psi
A	=	0.0625 in^2
I	=	3.255E-4 in^4
H	=	2 in

Illustrations of the Model

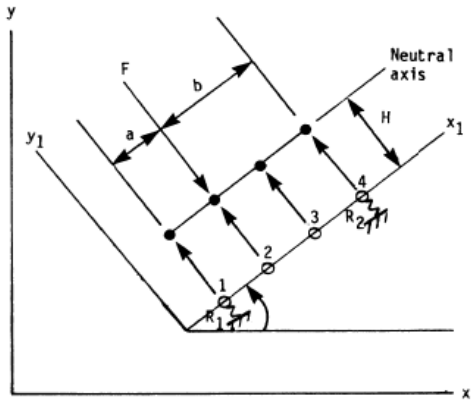


Figure 2.9-1 Bar with Spring system

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
`v10109.dat`	`V2409.dat`	Bar with offsets in skewed coordinate system	tpl\verifman\oldverif\

Results

Theoretical Solution

$$R_1 = \frac{Fb}{l} = \frac{600(2)}{3} = 400.0 \text{ lb}$$

$$R_2 = \frac{Fa}{l} = \frac{600(1)}{3} = 200.0 \text{ lb}$$

Result comparison of theoretical solution and MSC Nastran results are as follows:

	Reaction Forces / Spring Forces (lb)	
	R_1	R_2
Theory	400	200
MSC Nastran	400	200
Difference	0.00%	0.00%

Conclusion

The results from MSC Nastran are comparable with theoretical results.

2.10 Thick Walled Cylinder, Internal Pressure

Problem Description

Find the stress distribution for a thick-walled cylinder subjected to internal pressure.

Solution Number

SOL 101

Features Used

- Static Analysis

Reference

1. S. H. Crandall, N. C. Dahl, and T. J. Lardner, An Introduction to the Mechanics of Solids, 2nd ed. (New York: McGraw-Hill Book Company, 1972) P. 293-297

Modeling Techniques Used

- Thick Cylinder is modeled using CTRIAX6 element. The model details are given below:

Engineering Data

r_i	=	6.0 in
r_o	=	12.0 in
h	=	8.0 in
E	=	30.0E+6 psi
ρ	=	0.28 lb/in^3
ν	=	0.0
p	=	10.0 lb/in^2

Illustrations of the Model

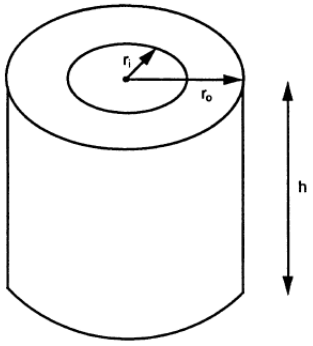


Figure 2.10-1 Thick walled Cylinder

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10110.dat	V2410.dat	Thick walled cylinder, internal pressure	tpl\verifman\oldverif\

Results

Theoretical Solution

$$\sigma_r = \frac{-P_i[(r_o/r)^2 - 1]}{(r_o/r_i)^2 - 1}$$

$$\sigma_\theta = \frac{P_i[(r_o/r)^2 + 1]}{(r_o/r_i)^2 - 1}$$

Result comparison of theoretical solution and MSC Nastran results are as follows:

		σ_r lb/in^2			σ_θ lb/in^2		
r	r/r_i	Theory	MSC Nastran	% Error	Theory	MSC Nastran	% Error
6.0	1	-10.00			16.67		
6.50	1.083	-8.030	-8.095	0.811%	14.69	14.852	1.104%
7.0	1.167	-6.460			13.13		
7.50	1.25	-5.200	-5.238	0.731%	11.87	11.956	0.722%

		σ_r lb/in^2			σ_θ lb/in^2		
r	r/r_i	Theory	MSC Nastran	% Error	Theory	MSC Nastran	% Error
8.0	1.333	-4.170			10.83		
8.50	1.417	-3.310	-3.333	0.704%	9.98	10.031	0.509%
9.0	1.5	-2.590			9.26		
9.50	1.583	-1.990	-2.000	0.502%	8.65	8.686	0.421%
10.0	1.667	-1.470			8.13		
10.50	1.75	-1.020	-1.030	1.009%	7.69	7.710	0.262%
11.0	1.833	-0.630			7.3		
11.50	1.917	-0.300	-0.303	1.009%	6.96	6.979	0.271%
12.0	2	0.00			6.67		

Conclusion

MSC Nastran results are comparable with theoretical results.

2.11 Cantilevered Cylindrical Shell, Axisymmetric Force

Problem Description

Find the radial deflection and meridional moments in a cantilevered cylindrical shell.

Solution Number

SOL 101

Features Used

- Static Analysis

Reference

1. "Analysis of Axisymmetrical shells by the Direct Stiffness Method" P. E. Grafton and D. R. Strome, AIAA Journal, 1(10): 2342-2347
2. "Evaluation of NASTRAN" J. W. Jones and H. H. Fong, Structural Mechanics Software Series, Vol. IV (N. Perrone and W. Pilkey, eds.), 1982

Modeling Techniques Used

Engineering Data

E	=	1.0E+7 psi
ν	=	0.3

Moment of inertia per unit width: $I = \frac{Bh^3}{12} = \frac{1(0.01)^3}{12} = 8.33\times10^{-8}$

Illustrations of the Model

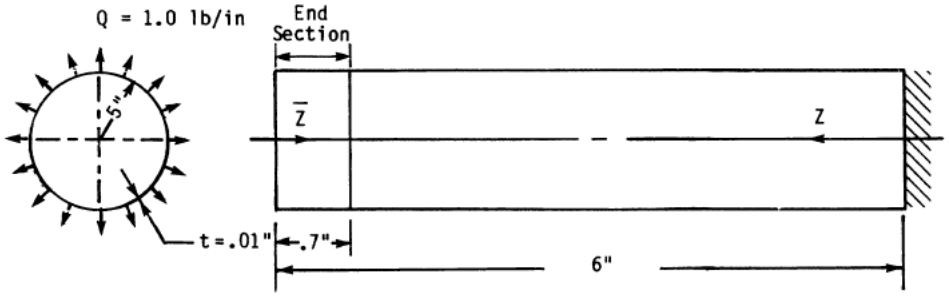


Figure 2.11-1 Cantilevered Cylinder shell

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10111.dat	V2411.dat	Cantilevered Cylinder shell	tpl\verifman\oldverif\

Results

Theoretical Solution

Radial deflection: $U_r = \frac{Q}{2\beta^3 D} e^{-\beta X} \cos \beta X$

Meridional moment: $M = \frac{Q}{\beta} e^{-\beta X} \sin \beta X$

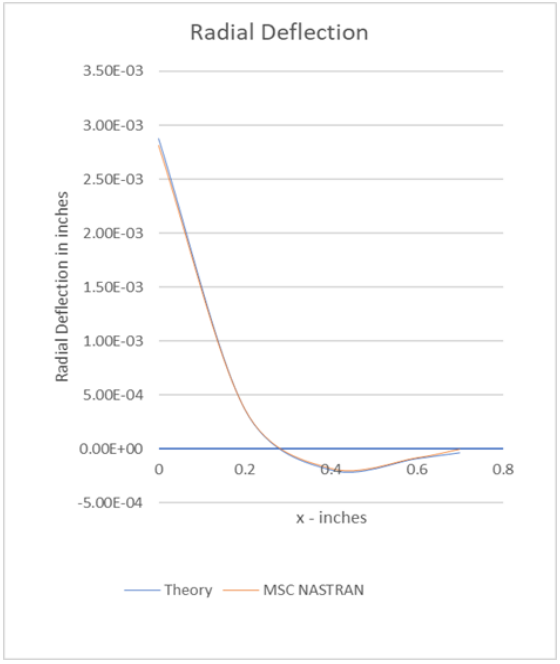
where,

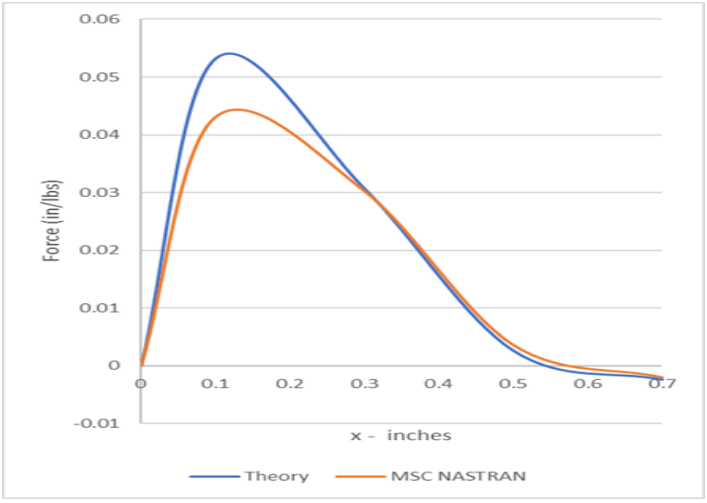
$\beta = 1.285 / \sqrt{Rt} = 5.746695 \text{ in}^{-1}$

Result comparison of theoretical solution and MSC Nastran results are as follows:

Z (in)	Radial Deflection (10E-3 in)		
	Theory	MSC Nastran	% Error
0	2.8769	2.814938	-2.15%
0.2	0.37291	0.3712448	-0.45%
0.4	-0.19215	-0.1812774	-5.66%
0.6	-0.08725	-0.08327	-4.56%

	Bend moment (in/lbs)	
Z (in)	Theory	MSC Nastran
0.1	5.32E-02	4.31E-02
0.3	3.07E-02	3.02E-02
0.5	2.61E-03	3.55E-03
0.7	-2.40E-03	-2.06E-03





Conclusion

MSC Nastran results are comparable with theoretical results.

2.12 Static Analysis of a Pin-Joined Truss Using Superelements

Problem Description

A truss consisting of two pin-jointed axial elements is loaded vertically. Find the vertical deflection of the end and the internal forces.

Solution Number

SOL 101

Features Used

- Static Analysis using Superelements

Reference

1. Egor P. Popov, Introduction to Mechanics of Solids (Englewood Cliffs: Prentice-Hall, Inc.. 1968) P. 499.

Modeling Techniques Used

- The conventional statics deck v10101 is converted by adding the SEALL, SUPER and SESET cards. Their function is described by comments in the data deck. The model details are given below:

  Engineering Data

E	=	3.0E+7 lb/in^2
A_1	=	0.15 in^2
A_2	=	0.25 in^2
l	=	60.0 in
p	=	3000 lb

Illustrations of the Model

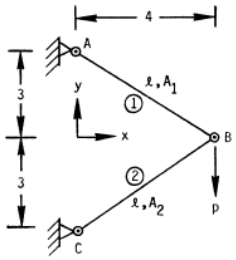


Figure 2.12-1 Pin Joined Truss

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10112s.dat	V6101s.dat	Static analysis of pin joined truss	tpl\verifman\oldverif

Results

Theoretical Solution

At point B:	U_y= –0.0444 in
Force in Element 1:	+2500. lb (tension)
Force in Element 2:	-2500. lb (compression)

MSC Nastran Solution

At point B:	U_y= –0.0444 in
Force in Rod 1:	+2500. lb
Force in Rod 1:	-2500. lb

Comparison of Results

	Displacement (in)	Element Force (lb)
Theory	-0.444	±2500.
MSC Nastran	-0.444	±2500.
% Difference	0%	0%

Conclusion

MSC Nastran results are comparable with theoretical results.

2.13 Beam with Point Masses, Enforced Acceleration

Problem Description

A beam, with equal point masses on each end, is given a unit acceleration on its center point normal to its axis. Find the reaction force, Q, required to produce this acceleration, and the resulting deflections relative to the point of application of the force.

Solution Number

SOL 101

Features Used

- Superelement Statics with Inertial relief

Reference

1. Raymond J. Roark, Formulas for Stress and Strain, 2nd ed. (New York: mcGraaw-Hill Book Company, 1965), P. 104.

Modeling Techniques Used

1. The BAR elements are given section properties for A, I_1 and J, as well as I_2, the section property in the loaded direction, so that the model is fully defined in three dimensions.
2. The offsets of the masses provide moments of inertia of the total structure for all directions except R_1. A nominal I_1 is therefore added to again give full three-dimensional capability to the model.
3. The enforced acceleration is input on the DMIG cards named UACCEL. The column numbers start with 1, 0, and would be incremented to 2, 0; 3, 0;... if there were more loading conditions.
4. The loading condition is applied to the same grid point used as a reference point on the DMIG, SUPORT, and PARAM, GRDPNT cards.

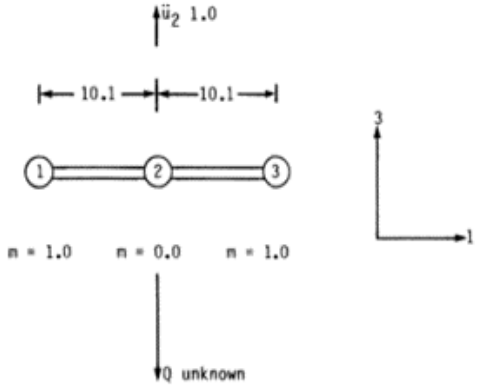
- The model details are given below:

Engineering Data

E	=	1.0E+07 lb/in^2
l	=	10.0 in
$I_1 = I_2 = I_3$	=	3.0 in^4

Main Index

Illustrations of the Model



Model File Name and Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10113s.dat	V9101s.dat	Beam with point masses, enforced acceleration	tpl\verifman\oldverif

Results

Theoretical Solution

From symmetry arguments, the loading condition is equivalent to a point load on a cantilevered beam of half the total length of this beam. The reference gives the load-deflection equation:

$$u_3 = \frac{p_3 l^3}{3\mathrm{EI}}$$

Where P_3 is the applied load equal to the unit tip mass times the unit acceleration, or unity.

Therefore,

$$u_3 = \frac{10^3}{3\times10^7 \times 3} = 1.111\ldots \times 10^{-5}$$

The reaction force is twice the applied load for one side, therefore,

$$Q = -2.0$$

MSC Nastran Results

The reaction force, defined as the force applied by the structure on the supported point, appears in the SPCFORCES. It agrees with the theoretical value to all places printed. The displacements also agree with the theoretical values to all places printed.

Conclusion

The results from MSC Nastran are comparable to the theoretical results.

2.14 Two Degree of Freedom Modal Analysis

Problem Description

A simple frictionless two degree of freedom system is constructed using springs and concentrated masses. Find all natural frequencies and mode shapes.

Solution Number

SOL 103

Features Used

- Normal Modes Analysis
- Inverse Power Method and Lanczos Method

Reference

1. Donald. T. Greenwood, Principles of Dynamic (Englewood Cliffs: Prentice-Hall, Inc., 1965) p.459, ex. 9-1.

Modeling Techniques Used

- The spring and concentrated mass assembly are modeled using rods (CONROD) and concentrated masses (CONM2) respectively. The model details are given below:

  Engineering Data

E	1.0E+5 lb/in^2
A	0.1 in^2
L	10. in
M1	4.0 lb sec^2/in
M2	1.0 lb sec^2/in

Illustrations of the Model

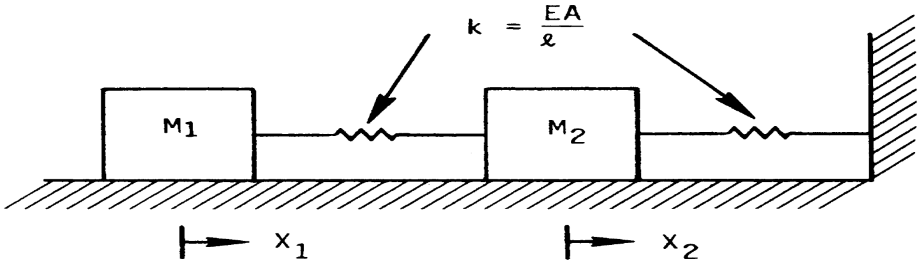


Figure 2.14-1 Two degree of freedom system using spring and concentrated masses

Model File Name and Location

Model File Name	Description	Location
v10301.dat (Reference deck from old manual is v0301.dat)	TWO D.O.F. SYSTEM MODAL ANALYSIS - Inverse Power method	tpl\verifman\oldverif
v103011.dat	TWO D.O.F. SYSTEM MODAL ANALYSIS - Lanczos Method	tpl\verifman\oldverif

Results

MSC Nastran by using rods (CONROD) and Concentrated mass (CONM2) the inverse power method gives following results and are compared with theoretical results:

	Theoretical		MSC Nastran			
			INV		LAN	
	Frequency (rad/sec)	Eigenvectors	Frequency (rad/sec)	Eigenvectors	Frequency (rad/sec)	Eigenvectors
Mode 1	10.83	1.000	10.83	1.000	10.83	1.000
		0.531		0.531		0.531
Mode 2	46.18	-0.133	46.18	-0.133	46.18	-0.133
		1.000		1.000		1.000

Conclusion

The results from MSC Nastran are comparable to the theoretical results.

2.15 Normal Modes of a Beam Structure using Component Modal Synthesis

Problem Description

Find all the natural frequencies of the clamped-clamped massless beam with the concentrated masses, shown below. (Note that the model produces uncoupled modes with integer eigenvalues.)

Solution Number

SOL 103

Features Used

- Modal Analysis.
- Component Modal Synthesis
- Givens Method

Reference

1. J. P. Den Hartog, Mechanical Vibrations 4th ed., (New York: McGraw-Hill Book Company, 1956), pp. 429-430.

Modeling Techniques Used

- As Generalized Dynamic Reduction is no longer supported, this file has been modified to use Component Modal Synthesis. As the A-set defined contains no physical GRID points, the component modes are identical to the system modes.
  The beam and concentrated mass assembly are modeled using beam (CBEAM) and concentrated masses (CONM2) respectively. The model details are given below:

  Engineering Data

  Beam Properties

A	=	1.0 (Extensional Area)
I_1	=	2. 0 (Bending moment of inertia in 1-3 plane)
I_2	=	1.0 (Bending Moment of inertia in 1-2 plane)

J	=	1.25 (Moment of inertia for Torsion)
E	=	1.0 (Young's Modulus)
G	=	1.0 (Shear Modulus)
L	=	1.0 (Length)
Mass Properties		
M1	=	1.0 (Axial)
M1, M2	=	64.0 (Lateral)
I_{11}	=	1.0
I_{22}, I_{33}	=	16.0

Note: Consistent, hypothetical units are used.

Illustrations of the Model

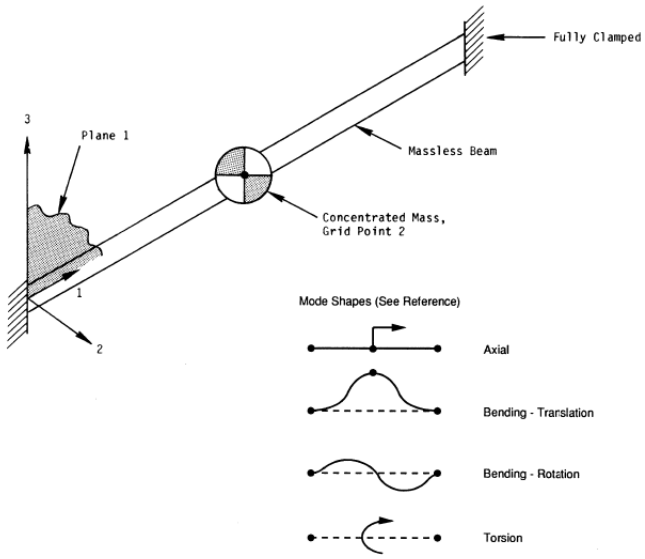


Figure 2.15-1 clamped-clamped massless beam with the concentrated masses

Model File Name and Location

Model File Name	Description	Location
`v10302.dat` (Reference deck from old manual is `v0302.dat`)	Beam-Mass Structure, Normal Modes, Gen. Dynamic Red	tpl\verifman\oldverif

Results

Theoretical Solution

For single degree of freedom systems (see (2.15-18), for reference)

$$\sqrt{\lambda} = \frac{K}{N} \tag{2.15-1}$$

$$\omega = \lambda, \text{ rad/unit time} \tag{2.15-2}$$

$$f = \frac{u}{2\pi}, \text{ cycles/ unit time} \tag{2.15-3}$$

Axial Extension Mode (T_1)

(Axial springs in parallel)

$$K_1 = \frac{EA}{l/2} + \frac{EA}{l/2} = \frac{4EA}{l} = 4.0 \tag{2.15-4}$$

$$\lambda_{T1} = 4.0/1.0 = 4.0, \omega = 2.0, f = 0.3183 \tag{2.15-5}$$

Beam Bending - Translation Mode (T_2, T_3)

See (2.15-6) for reference.

$$K_2 = \frac{192EI_2}{l^3} = 192 \tag{2.15-6}$$

$$m_2 = 64 \tag{2.15-7}$$

$$\lambda_{T_2} = 3., u = 1.732, f = 0.2757 \tag{2.15-8}$$

$$K_3 = \frac{192EI_1}{l^3} = 384 \tag{2.15-9}$$

$$m_3 = 64 \tag{2.15-10}$$

$$\lambda_{T_3} = 6., \omega = 2.4495, f = 0.3898 \tag{2.15-11}$$

Torsion Mode (R_1)

See (2.15-19) for reference.

$$K_1 = \frac{4GJ}{l} = 5 \tag{2.15-12}$$

$$l_1 = 1.0 \tag{2.15-13}$$

$$\lambda_{R_1} = 5., u = 2.2361, f = 0.3559 \quad (2.15\text{-}14)$$

Beam Bending - Rotation Mode (R_2, R_3)

See (2.15-17) for reference.

$$K_3 = \frac{16EI_2}{l^3} = 32 \quad (2.15\text{-}15)$$

$$I_{33} = 16 \quad (2.15\text{-}16)$$

$$\lambda_{R_3} = 1.0, \omega = 1.0, f = 0.1592 \quad (2.15\text{-}17)$$

$$K_2 = \frac{16EI_1}{l^3} = 32 \quad (2.15\text{-}18)$$

$$I_{22} = 15. \quad (2.15\text{-}19)$$

$$\lambda_{R_2} = 2., \omega = 1.4142, f = 0.2251 \quad (2.15\text{-}20)$$

Comparison of MSC Nastran results with theoretical results is as follows:

Mode No.	Frequency (rad/sec)	
	Theoretical Results	MSC Nastran Results
1	1.00000E+00	1.00000E+00
2	1.41420E+00	1.41421E+00
3	1.73200E+00	1.73205E+00
4	2.00000E+00	2.00000E+00
5	2.23610E+00	2.23607E+00
6	2.24495E+00	2.44949E+00

Conclusion

The results from MSC Nastran are comparable with theoretical results.

2.16 Torsional Vibration of a Shaft with Three Disks

Problem Description

Find the torsional modes of a shaft fixed at one end.

Solution Number

SOL 103

Features Used

- Modal Analysis.

Reference

1. Walter C. Hurty and Moshe F. Rubinstein, Dynamics of Structures (Englewood Cliffs: Prentice-Hall, Inc.. 1964).

Modeling Techniques Used

- The shaft and three disk assembly is modelled using beam (CBEAM) and concentrated masses (CONM2) respectively. The model details are given below:

  Engineering Data

M	=	100.0 lb-sec^2 /in (mass)
$I_1 = I_2 = I_3$	=	100.0 lb-sec^2 /in (mass moment of inertia)
L	=	10.0 in
G	=	4.0E+6 lb/in^2
ν	=	0.3
J	=	1.0 in^4 (torsional stiffness coefficient)
A	=	1.0 in^2
$I_2 = I_3$	=	0.5 in^4 (area moment of inertia)

Illustrations of the Model

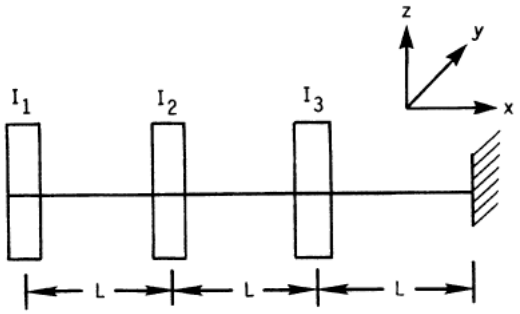


Figure 2.16-1 Shaft fixed at one end

Model File Name and Location

Model File Name	Description	Location
`v10303.dat` (Modified Givens Method) (Reference deck from old manual is v0303.dat)	Torsional Vibration of a shaft with three disks.	tpl\verifman\oldverif
`v103031.dat` (Lanczos Method)	Torsional Vibration of a shaft with three disks.	tpl\verifman\oldverif

Results

Frequency (rad/sec)	Theory	MSC Nastran	
		MGIV	LAN
ω_1	89.00	89.008	89.008
ω_2	249.40	249.396	249.396
ω_3	360.40	360.388	360.388

Conclusion

The results from MSC Nastran are comparable with theoretical results.

2.17 Cantilever Beam, Normal Modes, Restart

Problem Description

Find the two lowest natural frequencies of a cantilever beam, Restart the run from a previous statics solution, v10105d.

Solution Number

SOL 103

Features Used

- Eigenvalue solution using Modified Givens method. (Restart)

Reference

1. R.J. Roark and W. C. Young, Formulas for stress and strain (New York : McGraw-Hill Book Co.. 1975) P. 576

Modeling Techniques Used

- The cantilever beam is modelled using beam (CBEAM) element.

Engineering Data

E	=	1.0E+7 lb/in^2
ν	=	0.3
l	=	30.0 in
h	=	1.0 in
w	=	0.75 in
t	=	0.10 in
A	=	0.310 in^2
I_1	=	0.0390 in^4
I_2	=	0.0241 in^4
J	=	0.0631 in^4

Mass	=	1.0 lb-sec^2 /in
η	=	1.0 in/sec^2

Illustrations of the Model

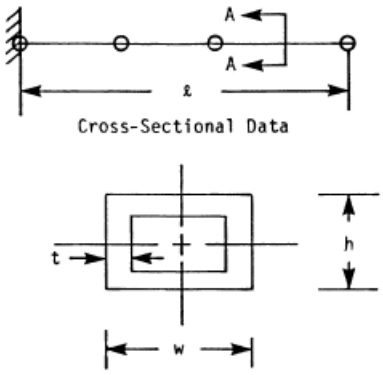


Figure 2.17-1 Cantilever Beam

Model File Name and Location

Model File Name	Description	Location
`v10105d.dat` (Reference deck from old manual is `v2405N.dat`)	Cantilever Beam Statics.	tpl\verifman\oldverif
`v10305r.dat` (Reference deck from old manual is `v0305r.dat`)	Restart from statics to calculate modes.	tpl\verifman\oldverif

Results

Theoretical Solution

$$f_1 = \frac{1.732}{2\pi}\sqrt{\frac{EIg}{wl^3}}$$

For $I_2 = 0.0241$

$$f_1 = \frac{1.732}{2\pi}\sqrt{\frac{1.0 + 7(0.0241)}{1.0(30)^3}}$$

$f_1 = 0.823$ Hz

For $I_1 = 0.039$

$$f_2 = \frac{1.732}{2\pi}\sqrt{\frac{1.0 + 7(0.039)}{1.0(30)^3}}$$

f_2= 1.048 Hz

Result comparison of MSC Nastran and theoretical results are as follows:

	Frequency (Hz)	
	Theory	MSC Nastran
Mode 1	0.823	0.823583
Mode 2	1.048	1.047686

Conclusion

The results from MSC Nastran are comparable with theoretical results. Note: Modified Givens method is good for use on small problems, but for large problems, Lanczos method is preferred.

2.18 Normal Modes Analysis with Reduction Techniques

Problem Description

Find the two lowest modes of a simply supported shaft. Compare the results of the inverse power iteration method without reduction, the Givens method with Guyan reduction and the Lanczos method.

Solution Number

SOL 103

Features Used

- Modal Analysis

Reference

1. J. P. Den Hartog, Mechanical Vibration 4th ed.. (New York: McGraw- Hill, 1956), P. 432

Modeling Techniques Used

1. General Comments

   The data generator will produce 20 equal bar elements, whose associated grid point identification numbers are 10000, 10001,...,10020.

2. Guyan Reduction

   a. A rule of thumb for the number of degrees of freedom retained for dynamic analysis is two to four times the number of modes required. Six degrees of freedom are selected by ASET1 cards.

   b. The retained degrees of freedom are placed at the expected antinodes of the two modes. Both translational and rotational degrees of freedom are used at the quarter spans.

3. Inverse Power Iteration
4. Lanczos Method

Engineering Data

L	=	100.0 in (Shaft length)
r	=	1.0 in (Shaft radius)
t	=	0.05 in (Wall thickness)

Main Index

E	=	30E+7 lb/in^2
ν	=	0.3
ρ	=	7.764E-4 lb sec^2 / in^4 (Mass density)

Illustrations of the Model

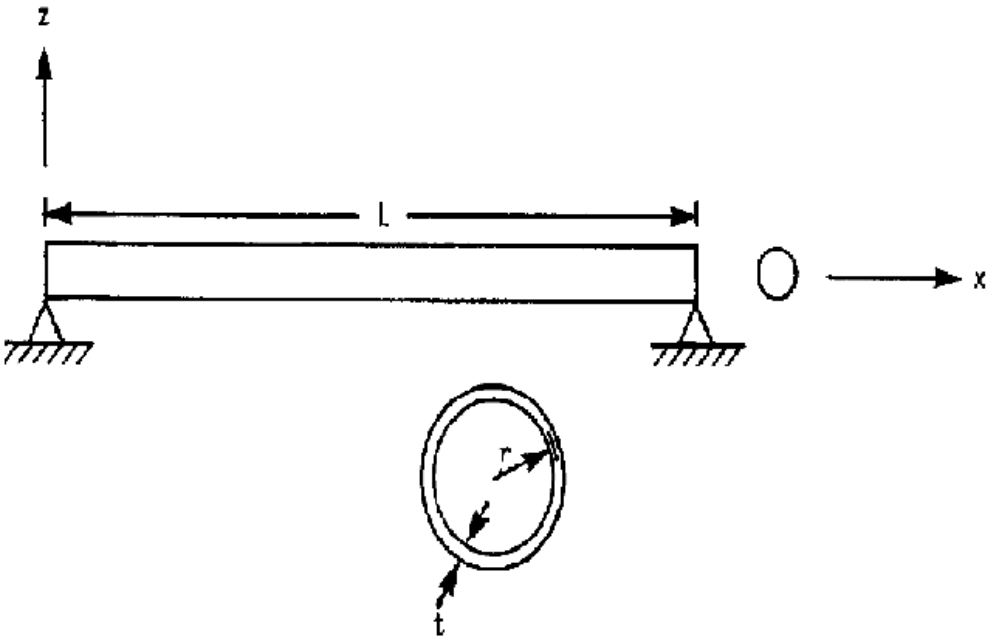


Figure 2.18-1 Simply Supported Shaft

Model File Name and Location

Model File Name	Description	Location
`v10306guy.dat` (Reference deck from old manual is `v03guy.dat`)	Simply supported shaft using Guyan reduction	tpl\verifman\oldverif
`v10306inv.dat` (Reference deck from old manual is `v03inv.dat`)	Simply supported shaft using inverse power	tpl\verifman\oldverif
`v10306lan.dat` (Reference deck from old manual is `v03lan.dat`)	Simply supported shaft using Lanczos	tpl\verifman\oldverif

Results

Theoretical Solution

The reference gives the natural frequency equation as

$$f_n = \frac{a_n}{2\pi}\sqrt{\frac{EI}{\mu_1 L^4}} \tag{2.18-1}$$

where μ_1 is the mass per unit length, n is the mode index and $a_n = (n\pi)^2$. Approximate equations for thin walled circular cylinder section properties are:

$$I = \pi r^2 t = 0.15708 \tag{2.18-2}$$

$$A = 2\pi r t = 0.31416 \tag{2.18-3}$$

$$\mu_1 = \rho A = 2.439 \times 10^{-4} \tag{2.18-4}$$

Substituting the terms

$$f_n = \left(\frac{\pi r}{e^2}\sqrt{\frac{E}{2\rho}}\right) n^2 = 21.8335 n^2 \text{ cycles/second} \tag{2.18-5}$$

Mode No.	Theory	MSC Nastran					
	Natural Frequency (Hz) Closed Form	Inverse Power No Reduction	% Difference compared to theoretical	Guyan Reduction A-set = 9	% Difference compared to theoretical	Lanczos Method	% Difference compared to theoretical
1	21.8335	21.83346	0.000%	21.8398	0.029%	21.83346	0.000%
2	87.3339	87.33327	-0.001%	87.8204	0.557%	87.33327	-0.001%
3	196.501	196.49410	-0.004%	202.7124	3.161%	196.49410	-0.004%
4	349.335	349.29400	-0.012%	411.6080	17.826%	349.29400	-0.012%
5	545.837	545.67020	-0.031%	713.3204	30.684%	545.67020	-0.031%
6	786.005	785.47620	-0.067%	1269.1490	61.468%	785.47620	-0.067%

Conclusion

The model was run using three methods. The inverse power method and Lanczos method involve no approximation and are used to give a measure of the discretization error caused by using 20 elements with lumped masses. The error in natural frequency is less than one tenth of one percent, even for worst case.

For Guyan reduction, the error for the first and second mode is well below one percent, the criteria used for picking the number of A-set points. However, the error rapidly climbs to 60 percent for the sixth mode.

2.19 Modes of Fixed Circular Plate with Trapezoidal Ring Element

Problem Description

Find the natural modes of fixed circular plate.

Solution Number

SOL 103

Features Used

- Modal Analysis.
- Inverse Power method with enhancements and Lanczos method

Reference

1. S. Timoshenko, D. H. Young and W. Weaver. Jr., Vibration Problems in Engineering, 4th ed. (New York: John Wiley and Sons, 1947) p. 501

Modeling Techniques Used

- The circular plate is modeled using triangular axisymmetric element (CTRIAX6) elements.

Engineering Data

a	=	3.5 in (Circular plate radius)
h	=	0.125 in (Plate thickness)
E	=	30E+6 lb/in^2
ν	=	0.3
g	=	386.4 in/sec^2
ρ	=	0.3 lb $/in^3$

Illustrations of the Model

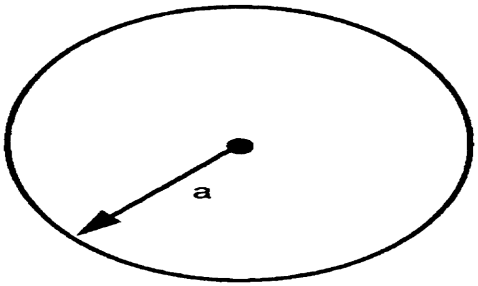


Figure 2.19-1 Circular Plate

Model File Name and Location

Model File Name	Description	Location
`v10307.dat` (Reference deck from old manual is `v0307.dat`)	Modes of Fixed Circular Plate with Trapezoidal Ring Element using Inverse Power Method with Enhancement	tpl\verifman\oldverif
`v10307lan.dat` (Reference deck from old manual - NA)	Modes of Fixed Circular Plate with Trapezoidal Ring Element using Lanczos Method	tpl\verifman\oldverif

Results

Theoretical Solution

$$\omega = \frac{\alpha}{a^2}\sqrt{\frac{D}{\rho h}} \tag{2.19-1}$$

$$D = \frac{Eh^3}{12(1-\upsilon^2)} \tag{2.19-2}$$

$$D = \frac{(30.E+6)(0.125)^3}{12(1-0.3^2)} \tag{2.19-3}$$

$$D = 5365.72 \tag{2.19-4}$$

$$\omega = \frac{\alpha}{(3.5)^2}\sqrt{\frac{5365.72}{(7.764\text{E-}4)0.125}} \tag{2.19-5}$$

$$F = \alpha(606.988) \tag{2.19-6}$$

Frequencies for N = 0		
S	α	ω (rad/sec)
0	10.21	6197.35
1	39.78	24145.98
2	88.90	53961.23

Comparison of Results

Mode No.	MSC Nastran (Frequency rad/sec)				
	Theory	SINV	%Difference	LAN	%Difference
Theory	6197.35	6191.7350	-0.091%	6191.7350	-0.091%
MSC Nastran	24145.98	23911.2700	-0.972%	23911.2700	-0.972%
Difference	53961.23	52928.0600	-1.915%	52928.0600	-1.915%

Conclusion

For MSC Nastran results using Inverse power method with enhancement and Lanzos, error is less than one percent for first two modes and it rises to approximately two percent for third mode.

2.20 Two Degree of Freedom Modal Analysis Using Superelements

Problem Description

A simple frictionless two degree of freedom system is constructed using spring and concentrated masses. Find all natural frequencies and mode shapes.

Solution Number

SOL 103

Features Used

- Modal Analysis
- Superelements Method

Reference

1. Donald. T. Greenwood, Principles of Dynamic (Englewood Cliffs: Prentice-Hall, Inc., 1965) P.459, Ex. 9-1.

Modeling Techniques Used

- The spring and concentrated mass assembly are modeled using rods (CONROD) and concentrated masses (CONM2) respectively. The model details are given below:

Engineering Data

E	=	1.0E+5 lb/in^2
A	=	0.10 in^2
R	=	10.0 in
M_1	=	4.0 lb-sec^2/in
M_2	=	1.0 lb-sec^2/in

Illustrations of the Model

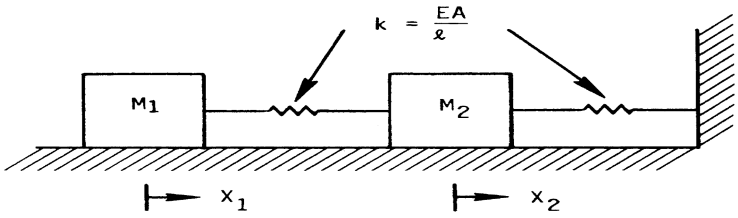


Figure 2.20-1 Two degree of freedom system using spring and concentrated masses

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10308s.dat	V6301s.dat	Two D.O.F. system modal analysis using superelement (Inverse Power Method).	tpl\verifman\oldverif

Results

MSC Nastran by using rods (CONROD) and Concentrated mass (CONM2) the inverse power method gives following results and are compared with theoretical results.

		Eigenvectors	
	Frequency (rad/sec)	Theoretical	MSC Nastran (INV)
Mode 1	10.83	1.000	1.000
		0.531	0.531
Mode 2	46.18	-0.133	-0.133
		1.000	1.000

Conclusion

MSC Nastran results are comparable with theoretical results.

2.21 Modes of Fixed Circular Plate with Superelement Method

Problem Description

Find the natural modes of fixed circular plate.

Solution Number

SOL 103

Features Used

- Modal Analysis
- Superelements Method

Reference

1. S. Timoshenko, D. H. Young and W. Weaver. Jr., Vibration Problems in Engineering, 4th ed. (New York: John Wiley and Sons, 1947) P. 501

Modeling Techniques Used

- The circular plate is modeled using CTRIAX6 elements. The model details are given below:

Engineering Data

a	=	3.5 in (Circular plate radius)
h	=	0.125 in (plate thickness)
E	=	30E+6 lb/in^2
ν	=	0.3
g	=	386.4 in/sec^2
ρ	=	0.3 lb/in^3

Illustrations of the Model

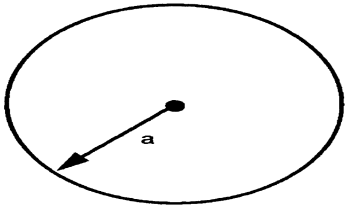


Figure 2.21-1 Circular plate

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10309s.dat	V6302s.dat	Modes of fixed circular plate with trapezoidal ring element using inverse power method with enhancement	tpl\verifman\oldverif

Results

Theoretical Solution

$$\omega = \frac{\alpha}{a^2}\sqrt{\frac{D}{\rho h}} \tag{2.21-1}$$

$$D = \frac{Eh^3}{12(1-\upsilon^2)} \tag{2.21-2}$$

$$D = \frac{(30.E+6)(0.125)^3}{12(1-0.3^2)} \tag{2.21-3}$$

$$D = 5365.72 \tag{2.21-4}$$

$$\omega = \frac{\alpha}{(3.5)^2}\sqrt{\frac{5365.72}{(7.764\text{E-}4)0.125}} \tag{2.21-5}$$

$$F = \alpha(606.988) \tag{2.21-6}$$

Frequencies for N = 0		
S	α	ω (rad/sec)
0	10.21	6197.35
1	39.78	24145.98
2	88.90	53961.23

Comparison of Results

Mode No.	Frequency (rad/sec)		
	Theory	MSC Nastran	%Difference
1	6197.35	6107.253	-1.4538%
2	24145.98	23568.68	-2.3909%
3	53961.23	52342.36	-3.0001%

Conclusion

MSC Nastran results are comparable with theoretical results using superelement method.

2.22 Natural Frequencies of a Circular Plate Immersed in Fluid

Problem Description

Find the natural modes of fixed circular plate immersed in fluid.

Solution Number

SOL 103

Features Used

- Modal Analysis

Reference

1. S. Timoshenko, D. H. Young and W. Weaver. Jr., Vibration Problems in Engineering, 4th ed. (New York: John Wiley and Sons, 1947) P. 501

Modeling Techniques Used

- The circular plate is modeled using CTRIA3 and virtual mass:

  Engineering Data

  Plate:

r	=	3.5 in (Circular plate radius)
t	=	0.125 in (plate thickness)
E	=	30E+6 lb/in^2
ν	=	0.3
g	=	386.4 in/sec^2
ρ_p	=	0.3 lb/in^3

  Fluid:

ρ_f	=	0.0385 lb/in^3 (1/g = 0.002588 sec^2/in)

Illustrations of the Model

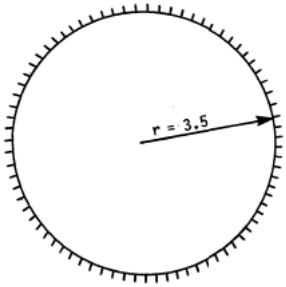


Figure 2.22-1 Circular plate immersed in fluid

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10310s.dat	V6303s.dat	Natural frequency of a circular plate immersed in fluid	tpl\verifman\oldverif

Results

Theoretical Solution

First mode:

$$f = \frac{1}{2\pi}\frac{10.21}{\sqrt{1+\eta}}\frac{1}{r^2}\sqrt{\frac{Et^2}{(1-\upsilon^2)(0.002588)\rho_p(12)}} \tag{2.22-1}$$

$$\eta = 0.6689\frac{\rho_f r}{\rho_p t} \tag{2.22-2}$$

$$\eta = 0.6689\frac{0.0385}{0.3}\frac{3.5}{0.125} = 2.4036 \tag{2.22-3}$$

Natural frequencies of a circular plate immersed in a fluid

$$f = \frac{1}{2\pi}\frac{10.21}{1+2.4036}\frac{1}{3.5^2}\sqrt{\frac{3.0\times10^7(0.125)^2}{(1-0.3^2)(0.002588)(0.3)(12)}} = 534.6 \text{ Hz} \tag{2.22-4}$$

Comparison of Results

Mode No.	Frequency (rad/sec)		
	Theory	MSC Nastran	%Difference
1	534.6	568.16	6.27%

Conclusion

MSC Nastran results are comparable with theoretical results.

2.23 DDAM Method of Response Spectra

Problem Description

The purpose of this Verification Problem is to show how to calculate peak structural response estimates in compliance with the U.S. Navy shock design modal summation convention (knowns as the NRL method). The NRL convention calls for a choice among modal contributions to peak response

$$u_{ai} = \left|\phi_{ai}\bar{q}_i\right|$$

of the maximum, u_{aM}, and summation according to the rule

$$\bar{u}_a = u_{aM} + \sqrt{\sum_{i \neq M} u_{ai}^2}$$

It is noted that the maximum contributing mode for a particular response is not necessarily the same for all physical responses.

This problem demonstrates input of a shock spectrum to a frame structure, to calculate internal loads and accelerations at points on the structure.

Solution Number

SOL 103

Features Used

- Modal Analysis

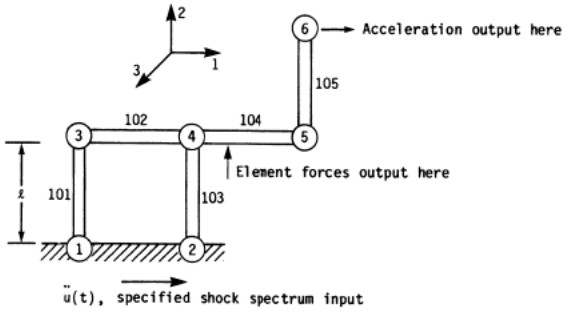
Reference

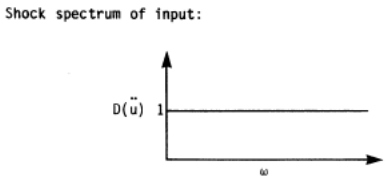
1. "Shock Design Criteria for Surface Ships", NAVSEA 0908-LP-000-3010, May 1976, P. 21.

Modeling Techniques Used

- Response spectra output may be computed in one run, without restart. A two-run sequence was used here to print out the modal displacements and element forces for inspection. Note that for efficiency the restart run only adds the SUBCASEs and bulk data entries for the shock spectrum.

Illustrations of the Model





Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10311d.dat	V6304d.dat	DDAM method of response spectra	tpl\verifman\oldverif
v10311r.dat	V6304r.dat		tpl\verifman\oldverif

Results

Theoretical Solution

Peak absolute modal responses were calculated for the eight flexible body modes (4-11). They are printed in the run with the title “MATRIX UHVR”.

Mode	Hz	$\lvert u_{hmax} \rvert$	$\lvert \dot{u}_{hmax} \rvert$	$\lvert \ddot{u}_{hmax} \rvert$
4	26.655	2.3457E-5	3.9286E-3	6.4796E-1
5	69.617	7.6307E-6	3.3378E-3	1.4600E+0
6	78.709	2.3468E-6	1.1606E-3	5.7396E-1
7	281.127	9.1068E-9	1.6086E-5	2.8414E-2
8	328.144	2.7952E-9	5.7631E-6	1.1882E-2

Mode	Hz	$\lvert u_{hmax}\rvert$	$\lvert \ddot{u}_{hmax}\rvert$	$\lvert \ddot{u}_{hmax}\rvert$
9	337.648	1.6610E-8	3.5237E-5	7.4757E-2
10	499.944	7.7927E-10	2.4479E-6	7.6894E-3
11	562.488	2.2884E-10	8.0878E-7	2.8584E-3

Estimates for peak acceleration of grid point 6 in the x-direction and bending moment in element 104 (end A) are shown below on a mode-by-mode basis as well as the NRL sum. Corresponding results for the standard SRSS and ABS conventions are also given.

The $\phi_{6x,\,i}$ column is extracted from the "REAL EIGENVECTOR" output of the first run.

The $\ddot{u}_{6x,\,i}$ column can be calculated from the product of the $\ddot{u}_h$ column times the $\phi_{6x,\,i}$ data.

The ϕ_{M104A} column is extracted from the "FORCES IN BAR ELEMENT" output.

The $M104A$ column can be calculated from the product of the u_h column times the ϕ_{M104A} data.

Mode	$\phi_{6x,\,i}$	$\ddot{u}_{6x,\,i}$	$\phi_{M104A,\,i}$	$M_{104A,\,i}$
4	1.1815+0	0.7774	3.2118+6	7.5339+1
5	5.5595-1	0.8117	6.3625+6	4.8550+1
6	1.0234-0	0.5874	3.9504+6	9.2708+0
7	6.8641-2	1.9504-3	1.1626+7	1.0588-1
8	3.1245-2	3.7125-4	4.2000+6	1.1740-2
9	2.8401-3	2.1232-4	6.1150+6	1.0157-1
10	1.1122-2	8.5520-5	5.9823+6	4.6618-3
11	1.0800-2	3.0870-5	6.4437+6	1.4746-3
	NRL	1.7861		124.77
	SRSS	1.2684		90.11
	ABS	2.1791		133.39

The enclosed modal quantities represent the largest absolute contributor among selected nodes.

Note that the second mode provides the highest acceleration component.

MSC Nastran Solution

The modes of the structure, and the eigenvectors and element forces per unit modal deflection are computed in the first run. The displacement and internal loads due to the response spectrum input are computed in the restart run.

Comparison of Results

Mode	Theory				MSC Nastran			
	Hz	$\lvert u_{hmax}\rvert$	$\lvert \dot{u}_{hmax}\rvert$	$\lvert \ddot{u}_{hmax}\rvert$	Hz	$\lvert u_{hmax}\rvert$	$\lvert \dot{u}_{hmax}\rvert$	$\lvert \ddot{u}_{hmax}\rvert$
4	26.655	2.35E-05	3.93E-03	6.48E-01	26.6550	2.3457E-05	3.9286E-03	6.5796E-01
5	69.617	7.63E-06	3.34E-03	1.46E+00	69.6170	7.6307E-06	3.3378E-03	1.4600E+00
6	78.709	2.35E-06	1.16E-03	5.74E-01	78.7090	2.3468E-06	1.1606E-03	5.7396E-01
7	281.127	9.11E-09	1.61E-05	2.84E-02	281.1300	9.1068E-09	1.6086E-05	2.8414E-02
8	328.144	2.80E-09	5.76E-06	1.19E-02	328.1400	2.7952E-09	5.7631E-06	1.1882E-02
9	337.648	1.66E-08	3.52E-05	7.48E-02	337.6500	1.6610E-08	3.5237E-05	7.4757E-02
10	499.944	7.79E-10	2.45E-06	7.69E-03	499.9400	7.7927E-10	2.4479E-06	7.6894E-03
11	562.488	2.29E-10	8.09E-07	2.86E-03	562.4900	2.2884E-10	8.0878E-07	2.8584E-03

Mode	Theory				MSC Nastran			
	$\phi_{6x,\,i}$	$\ddot{u}_{6x,\,i}$	$\phi_{M104A,\,i}$	$M_{104A,\,i}$	$\phi_{6x,\,i}$	$\ddot{u}_{6x,\,i}$	$\phi_{M104A,\,i}$	$M_{104A,\,i}$
4	1.1815E+0	0.7774	3.2118E+6	7.5339E+1	1.1815+0	0.7773797	-3.2118E+06	-7.5339E+01
5	5.5595E-1	0.8117	6.3625E+6	4.8550E+1	5.56E-01	0.8116918	6.3625E+06	4.8551E+01
6	1.0234E-0	0.5874	3.9504E+6	9.2708E+0	-1.02E+00	-0.587405	-3.9504E+06	-9.2709E+00
7	6.8641E-2	1.9504E-3	1.1626E+7	1.0588E-1	6.86E-02	0.0019504	1.1626E+07	1.0587E-01
8	3.1245E-2	3.7125E-4	4.2000E+6	1.1740E-2	-3.12E-02	-0.000371	1.1626E+07	3.2496E-02
9	2.8401-E3	2.1232E-4	6.1150+E6	1.0157E-1	-2.84E-03	-0.000212	-6.1150E+06	-1.0157E-01
10	1.1122-E2	8.5520E-5	5.9823+E6	4.6618E-3	1.11E-02	8.552E-05	5.9823E+06	4.6618E-03
11	1.0800-E2	3.0870E-5	6.4437+E6	1.4746E-3	-1.08E-02	-3.09E-05	-6.4437E+06	-1.4746E-03

Conclusion

MSC Nastran results are comparable with theoretical results.

2.24 Lateral Buckling of Cantilever Beam

Problem Description

A Cantilever beam of constant cross section is loaded at the free end with a vertical shear load. The critical buckling condition occurs when the deflected shape in the vertical plane is unstable and lateral buckling occurs. Determine the critical buckling eigenvalues and loads for the structure.

Solution Number

SOL 105

Features Used

- Buckling Analysis

Reference

1. Stephen P. Timoshenko and James M. Gere. Theory of Elastic Stability, 2nd ed. (New York: Mc Graw-Hill Book Co. 1961), P. 257

Modeling Techniques Used

- The model must have sufficient nodes to accurately represent the buckled shape. Six or more grids per half wavelength are recommended.

Engineering Data

b	=	0.05 in
A	=	0.05 in^2
I_1	=	0.00417 in^4
J	=	4.035E-5 in^4
E	=	1.0E+8 lb/in^2
G	=	3.0E+7 lb/in^2
L	=	20. in
h	=	1.0 in
I_2	=	1.0417 E^{-5} in^4

Illustrations of the Model

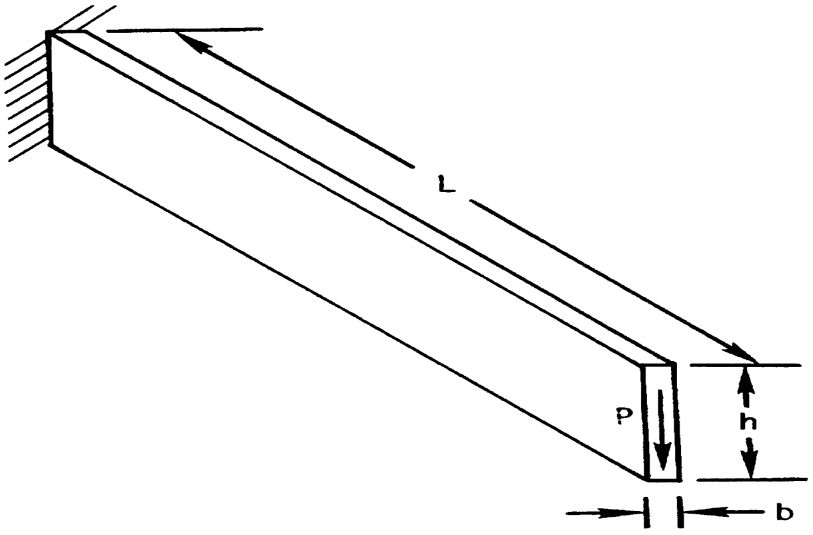


Figure 2.24-1 Cantilever Beam

Model File Name and Location

Model File Name	Description	Location
`v10501.dat` (Reference deck from old manual is `v0501.dat`)	Lateral Buckling of Cantilever Beam.	tpl\verifman\oldverif

Results

Theoretical Solution

$$P_{CR} = \sqrt{\frac{(EI)(GJ)}{L^2}} N_{CR} \qquad (2.24\text{-}1)$$

Lateral Buckling of a Cantilever Beam

$N_{CR} = \pm 2.0063, \pm 5.1231, \pm 8.2580$ (Derived in the reference) (2.24-2)

$P_{CR} = \pm 11.27, \pm 28.77, \pm 46.37$ (2.24-3)

MSC Nastran Solution

Eigenvalue $\lambda = \frac{P_{CR}}{P}$, solved from $[K - \lambda K_d]\{u\} = 0$

Letting P = 50, then

$\lambda = \pm 0.2254, \pm 0.5783, \pm 0.9402$

$P_{CR} = \pm 11.27, \pm 28.91, \pm 47.01$

Comparison of Results

	Solution for P_{CR}		
	Buckling Mode (lb)		
	1st	2nd	3rd
Theory	±11.27	±28.77	±46.37
MSC Nastran	±11.27	±28.91	±47.01
Difference	0.0%	0.50%	1.38%

Conclusion

MSC Nastran results are comparable with theoretical results.

2.25 Simple Frame Analysis with Buckling

Problem Description

A simple frame with lateral motion constrained, is loaded vertically. Using Static analysis, find the vertical column forces and stresses. Then release the constraint to allow lateral motion and find the lowest critical buckling load.

Solution Number

SOL 105

Features Used

- Static and Buckling Analysis

Reference

1. Stephen P. Timoshenko and James M. Gere. Theory of Elastic Stability, 2nd ed. (New York: Mc Graw-Hill Book Co. 1961), P. 149

Modeling Techniques Used

- Simple frame is modeled using Bar elements (CBAR).

Engineering Data

E	=	1.0E+6 lb/in^2
A	=	1.0 in^2
l	=	100. in
$I_1 = I_2$	=	1.0 in^4
P	=	500. lb

Illustrations of the Model

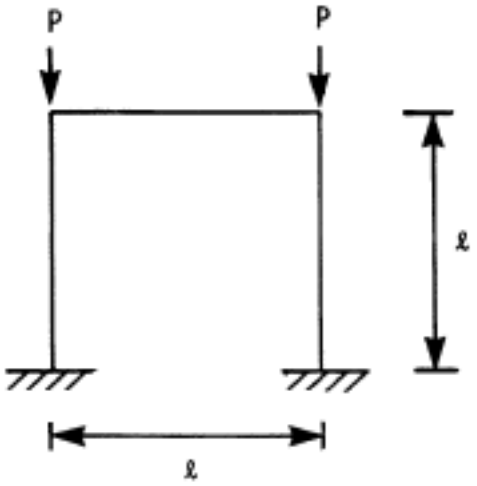


Figure 2.25-1 Simple Frame

Model File Name and Location

Model File Name	Description	Location
`v10502a.dat` (Reference deck from old manual is `v0502a.dat`)	Lateral Buckling of Cantilever Beam.	tpl\verifman\oldverif

Results

Theoretical Solution

Static Analysis of Symmetric Deformation Mode

Axial force in vertical column: -500.0 lb (comp.)

Axial stress of vertical column: -500.0 lb/in^2

Buckling Analysis of Antisymmetric Deformation Mode

$$k^2 = \frac{P_{CR}}{EI}$$

where kL = -6 tan (kL)

lowest k: $k_1 = 2.71645974 \times 10^{-2} \Rightarrow p = k^2EI = 737.9$ lb

MSC Nastran Solution

Static Analysis

Axial force in vertical elements: -500.0 lb

Axial stress of vertical elements: -500.0 lb/in^2

Buckling Analysis

Mode 1 Eigenvalue

$$\lambda_1 = 1.475288 \Rightarrow P_{CR} = P_{Applied} \cdot \lambda_1 = 737.6 \text{ lb}$$

Comparison of Results

	Force (lb)	Stress (lb/in^2)	P_{CR} (lb)
Theory	-500	-500	737.9
MSC Nastran	-500	-500	737.6
Difference	0.0%	0.0%	0.041%

Conclusion

MSC Nastran results are comparable with theoretical results.

2.26 Beam with Gap/Lift-Off

Problem Description

A simply supported beam is hinged at one end and supported by lifting rollers at two other locations. Allowing lift-off to occur at the rollers, determine the vertical deflections under the load points.

Solution Number

SOL 106

Features Used

- Nonlinear statics

Reference

1. Jack C. McCormac, Structural Analysis, 3rd ed. (New York: Intext Educational Publishers, 1957) P. 323, ex 17.12.

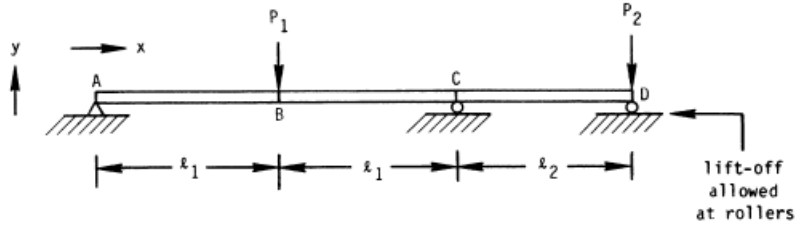
Modeling Techniques Used

- The model details are given below:

Engineering Data

E	=	29.0E+06 lb/in^2
L_1	=	15 ft
L_2	=	10 ft
I	=	1000 in^4
P_1	=	40000 lb
P2	=	10000 lb

Illustrations of the Model



Model File Name and Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10601s.dat	V6601s.dat	Beam with gap/lift-off	tpl\verifman\oldverif

Results

Theoretical Solution

(Lift-off occurs at point D only)

Point B: Uy = -1.01 in

Point D: Uy = +0.546 in

MSC Nastran Solution

Gap element were used to model the roller supports at points C and D. Iteration allowed lift-off to occur at D. Final Deflection were:

Point B: Uy = -1.009449E+00 in

Point D: Uy = 5.443316E-01 in

Conclusion

The results from MSC Nastran are comparable to the theoretical results.

2.27 Cantilever Beam with Plastic Hinge

Problem Description

Given a cantilever beam with a concentrated end loading, find the collapse load using plastic hinge analysis.

Solution Number

SOL 106

Features Used

- Nonlinear statics

Reference

1. A. Mendelson, PLASTICITY: Theory and Application (New York: The Macmillan Company)P. 308

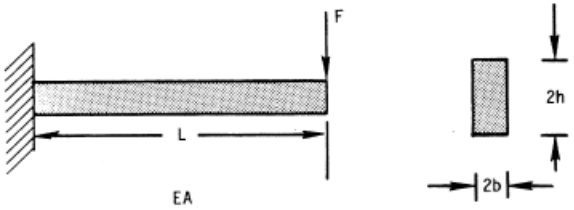
Modeling Techniques Used

- The model details are given below:

Engineering Data

E	=	1.0E+07
ν	=	0.3
E_T	=	0.0
σ_{yld}	=	1.0E+4
A	=	0.1
b	=	0.05
h	=	0.5
F	=	$F_i + \Delta F$
F_i	=	16.0
ΔF	=	0.5

Illustrations of the Model



Model File Name and Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10603s.dat	V6603s.dat	Cantilever beam with plastic hinge	tpl\verifman\oldverif

Results

Theoretical Solution

With a concentrated end load, the maximum moment occurs at the base. Collapse occurs when the moment at the base causes the root cross section to become fully plastic.

Collapse Load: $F = 2bh^2\sigma_{yld}/L = 25.0$

MSC Nastran Solution

Using a Beam element with the elastoplastic, Von Mises criteria, the beam is loaded ($\delta F = 0.5$) until collapse occurs (large tip deflection).

Collapse Load: F = 25.0

Conclusion

The results from MSC Nastran are comparable to the theoretical results.

2.28 Complex Eigenvalues of Three-by-Three Matrix

Problem Description

Given the real, unsymmetric matrix [A], find its characteristic values ("eigenvalues") and modal matrix ("matrix of eigenvectors").

Solution Number

SOL 107

Features Used

- Direct Complex Eigenvalues

Reference

1. Paul M. DeRusso, Rob J. Roy, and Charles M. Close, State Variables for Engineers (New York: John Wiley & Sons, 1967), ex. 4.7-2

Modeling Techniques Used

1. The Program requires at least one structural element. When using DMIG input, put in at least one ELAS or other element.
2. The standard form of the eigenproblem, used in the reference, is $[A-\lambda I]\{u\} = 0$, where [I] is the identity matrix. The program solves the equation $[p^2M+pB+K]\{u\} = 0$, One Method to 'model' the original equation is to input [1] for [M], let [B] be null, and input [A] for [K]. The output quantity p_i is then converted to λ_i units by,

   $$\lambda_i = -p_i^2$$

   Engineering Data

   $$A = \begin{bmatrix} 2 & -2 & 3 \\ 1 & 1 & 1 \\ 1 & 3 & -1 \end{bmatrix}$$

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10701.dat	V2801.dat	Complex eigenvalues of three-by-three matrix	tpl\verifman\oldverif

Results

Theoretical Solution

Evaluate $[A - \lambda I] = 0$ to produce the characteristic equation $\lambda^3 - 2\lambda^2 - 5\lambda + 6 = 0$, whose roots are tabulated below. Their validity can be determined by substitute into the characteristic equation. The reference calculates the eigenvector matrix [M], tabulated below, by use of adjoint matrix techniques. Its validity can be checked by back-substitution into the equation $[A - \lambda_i I]\{M\} = 0$, where $\{M_i\}$ is the ith column of M.

Results	First	Second	Third
Eigenvalues (λ_i)	-2	1	3
Roots of quadratic (p_i)	1.414214	(0, 1.0i)	(0, 1.732051i)
Eigenvectors	11	-1	1
	1	1	1
	-14	1	1

Eigenvectors are arbitrary to a scaling constant. The MSC Nastran results were renormalized by hand to obtain the results tabulated above.

Conclusion

The closed form and tabulated solutions agree exactly to all printed digits. (MSC NASTRAN outputs 6 or 7 digits depending on machine type.)

2.29 Complex Roots as a Function of Gain

Problem Description

Find the root-locus of the control system shown in the figure.

Solution Number

SOL 107

Features Used

- Direct Complex Eigenvalues

Reference

1. C. J. Savant, Jr., Basic Feedback Control System Design (New York: McGraw-Hill Book Company, Inc., 1958)

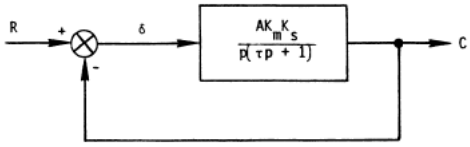
Modeling Techniques Used

1. The equation is multiplied by t. The coefficient of the p^2 term is then t, and the coefficient of the p term is unity.
2. Three uncoupled oscillators are modeled, to simulate changes in gain. Values were chosen that correspond to under-damped, critically damped, and over-damped systems.

   Engineering Data

t	=	0.01 (Time constant)
K_m	=	5.0 (Motor constant)

Solve for AKs = 1.0, 5.0, 10.0



Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10702.dat	V2802.dat	Complex roots as a function of gain	tpl\verifman\oldverif

Results

Theoretical Solution

The denominator of the transfer function is,

$$p^2 + \frac{1}{\tau}p + \frac{AK_mK_s}{\tau} = 0$$

Solutions of this quadratic equation are tabulated for three values of AKs as given on page 82 of the reference. They are:

AKs	Root Location	
	P_1	P_2
1.0	-94.7	-5.3
5.0	-50.0	-50.0
10.0	-50 + j50	-50 - j50

MSC Nastran Solution

All roots agree to all places printed, when rounded off.

Conclusion

MSC Nastran results are comparable with theoretical results.

2.30 Frequency Response of a Single Degree of Freedom System

Problem Description

A 3000-pound weight is attached to a platform by a spring and a damper. If the platform is moved up and down at the system's resonant frequency with an amplitude of one inch, find the maximum amplitude of the 3000-pound weight.

Solution Number

SOL 108

Features Used

- Direct Frequency Response

Reference

1. J.P. Den Hartog, Mechanical vibrations (New York: McGraw-Hill Book Company,1956), pp. 49-55

Modeling Techniques Used

Engineering Data

K	=	333.0 lb/in
C	=	28.0 lb/in/sec
a_0	=	1 in

$$M_1 = \frac{3000 \text{ lb}}{\frac{32.2 \text{ ft}}{\text{sec}^2} \cdot \frac{12 \text{ inch}}{1 \text{ ft}}}$$

$$M_1 = 7.764 \text{ lb-sec}^2/\text{in}$$

$$\omega_n = \sqrt{\frac{k}{M_1}} = 6.5491 \text{ rad/sec} = 1.042 \text{ cyc/sec}$$

Illustrations of the Model

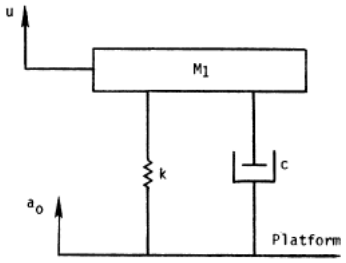


Figure 2.30-1 Single Degree of Freedom System

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10801.dat	V2601.dat	Frequency response of single degree of system	tpl\verifman\oldverif\

Results

Theoretical Solution

The motion of M_1 is given by equation 2.28a of the reference (p. 49)

$$u = \frac{P_0/k}{\sqrt{(1-\omega^2/\omega_n^2)^2 + (2\cdot c/c_c\cdot \omega/\omega_n)^2}}$$

where,

$P_0 = \sqrt{(ka_0)^2 + (ca_0\omega)^2}$

$c_c = 2\sqrt{km}$

$\omega = \omega_n$ = excitation frequency (Hz)

Solving the above equations yield,

$P_0 = 380.1517$ lb

Then, $u = \frac{P_0}{c\omega} = \frac{380.1517}{28.0(6.5491)} = 2.0731$ inch

	Amplitude (inches)
Theory	2.0731
MSC Nastran	2.073576
% Error	0.023%

Conclusion

MSC Nastran results are comparable with theoretical results.

2.31 Transient Response of a Rocket - Direct Time Integration

Problem Description

A rocket is subjected to axial loading for a finite duration of time. Using axial elements in a free-free structure, find the displacement histories of the midpoint and two ends, considering both the rigid body motion and the structural vibration.

Solution Number

SOL 109

Features Used

- Direct Transient Response

Reference

1. J.S. Przemieniecki, Theory of Matrix structural Analysis (New York: McGraw-Hill Book Co., 1968) P. 340

Modeling Techniques Used

- The following is details of the model which is modeled using CONROD element. The model details are given below:

  Engineering Data

E	=	1.0E+4 lb/in^2
A	=	1.0 in^2
L	=	140.0 in
ρ	=	0.1
P_0	=	100.0 lb
t_0	=	1.0 sec

Illustrations of the Model

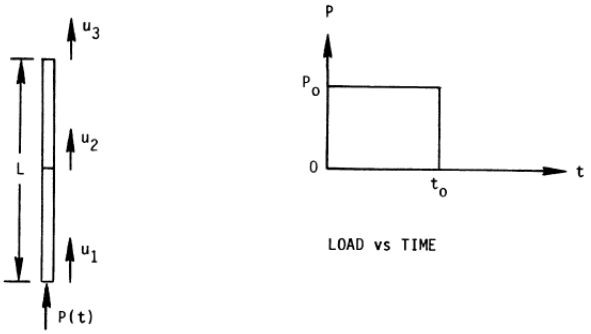


Figure 2.31-1 Single Degree of Freedom System

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10901.dat	V2701.dat	Transient response of a rocket - direct time integration	tpl\verifman\oldverif

Results

Theoretical Solution

Using the matrix forms for the equations of motion, with zero initial conditions, the base response for $t > t_0$ can be written as:

$$u_1(t) = \frac{P_0 t_0 (2t - t_0)}{2\rho AL} + \frac{P_0}{4A}\sqrt{\frac{3}{E_\rho}}\left\{\frac{2}{\omega_2}[\cos\omega_2(t-t_0) - \cos\omega_2 t] + \frac{1}{\omega_3}[\cos\omega_3(t-t_0) - \cos\omega_3 t]\right\}$$

where,

$$\omega_2 = \sqrt{\frac{12E}{\rho L^2}} \; ; \; \omega_3 = \sqrt{\frac{48E}{\rho L^2}}$$

For the given engineering data, the base displacement is:

at t=2.00; u_1=10.89972 in

at t=2.15; u_1=11.73228 in

	Vertical Displacement of Base, u_1	
	t=2.0	t=2.15
Theory	10.89972	11.73228
MSC Nastran	10.08471	11.06770
% Difference	-7.477%	-5.665%

Conclusion

MSC Nastran results are comparable with theoretical results.

2.32 Response Spectra of a Single Oscillator

Problem Description

A large mass, M_1, is subjected to a force doublet. A very small mass, M_2, is attached to it with a spring and damper, tuned to an undamped natural frequency of one Hz. A response spectrum is computed for the large mass at this same frequency. A variable, v_3, is connected by a multipoint constraint and is a model of the relative displacement between the large and small mass. As the small mass-spring combination has the same physical model as the response spectrum calculation, it should produce the same results as the inertial motion component (variable 2) and the relative motion component (variable 3).

Solution Number

SOL 109

Features Used

- Direct Transient Response

Reference

1. Harris, Cyril M. and Crede, Charles E., Shock and Vibration Handbook, (New York: McGraw-Hill Book Company, Inc., 1961)

Modeling Techniques Used

- The model details are given below:

Engineering Data

M_1	=	1.0E+6
M_2	=	1.0
k_{12}	=	$2\pi^2$
c_{12}	=	$0.01 \times 2M_2 \times 2\pi$
	=	0.125664
ρ_0	=	1.0E+6

Illustrations of the Model

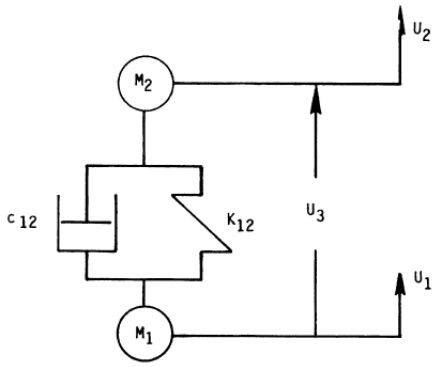


Figure 2.32-1 Single Degree of Freedom System

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10902.dat	V2702.dat	Response spectra of a single oscillator	tpl\verifman\oldverif

Results

Magnitude of Peak Response, Inertial Coordinates

	Theory	MSC Nastran	% Difference
Displacement	2.000315	2.0000	-0.02%
Velocity	1.000107	1.0000	-0.01%
Acceleration	1.00237	1.0019	-0.05%

Magnitude of Peak Response, Relative Coordinates

	Theory	MSC Nastran	% Difference
Displacement	0.025406	2.5377E-02	-0.1136%
Velocity	0.051437	5.1440E-02	0.0055%

Conclusion

MSC Nastran results are comparable with theoretical results.

2.33 Transient Response with Combined Loading Functions

Problem Description

Three identical structures are given different time-varying loads. The loads on the third structure are a linear combination of the loads on the other two structures. It is shown that the response of the third structure is also the same linear combination of the response of the first two structures.

Solution Number

SOL 109

Features Used

- Direct Transient Response

Modeling Techniques Used

- The model details are given below:

Engineering Data

E	=	1.0E+7 lb/in^2
L	=	10 in
A	=	10^4 in^2
I_1	=	1.0 in^4

Illustrations of the Model

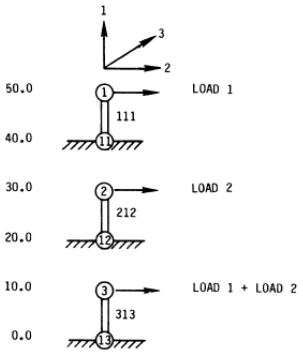


Figure 2.33-1 Single Degree of Freedom System

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10903.dat	V2703.dat	Transient response with combined loading function	tpl\verifman\oldverif

Results

All three systems are sampled at t = 1.5, with their displacements tabulated below seen that the response of the third structure is the sum of the other two.

Response at t=1.5	
Mass 1	1.576766E-05
Mass 2	-1.361804E-05
sum	**2.149620E-06**
Mass 3	2.149617E-06
% Difference	0.000%

Conclusion

It is proved by MSC Nastran that response of the third structure is also same linear combination of the response of the first two structures.

2.34 Direct Transient Response to an Internal Pressure

Problem Description

Find the peak displacements in a thick-walled cylinder resulting from a suddenly applied internal pressure load.

Solution Number

SOL 109

Features Used

- Direct Transient Response

Reference

1. S. H. Crandall, N. C. Dahl, and T.J. Lardner, An Introduction to the Mechanics of Solids, 2nd ed. (New York: McGraw-Hill book Company, 1972) pp. 293-297

Modeling Techniques Used

- The model is the same thick-walled cylinder given in Verification Problem No. v10110. The model details are given below:

Engineering Data

r_i	=	6.0 in
r_o	=	12.0 in
h	=	8.0 in
E	=	30.0E+6 psi
P	=	0.28 lb/in^3
ν	=	0.0
p	=	10.0 lb/in^2

Illustrations of the Model

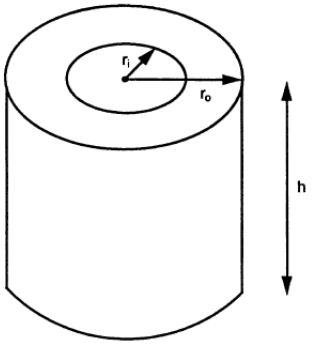


Figure 2.34-1 Thick walled Cylinder

Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v10904.dat	V2704.dat	Direct transient response to an internal pressure	tpl\verifman\oldverif

Results

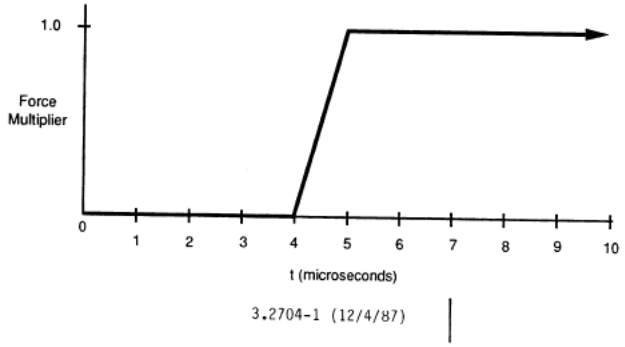
Theoretical Solution

Radial displacements from a static analysis (SOL 101) were: Grid 1, 3.326E-6 in.; Grid 7, 2.773E-6 in.; and Grid 11, 2.673E-6 in.

MSC Nastran Solution

The force was applied in an instantaneous manner, as shown below. Several time steps of zero load were input to ensure simulation of a step function in the loading. The integration time step was chosen to be 1.0 microsecond. Approximately three cycles of response were simulated, and damping was neglected. Note that

for a single degree-of-freedom system, an instantaneously applied load will give a maximum displacement of twice the static displacement.



Comparison of Results

Node	Static Displacement	Peak Transient Displacement	Ratio Transient/Static
1	3.33E-06	6.56238E-06	1.97
7	2.77E-06	5.65275E-06	2.04
11	2.67E-06	5.74974E-06	2.15

Conclusion

For a Single degree of freedom system, an instantaneously applied load will give a maximum displacement of twice the static displacement.

2.35 Complex Eigenvalues of Three-by-Three Matrix, Modal Method

Problem Description

Given the same unsymmetric matrix [A] used in problem v10701, find its eigen solution in two steps.

1. Add enough terms to [A] to produce a symmetric matrix. Find the real eigensolution of this matrix to use as modal coordinates.
2. Add in the terms to reconstruct [A], and compute its eigensolution.

Use the so-called 'particular' matrix [A_B] of reference 2 as the symmetric matrix, as it has a closed-form solution for its eigenvalues. Use the negative of this matrix, as this is more typical of the values encountered on structures.

Solution Number

SOL 110

Features Used

- Modal Complex Eigenvalues

Reference

1. Verification Problem No. v10701
2. Marvin Marcus, 'Basic Theorems in Matrix Theory, "National Bureau of Standards Applied Mathematics Series, 57 (1964)"

Modeling Techniques Used

1. The mass matrix is input to DMIG cards as the identity matrix.
2. The [D] matrix is input on the system after the transformations based on the real eigensolutions are made on the symmetric portion. As there was no truncation of the modal coordinates, an exact answer can be expected, even though the terms in [K2PP] are of the same magnitude as [A_B].

Engineering Data

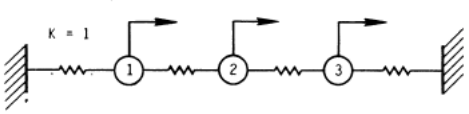
[A] is tabulated in v10701. $[A_B]$ has -2 on the diagonal, +1 above and below the diagonal, and zeros elsewhere. The difference matrix [D] is then

$[D] = [A] - [-A_B] = [A] + [A_B]$

$$\begin{bmatrix} 0 & -1 & 3 \\ 2 & -1 & 2 \\ 1 & 4 & -3 \end{bmatrix} = \begin{bmatrix} 2 & -2 & 3 \\ 1 & 1 & 1 \\ 1 & 3 & -1 \end{bmatrix} + \begin{bmatrix} -2 & 1 & 0 \\ 1 & -2 & 1 \\ 0 & 1 & -2 \end{bmatrix}$$

$-[A_B]$ is equivalent to the physical system

Illustrations of the Model



Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v11001.dat	V2901.dat	Complex eigenvalues of three-by-three matrix, modal method	tpl\verifman\oldverif

Results

Theoretical Solution

The theory of the complex eigensolution is described in v10701. A closed form solution for the real eigensolution to $[A_B]$ is given in Reference 2 as

$$\lambda_i = -4 \sin\{i\pi/[2(N+1)]\}^2$$

where N is the size of the matrix.

For N = 3, the roots of $[-A_B]$ are:

$$\lambda_1 = 4 \sin\left(\frac{\pi}{8}\right)^2 = 0.585786$$

$$\lambda_2 = 4 \sin\left(\frac{\pi}{4}\right)^2 = 2.0$$

$$\lambda_3 = 4 \sin\left(\frac{3\pi}{8}\right)^2 = 3.414214$$

Comparison of Results

	Eigenvalues		
	Theory	MSC Nastran	% Difference
Mode 1	0.5858	0.5858	0.00%
Mode 2	2.0000	2.0000	0.00%
Mode 3	3.4142	3.4142	0.00%

Both the intermediate solution to the real eigenvalue problem, tabulated above, and the final complex eigen solution, tabulated in v10701, agree to all places printed.

Conclusion

The MSC Nastran results are comparable with theoretical results.

2.36 Modal Frequency Response of a Beam Structure

Problem Description

Find the response at response of the model described in problem v10302 for six values of damping.

Solution Number

SOL 111

Features Used

- Modal Frequency Response

Reference

1. J. P. Den Hartog, Mechanical Vibrations, 4th ed. (New York: McGraw- Hill Book Company, 1956) pp. 47-49

Modeling Techniques Used

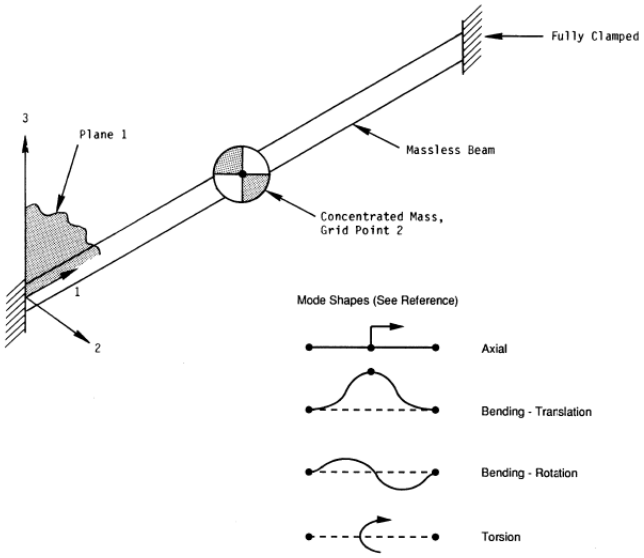
Please refer problem v10302.

Engineering Data

The structure is described in Problem v10302. In that problem the natural frequency of each of the structure's uncoupled modes were found. Here we apply damping to each of these modes as follows:

Mode	Natural Frequency (cyc/sec)*	% Critical Damping (c/c_c)
Axial (T_1)	0.3183	50%
Bending - Translational (T_2)	0.2757	20%
Bending - Translational (T_3)	0.3898	200%
Torsion (R_1)	0.3559	100%
Bending - Rotational (R_2)	0.2251	2%
Bending - Rotational (R_3)	0.1592	1%

Illustrations of the Model



Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v11101.dat	V3001.dat	Modal frequency response of a beam structure	tpl\verifman\oldverif

Results

Theoretical Solution

The response of a single degree of freedom oscillator at resonance is given in the reference by equation 2.28a (page 49). Simplifying that equation for our case gives,

$$x|_{\omega = \omega_n} = \frac{P_0/k}{2c/c_c} \qquad (2.36\text{-}1)$$

Where,

x = Magnitude of displacement

P_0 = Magnitude of applied load

c = viscous damping coefficient

c_c = viscous damping coefficient required for critical damping

Phase angle data is plotted on page 51 of the reference. The graph shows a value of 90^0 for all values of damping.

As shown in problem v0302, each degree of freedom shows different stiffness. Here we apply a load proportional to each stiffness at its corresponding degree of freedom ($|P_0| = |k|$).

With this assumption, equation reduces to

$$\frac{x}{x_{static}} = \frac{1}{2c/c_c} \tag{2.36-2}$$

Where, $x_{static} = \frac{P_0}{k}$

We obtain for each degree of freedom:

Mode	Response (x/x_{static})
Axial (T_1)	1.0000
Bending - Translational (T_2)	2.5000
Bending - Translational (T_3)	0.2500
Torsion (R_1)	0.5000
Bending - Rotational (R_2)	25.000
Bending - Rotational (R_3)	50.000

Comparison of Results

Frequency	Mode	Theory	MSC Nastran	% Difference
3.1831E-01	Axial (T_1)	1.00E+00	1.00E+00	0.00%
2.7566E-01	Bending - Translational (T_2)	5.00E+00	5.00E+00	0.00%
3.8985E-01	Bending - Translational (T_3)	2.50E-01	2.50E-01	0.00%
3.5588E-01	Torsion (R_1)	5.00E-01	5.00E-01	0.00%
2.2508E-01	Bending - Rotational (R_2)	2.50E+01	2.50E+01	0.00%
1.5915E-01	Bending - Rotational (R_3)	5.00E+01	5.00E+01	0.00%

Conclusion

The MSC Nastran response solutions agree with the above closed-form solutions to all places printed. The MSC Nastran phase angles are also in agreement with the theoretical results, after accounting for the MSC Nastran sign convention.

2.37 Transient Response of a Beam Modal Method

Problem Description

A simply supported beam is mounted on elastic springs. Given a time-dependent symmetric distributed loading function, find the transient response of the midspan deflections. Use the Modal method, considering symmetric modes only.

Solution Number

SOL 112

Features Used

- Modal Transient Response

Reference

1. Walter C. Hurty and Moshe F. Rubinstein, Dynamics of Structure, (Englewood Cliffs: Prentice-Hall, Inc., 1964), P. 294.

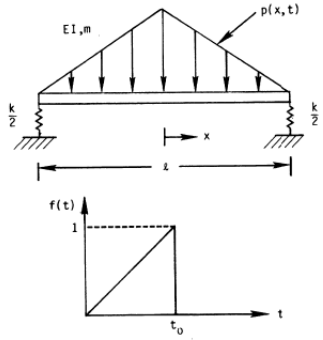
Modeling Techniques Used

- The model details are given below:

Engineering Data

m	=	0.1 (mass per length)
E	=	1.0E+07
l	=	100
K	=	$\pi^4 EI/l^3 = 974.1$
$P(x, t)$	=	$\left(1 - \frac{2x}{l}\right)\frac{2P_0}{l} \cdot f(t),\ 0 \le x \le \frac{l}{2}$
t_0	=	1.0
P_0	=	100.0

Illustrations of the Model



Model File Name and Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v11201.dat	V3101.dat	Modal frequency response of a beam structure	tpl\verifman\oldverif

Results

Theoretical Solution

Using a Rayleigh-Ritz approximation of the first three symmetric modes, an equation for the center deflection from the modal method is

$$v = \omega(0,t) = \frac{-P_0 l^3}{\pi^4 EI}[2.603D_1(t) + 0.050D_2(t)] \tag{2.37-1}$$

where,

$$D_1(t) = \frac{1}{\omega_1 t_0}[\sin\omega_1(t-t_0) - \sin\omega_1 t] + \cos\omega_1(t-t_0) \tag{2.37-2}$$

$$D_2(t) = \frac{1}{\omega_2 t_0}[\sin\omega_2(t-t_0) - \sin\omega_2 t] + \cos\omega_2(t-t_0) \tag{2.37-3}$$

$$\omega_1 = 0.725\frac{\pi^2}{l^2}\sqrt{\frac{EI}{m}} = 7.155 \tag{2.37-4}$$

$$\omega_2 = 2.860\frac{\pi^2}{l^2}\sqrt{\frac{EI}{m}} = 28.3 \tag{2.37-5}$$

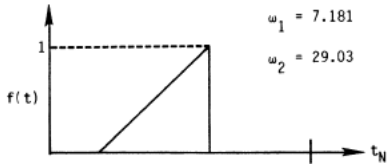
Local maximum midspan deflections:

$$t = 1.45 , \nu = 0.23382 \quad (2.37\text{-}6)$$

$$t = 1.9 , \nu = -0.24283 \quad (2.37\text{-}7)$$

MSC Nastran Solution

Since only symmetrical modes are considered, only half of the beam is modelled. Four CBEAM elements for the half span are used, and one CONROD is used to model the spring. The loading was shifted by 0.1 to ensure zero initial conditions.



Local maximum midspan deflections:

$$t = t_N - 0.1 = 1.45 , \nu = 0.23418 \quad (2.37\text{-}8)$$

$$t = t_N - 0.1 = 1.90 , \nu = -0.24119 \quad (2.37\text{-}9)$$

Comparison of Results

	Local max, ν		Natural Frequency	
	t = 1.45	t = 1.9	ω_1	ω_2
Theory (Rayleigh-Ritz)	0.2338	-0.2428	7.1550	28.2300
MSC Nastran	0.2342	-0.2412	7.1808	29.0297
% Difference	0.15%	-0.67%	0.36%	2.83%

Conclusion

The results from MSC Nastran are comparable to the theoretical results.

2.38 Deflection of a Circular Plate, Cyclic Symmetry

Problem Description

Find the maximum displacement of a circular plate with a uniform pressure load and fixed edges.

Solution Number

SOL 114

Features Used

- Cyclic Static

Reference

1. Fred G. Seely and James O. Smith, Advanced Mechanics of Materials, 2nd ed. (New York: John Wiley & Sons, 1952) P. 227

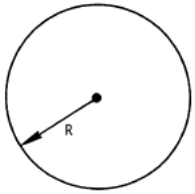
Modeling Techniques Used

- The model details are given below:

Engineering Data

R	=	11.0 in
T	=	0.10 in
E	=	10.0E+6 lb/in^2
ν	=	0.3
p	=	1.0 psi

Illustrations of the Model



Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v11401.dat	V4701.dat	Deflection of a circular plate, cyclic symmetry	tpl\verifman\oldverif

Results

Theoretical Solution

For a circular plate with the edges fixed and a uniform load

$$\delta_{max} = \frac{3}{16}(1-\upsilon^2)\frac{PR^4}{Et^3} \tag{2.38-1}$$

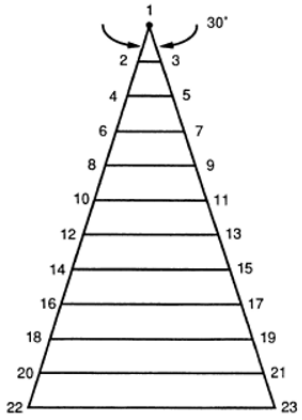
$$\delta_{max} = \frac{3}{16}(1-0.3^2)\frac{(1.0)(11.0)^4}{(10E+6)(0.1)^3} \tag{2.38-2}$$

$$\delta_{max} = 0.25 \text{ in} \tag{2.38-3}$$

MSC Nastran Solution

Using Cyclic Symmetry, a 30^0 segment was modeled as shown.

The resulting displacement at grid point one is δ_z = 0.2526 in.



Comparison of Results

	δ_z (in)
Theory	0.2500
MSC Nastran	0.2526
Difference	1.03%

Conclusion

MSC Nastran results are comparable with theoretical results.

2.39 Normal Modes of a Circular Plate, Cyclic Symmetry

Problem Description

Find the natural frequencies of a free circular plate.

Solution Number

SOL 115

Features Used

- Cyclic Modes

Reference

1. S. Timoshenko, D. H. Young and W. Weaver, Jr., Vibration Problems in Engineering, 4th ed. (New York: John Wiley & Sons, 1947) P. 501

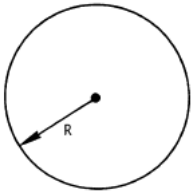
Modeling Techniques Used

- The model details are given below:

  Engineering Data

a	=	11.0 in
h	=	0.10 in
E	=	10.0E+6 lb/in^2
ν	=	0.3
p	=	0.10 (1/g = 0.002588 sec^2/in) = 0.0002588 ib-sec/in^4

Illustrations of the Model



Model File Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v11501.dat	V4801.dat	Normal modes of a circular plate- cyclic symmetry	tpl\verifman\oldverif

Results

Theoretical Solution

Frequency of vibration in rad/sec

$$\omega = \frac{\alpha}{a^2}\sqrt{\frac{D}{\rho h}} \tag{2.39-1}$$

$$D = \frac{Eh^3}{12(1-\upsilon^3)} \tag{2.39-2}$$

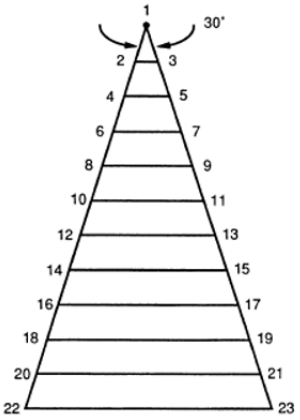
$$D = 915.75 \tag{2.39-3}$$

Table of α and ω in rad/sec for N nodal diameters and S nodal circles

$\frac{\alpha}{\omega}$				
S	**N = 0**	**N = 1**	**N = 2**	**N = 3**
0	-	-	5.251 (258.14)	12.23 (601.23)
1	9.076 (446.18)	20.52 (1008.78)	35.24 (1732.43)	52.91 (2601.11)
2	38.52 (1893.68)	59.86 (2937.86)	-	-

MSC Nastran Solution

Using Cyclic Symmetry, a 30^0 segment was modeled as shown.



Comparison of Results

Following table shows frequency (rad/sec) comparison of MSC Nastran and theoretical results.

	S	**N = 0**	**N = 1**	**N = 2**	**N = 3**
Theory	0			258.14	601.23
MSC Nastran				262.966	603.5286
% Difference				**1.87%**	**0.38%**

	S	N = 0	N = 1	N = 2	N = 3
Theory	1	446.18	1008.78	1732.43	2601.11
MSC Nastran		437.708	1008.877	1688.29	2447.44
% Difference		**-1.90%**	**0.01%**	**-2.55%**	**-5.91%**
Theory	2	1893.68	2937.86		
MSC Nastran		1842.125	2862.61		
% Difference		**-2.72%**	**-2.56%**		

Conclusion

MSC Nastran results are comparable with theoretical results.

2.40 Euler Buckling of a Simply Supported Beam, Cyclic Symmetry

Problem Description

A simply supported beam of constant cross sections is loaded at the ends with an axial force. Determine the critical buckling load.

Solution Number

SOL 116

Features Used

- Cyclic Symmetry Buckling

Reference

1. Stephen P. Timoshenko and James M. Gere, Theory of Elastic Stability, 2nd ed. (New York: McGrawHill Book Co., 1961) P.49.

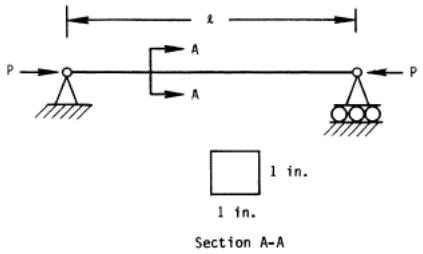
Modeling Techniques Used

- Four equal beams are used in MSC Nastran Solution. The model details are given below:

Engineering Data

I	=	$bh^3/12 = 0.08333\ in^4$
A	=	$bh = 1.00\ in^2$
L	=	100 in
E	=	$10.0E+06\ lb/in^2$
ν	=	0.3

Illustrations of the Model



Model File Name and Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v11601.dat	V7701s.dat	Euler Buckling of a simply supported beam	tpl\verifman\oldverif

Results

Theoretical Solution

$$P_{CR} = \frac{\pi^2 EI}{l^2}$$

$$P_{CR} = \frac{\pi^2 (10.0\times10^6)(0.08333)}{100^2}$$

$$P_{CR} = 822.467 \text{ lb}$$

MSC Nastran Solution

Eigenvalue $\lambda = \frac{P_{CR}}{P}$, solved from $[k - \lambda k_d]\{u\} = 0$

$$P_{applied} = 822.467$$

$$\lambda = 0.99982$$

$$P_{CR} = 0.99982(822.467)$$

$$P_{CR} = 822.319 \text{ lb}$$

Comparison of Results

	Solutions for P_{CR}
Theory	822.467
MSC Nastran	822.319
% Difference	0.018%

Conclusion

The results from MSC Nastran are comparable to the theoretical results.

2.41 Frequency Response of a Beam-Mass Structure using Dihedral Cyclic Symmetry

Problem Description

Find the response at resonance for the model described in v10302, modeling one-half of the model, using cyclic symmetry analysis.

Solution Number

SOL 118

Features Used

- Cyclic Frequency Response

Reference

1. J. P. Den Hartog, Mechanical Vibrations, 4th ed. (New York: McGraw-Hill Book Company, Inc., 1956) pp. 429-431.

Modeling Techniques Used

1. The model of V10302 is cut through the center grid point, with the plane of symmetry being normal to the bar centerline. Only the right-hand half is modeled.
2. The only elements entirely in the plane of symmetry are the mass elements. Their values are halved, because the mirrored component also has these values implicitly. The dihedral symmetry option is selected.
3. All loads are applied on the plane of symmetry. Full loads are placed on the right hand side, and none on the left side.
4. Structural damping that provides critical damping at resonance is used. This required a value of twice the fraction of critical damping, as is input on the card PARAM, G, 2.0.
5. The loads on each degree of freedom are proportional to the stiffness in that direction, to nondimensionalize the problem (see DAREA cards)

6. The default format for frequency response analysis output is the Cartesian form of the complex number (“RE_IMAG”). By requesting (“PHASE”) on the DISP case control card a polar form (“MAG_PHASE”) is output instead. The SORT2 option on the DISP card produces a more compact output.

- The model details are given below:

  Engineering Data

  Beam properties:

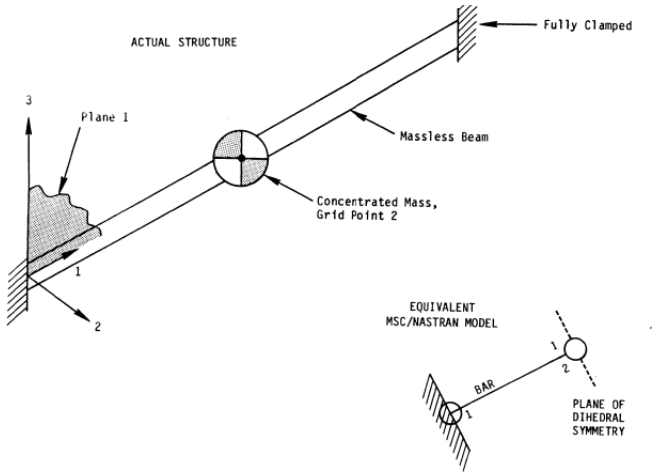
A	=	1.0 (Extensional Area)
I_1	=	2. 0 (Bending moment of inertia in 1-3 plane)
I_2	=	1.0 (Bending Moment of inertia in 1-2 plane)
J	=	1.25 (Moment of inertia for Torsion)
E	=	1.0 (Young's Modulus)
G	=	1.0 (Shear Modulus)
L	=	1.0 (Length)

Mass properties

M_1	=	1.0 (Axial)
M_2, M_3	=	64.0
I_{11}	=	1.0
I_{22}, I_{33}	=	16.0

Note: Consistent, hypothetical units are used.

Illustrations of the Model



Model File Name and Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v11801.dat	V7801s.dat	Frequency response of a beam-mass structure using Dihedral cyclic symmetry	tpl\verifman\oldverif

Results

Theoretical Solution

The response of a single degree of freedom oscillator at resonance is given in equation 2.28a (page 49 of the reference) as

$$X_0(\omega = \omega_n) = \frac{P_0/k}{2c/c_c} \qquad (2.41\text{-}1)$$

The phase angle between applied force and displacement is plotted on page 51 and shows a 90^0 value for all values of damping. The symbols are defined as:

X_0 = Magnitude of displacement

P_0 = Magnitude of applied load

k = Stiffness of oscillator

c = Viscous damping coefficient

c_c = Viscous damping required for critical damping

MSC Nastran Solution

MSC Nastran uses a different sign convention for the phase angle than does the reference. For example, in the reference, the phase angle for a single degree of freedom system starts at a small positive angle at low frequency, passes through 90^0 at resonance, and approaches 180^0 at high frequency. In MSC Nastran the Phase angle traverses the range 360^0 through 270^0, approaching 180^0.

As shown in Problem v10302, each degree of freedom of these single degree of freedom problems has a different stiffness. A load proportional to stiffness is used to nondimensionalize the solution. (see the definition of the static deflection X_s on page 49 of the reference and the nondimensional quantity X/X_s.)

The expected solutions should all have the magnitude of one-half and a phase of 270^0.

Conclusion

The results from MSC Nastran are comparable to the theoretical results.

2.42 Stress Wave Propagation

Problem Description

A rod of uniform cross section is fixed at one end and a constant force is suddenly applied to its free end. A stress wave propagates along the length of the rod. Analyze this wave propagation phenomenon by SOL 129 for transient response analysis.

Solution Number

SOL 129

Features Used

- Transient Response

Reference

1. Stephen P. Timoshenko and J. N. Goodier, Theory of Elastic Stability, 2nd ed. (New York: McGraw-Hill Book Co. 1951), pp. 492-496.
2. R. C. Juvinall, Engineering Consideration of Stress, Strain and Strength, McGraw-Hill, New York, 1967, pp. 185-188.

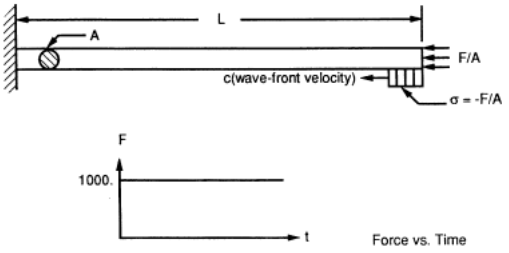
Modeling Techniques Used

- The model details are given below:

Engineering Data

L	=	100.0 m (Rod length)
A	=	1.0 m^2, (Cross-sectional area of the rod)
E	=	10^3 N/m^2, (Young's Modulus)
ν	=	0.3, (Poisson's ratio)
ρ	=	0.1 kg/m^3, (Mass density)
c	=	$\sqrt{E/\rho}$ = 100.0 m/s = L/s, (Stress wave velocity)

Illustrations of the Model



Model File Name and Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v12901.dat	V9901s.dat	Stress wave propagation	tpl\verifman\oldverif

Results

Theoretical Solution

Transient responses at two points of interest, the free end and the fixed end points are analyzed.

Due to constant force F, applied at the free end, the stress at the free end $\sigma = -F/A$ is constant at all times. Before the wave front first reaches the fixed end, the stress at the fixed end is zero. As soon as the wave reaches the fixed end, it is reflected and produces a stress of 2σ at the fixed end. The stress remains unchanged until the wave front returns to this point. The compressive wave leaves the fixed end and upon reaching the free end it is reflected as a tensile stress wave. When this tensile wave arrives at the fixed end and is reflected there, the stress at this point becomes zero. The zero stress is again unchanged for a complete cycle of the stress wave motion.

The displacement-time history at the free end can be expressed, in general, as:

$$u(t) = \int_0^L \frac{\sigma(x,t)}{E} dx$$

Or, expressing $u(t)$ separately for each time interval,

$$u(t) = \frac{-FL}{AE} t \quad \text{for } 0 \le t \le 2$$

$$u(t) = u(2) + \frac{FL}{AE}(t-2) \quad \text{for } 2 \le t < 4$$

etc.

This shows that the displacement for the free end is a linear function of time in each time interval.

MSC Nastran Solution

Using twenty CONROD elements, the rod is loaded axially at its free end with a step forcing function. The upper bound stability limit for the time increment is given by

$$\Delta t = \frac{1}{4.(\omega_n)_{max}}$$

Where, $(\omega_n)_{max}$ is the highest natural frequency in cycles/sec of all the interested modes of the corresponding model. Introduction of small damping in the rod elements is recommended to reduce the high frequency oscillations in the time history plots.

Comparison of Results

The stress time history plots given in Figure 2.42-1 and Figure 2.42-2 for the free end and fixed end, respectively show that the MSC Nastran results are consistent with the theoretical ones in their global trends and characters. The oscillation about the exact solution may be attributed to higher natural frequency modes.

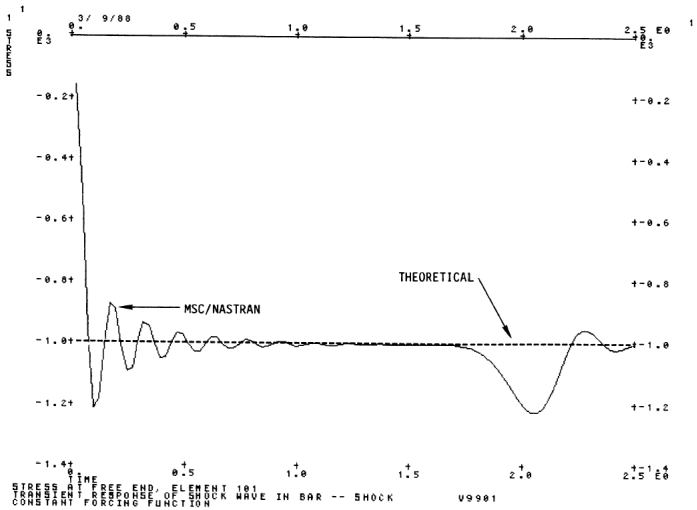


Figure 2.42-1 Comparison of Free End Stress Time History

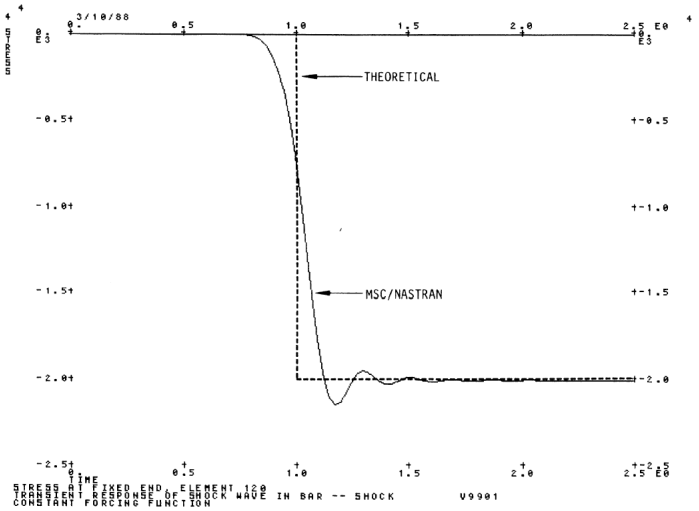


Figure 2.42-2 Comparison of Fixed End Stress Time History

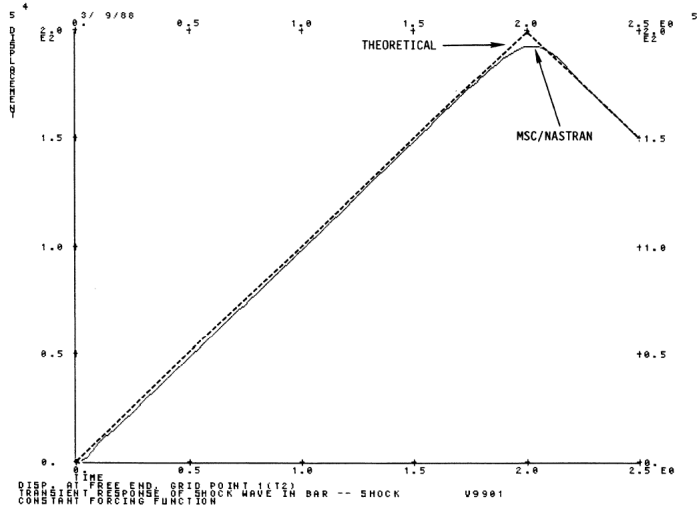


Figure 2.42-3 Comparison of Free End Displacement Time History

Conclusion

The displacement-time plot for the free end (Figure 2.42-3) Shows a close agreement with the theoretical solution.

2.43 Impact Analysis

Problem Description

A rod with fixed end is struck by a moving mass at the other end. Let v_0 be the initial velocity of the mass. Consider the mass body as absolutely rigid. The velocity of particles at the end of the rod at the instant of impact (t = 0) is v_0. Determine the time history of stress and displacement at various cross sections of the rod during the impact. Find the duration of the impact and the maximum stress in the rod for various values of mass ratios.

Solution Number

SOL 129

Features Used

- Transient Response

Reference

1. Stephen P. Timoshenko and J. N. Goodier, Theory of Elastic Stability, 2nd ed. (New York: McGraw-Hill Book Co. 1951), pp. 497-504.
2. April 1986 Application Note, Application Manual Section.

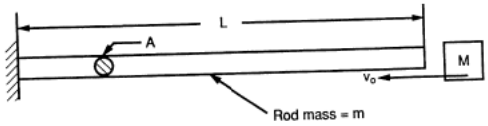
Modeling Techniques Used

- The model details are given below:

Engineering Data

L	=	100.0 m (Rod length)
A	=	1.0 m^2, (Cross-sectional area of the rod)
E	=	10^3 N/m^2, (Young's Modulus)
ν	=	0.3, (Poisson's ratio)
ρ	=	0.1 kg/m^3, (Mass density)
m	=	ρAL = 10.0 kg, (Rod mass)

Illustrations of the Model



Model File Name and Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v12902.dat	V9902s.dat	Impact analysis	tpl\verifman\oldverif

Results

MSC Nastran Solution

The rod is modeled with twenty CONRODs and a GAP element. One end of the rod is constrained completely and the other is left free axially. One CONM2 is used to represent the mass body, and the mass value is varied for various ratios of M to m. A gap element connects the CONM2 and the free end of the rod with a zero initial opening. An initial velocity of v_0 = -0.1 m/sec is specified for both the CONM2 and the free end grid point using TIC cards. A time step is chosen so that twenty vibration modes of the rod model can be adequately represented. A small damping (0.4% at resonance) is introduced in the rod elements to smooth out the high frequency oscillations. The consistent mass matrix option (PARAM, COUPMASS) is recommended for more stable responses.

Comparison of Results

The instant when the stress at the free end (or force in the gap element) vanishes signifies the end of the impact. The durations of the impact for four distinct values of M/m are determined from MSC Nastran analyses and compared with the calculations of Saint-Venant in the following table.

Table 2.43-1 Duration of impact (sec.)

M/m	1	2	4	6
Theory	3.068	4.708	5.900	7.419
MSC Nastran	3.065	4.708	6.700*	7.420
% Difference	0.1%	0.00%	13.56%	0.01%

*The discrepancy is due to a second contact on the rebound which is not accounted for in the theory as explained in the references.

Conclusion

MSC Nastran results are comparable with theoretical results.

2.44 Flutter of a Two-Dimensional Airfoil

Problem Description

A two-dimensional airfoil is mounted on bending and torsion springs that are aft of the aerodynamic center as shown in the figure below. A critical velocity will cause either a static instability (torsional divergence) or an oscillatory instability (flutter). Find both the divergence and flutter speeds for the system.

Solution Number

SOL 145

Features Used

- Flutter

Reference

1. Raymond L. Bisplinghoff, Holt Ashley, and Robert L. Halfman, Aeroelasticity (Reading: Addison-Wesley Publishing Co., Inc., 1955) pp. 538-541

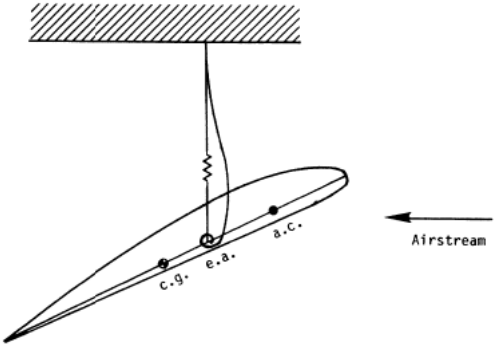
Modeling Techniques Used

- Large values of reduced frequency k may be chosen to illustrate the low speed behavior of the frequencies and artificial damping. Moderate values of k define the flutter speed and very low values of k lead to the divergence speed. The model details are given below:

Engineering Data

Mass Ratio	μ	=	$m/(\pi\rho b^2)$ = 20.0
Elastic Axis at 40%	a	=	-0.2
Centre of Gravity at 45%	x_α	=	0.1
Aerodynamic centre at 25%			
Dimensionless radius of gyration	r_a	=	0.5
Uncoupled bending frequency	ω_h	=	10 rad/sec
Uncoupled torsion frequency	ω_α	=	25 rad/sec

Illustrations of the Model



Model File Name and Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v14501q.dat	V7501q.dat	Flutter of two dimensional Airfoil	tpl\verifman\oldverif

Results

Theoretical Solution

The data correspond to Case (r) on page 538 of the reference. The divergence speed is given there as

$$U_D/b\omega_\alpha = 2.88 \tag{2.44-1}$$

Using an airfoil chord of c = 2b = 6.0 ft., the divergence speed is

$$U_D = 216 \text{ fps} \tag{2.44-2}$$

The flutter speed is found from the graph (r) on page 541 of the reference. For $X_\alpha = 0.1$ and frequency ratio $\omega_h/\omega_\theta = 0.4$, we find

$$U_F/b\omega_\alpha = 2.25 \tag{2.44-3}$$

from which

$$U_F = 169 \text{ fps} \tag{2.44-4}$$

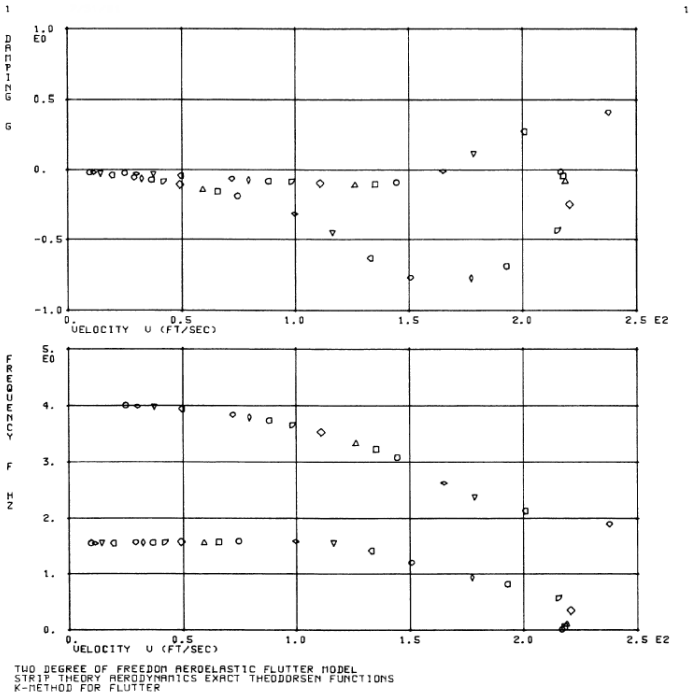
MSC Nastran Solution

Additional dimensional data are required and derived as follows:

Sea level density	ρ	=	0.002378 slugs/cu.ft.
Mass	m	=	$\pi\rho b^2\mu$ = 1.3447 slugs

Static unbalance	s_α	=	mbx_α = 0.40342 slugs
Moment of inertia	i_a	=	$mb^2r_\alpha^2$ = 3.0256 slug-ft^2
Bending stiffness	k_h	=	$m\omega_h^2$ = 134.47 lbs/ft
Torsional stiffness	k_α	=	$mi_\alpha\omega_\alpha^2$ = 1891.0 ft-lbs/rad

The solution is carried out by the k-method using the exact Theodorsen function. The plotted output is shown in the following figure. Interpolation of the printed output for the flutter speed at zero damping gives V_F = 166 fps; extrapolation of the printed output for the divergence speed at zero damping gives V_D = 216 fps.



Comparison of Results

	V_F	V_D
Theory	169	216
MSC Nastran	166	216
% Difference	1.8%	0.0%

Conclusion

The results from MSC Nastran are comparable to the theoretical results.

Main Index

2.45 Radiation Exchange Between Two Panels

Problem Description

Two Parallel 'grey' planes have emissivities of 0.8 and 0.7 and are maintained at 500°F and 1000°F respectively. Assuming all energy radiated from each plate is irradiated onto the other (View factors = 1.0) Calculate the net radiation exchange.

Solution Number

SOL 153

Features Used

- Nonlinear steady state Heat transfer

Reference

1. Alan J., Chapman, Heat Transfer, 3rd ed. (New York: Macmillan Publishing Co., Inc., 1974) P. 447, ex. 11.4

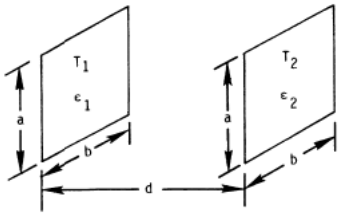
Modeling Techniques Used

- The model details are given below:

  Engineering Data

a = b	=	1 ft
d	=	2 ft
T_1	=	500^0 F
T_2	=	1000^0 F
ε_1	=	0.7
ε_2	=	0.8
T_{ABS}	=	$460.0^{\ 0}$ R
σ	=	1.714E-9 Btu/hr-ft^{2}-^{0} R^{4}

Illustrations of the Model



Model File Name and Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v15301s.dat	V7401s.dat	Radiation exchange between two panels	tpl\verifman\oldverif

Results

Theoretical Solution

The net radiation exchange between two "gray" surfaces is given by,

$$q = \sigma \frac{T_1^4 - T_2^4}{(1/\varepsilon_1) + (1/\varepsilon_2) - 1} \qquad (2.45\text{-}1)$$

Solving this equation yields,

$q = 3772.347 \ \text{Btu/hr-ft}^2$

Note: This is the exact answer obtained when above values are used in the above equation. Text example contains some round-off error.

MSC Nastran Solution

Total radiation exchange is calculated to be

q = 3772.347 Btu/hr

Since the model plates are one square foot each, the flux per square foot is

q = 3772.347 Btu/hr-ft^2

Conclusion

The results from MSC Nastran are comparable to the theoretical results.

2.46 Transient Temperature Distribution in a Slab

Problem Description

A 10-inch-thick slab, initially at a uniform temperature of 180° F, suddenly has its end surfaces reduced to 60° F. Find the temperature at a plane four inches from the end surface after an elapsed time of two hours.

Solution Number

SOL 159

Features Used

- Transient Heat Transfer

Reference

1. Alan J., Chapman, Heat Transfer, 3rd ed. (New York: Macmillan Publishing Co., Inc., 1974) P. 126, ex. 4.2.

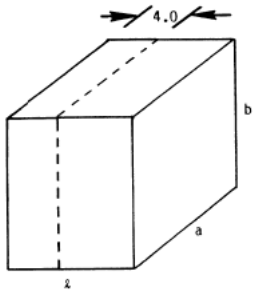
Modeling Techniques Used

- The model details are given below:

  Engineering Data

a = b	=	12 in
l	=	10 in
k	=	0.4 Btu/hr-ft-^{0}F
ρ	=	100.0 lb_m/ft^2
C_n	=	0.2 Btu/lb_m/^{0}F
α	=	$\frac{k}{\rho C_p}$ = 0.2 ft^2 /hr
$T_{initial}$	=	180^0F
$T_{ends}\|_{time\ =\ 0}$	=	60^0F

Illustrations of the Model



Model File Name and Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v15901s.dat	V8901s.dat	Transient temperature distribution in a slab	tpl\verifman\oldverif

Results

Theoretical Solution

For the given conditions, from a graph in the reference it can be determined that

$$\frac{T - T_s}{T_i - T_s} = 0.685 \qquad (2.46\text{-}1)$$

where, T_i = Initial temperature distribution

T_s = Applied surface temperature at time = 0

Thus, T = (0.685)(180-60) + 60 = 142.20 ^{0}F

MSC Nastran Solution

Using HEXA elements to divide the slab into five two inch thick sections, use a time step = 0.02 hr. For a plane four inches from the end surface, as shown in the figure, at time = 2 hours, the result is:

T = 142.05 ^{o}F

Comparison of Results

	Temperature
Theory	142.2
MSC Nastran	142.05
% Difference	-0.10549%

Conclusion

MSC Nastran results are comparable with theoretical results.

2.47 Transient Temperature Distribution in a Slab with Convective Effects

Problem Description

A same slab as in problem v15901s (10 inch thick, initially at 180°F) has a fluid at 60°F placed in contact with its end surfaces at time equal 0.0. Assuming a convective film coefficient of 4.0 Btu/hr-ft-F, find the temperature at a depth of one inch after 10 hours have passed.

Solution Number

SOL 159

Features Used

- Transient Heat Transfer

Reference

1. Alan J., Chapman, Heat Transfer, 3rd ed. (New York: Macmillan Publishing Co., Inc., 1974) P. 126, ex. 4.3.

Modeling Techniques Used

- The model details are given below:

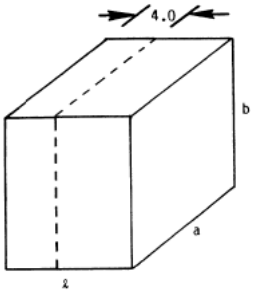
Engineering Data

a = b	=	12 in
l	=	10 in
k	=	0.4 Btu/hr-ft-^{0}F
ρ	=	100.0 lb_m/ft^2
C_n	=	0.2 $Btu/lb_m/^0F$
α	=	$\frac{k}{\rho C_p}$ = 0.2 ft^2 /hr

$T_{initial}$ = 180^0F

$T_{ends}|_{time = 0}$ = 60^0F

Illustrations of the Model



Model File Name and Location

Model Name	Reference Deck from Old Manual	Model Description	Location
v15902s.dat	V8902s.dat	Transient temperature distribution in a slab with convective effects	tpl\verifman\oldverif

Results

Theoretical Solution

For the given conditions, it can be graphically determined that

$$\frac{T - T_f}{T_i - T_f} = 0.13 \qquad (2.47\text{-}1)$$

where, T_f = Fluid temperature

T_i = Initial temperature distribution

Thus, T = (0.13)(180-60) + 60 = 75.6 ^{0}F

at depth of one inch after an elapsed time of 10 hours.

MSC Nastran Solution

Using HEXA elements to divide the slab into ten one inch thick sections, use a time step = 0.05 hr. For a plane one inch from the end surface, as shown in the figure, at time = 10 hours, the result is:

T = 71.89 ^{o}F

Comparison of Results

	Temperature
Theory	75.6
MSC Nastran	71.89
% Difference	-4.90741%

Conclusion

MSC Nastran results are comparable with theoretical results.

2.48 Large Deflection of Uniformly Loaded Plate

Problem Description

A square plate with clamped edges is loaded uniformly such that the center deflection is more than the plate thickness. Considering large deflection (small strain theory) find the deflection of the center of the plate.

Solution Number

SOL 400

Features Used

- Geometric Nonlinearity

Reference

1. S. Timoshenko and S. Woinowsky - Krieger, Theory of plates and Shells (New York: McGraw-Hill Book Company, Inc., 1959), P. 422.

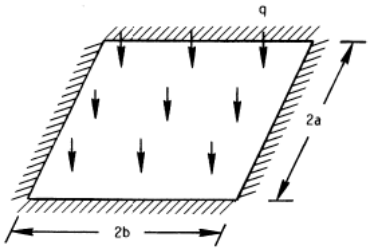
Modeling Techniques Used

- The model details are given below:

Engineering Data

a = b	=	100
h	=	1
E	=	2.0E+11
ν	=	0.3
q	=	2.0E+04
$D = \frac{Eh^3}{12(1-\nu^2)}$	=	1.83×10^{10}

Illustrations of the Model

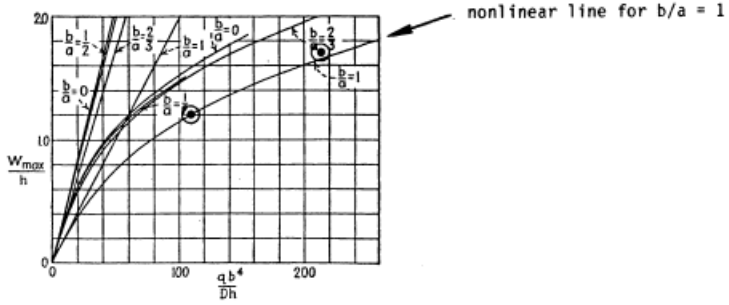


Model File Name and Location

Model Name	Reference Deck from Old Manual	Model Description	Location
V40002.dat	V6401.dat	Large deflection of uniformly loaded plate	tpl\verifman\oldverif

Results

Theoretical Solution



For $qb^4/Dh =$ 109.3 , $W_{max}/h =$ 1.20

For $qb^4/Dh =$ 218.6 , $W_{max}/h =$ 1.66

Comparison of Results

	W_{max}/h	
qb^4/Dh	**109.3**	**218.6**
Theory	1.2	1.66
MSC Nastran	1.26	1.75
% Difference	5.00%	5.42%

Conclusion

MSC Nastran results are comparable with theoretical results.

2.49 Large Deflection of a Beam

Problem Description

Find the displaced position of a post-buckled column, fixed at one end with an applied axial load at other end.

Solution Number

SOL 400

Features Used

- Geometric Nonlinearity

Reference

1. Stephen P. Timoshenko and James M. Gere, "Theory of Elastic Stability" (New York: McGraw-Hill Book Company, Inc., 1961), P. 48.

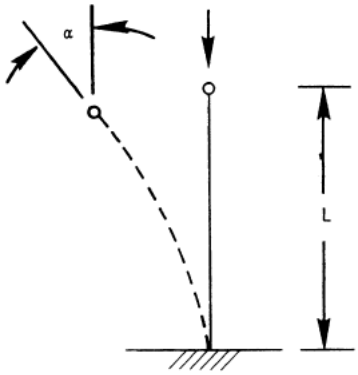
Modeling Techniques Used

- The model details are given below:

  Engineering Data

L	=	12.0
A	=	0.15
$I_1 = I_2$	=	2.8125E-04
E	=	2.0E+07
G	=	1.0E+07

Illustrations of the Model



Model File Name and Location

Model Name	Reference Deck from Old Manual	Model Description	Location
V40003.dat	V6402s.dat	Large deflection of a beam	tpl\verifman\oldverif

Results

Theoretical Solution

The critical Euler buckling load is found using equation 2-4 on page 48 of the reference.

$$P_{CR} = \frac{\pi^2 EI}{4L^2} \tag{2.49-1}$$

$$P_{CR} = \frac{\pi^2 (2.0 \times 10^7)(2.8125 \times 10^{-4})}{4(12)^2} \tag{2.49-2}$$

$$P_{CR} = 96.38286 \tag{2.49-3}$$

For a load greater than P_{CR}, the following tip rotations can be determined:

P/P_{CR}	α (deg)	α (rad)
1.152	60^0	1.0472
1.293	80^0	1.3963

MSC Nastran Solution

The column is modeled using 12 BEAMs, each of unit length. The final rotations for a converged solution are:

		R_3	
Subcase	Load ID	(rad)	(deg)
14	**200**	**1.04385**	**59.808**
26	300	1.3919	79.750

Comparison of Results

	α (deg)	
P/P_{CR}	1.152	1.293
Theory	60^0	80^0
MSC Nastran	59.808	79.750
% Difference	0.32%	0.31%

Conclusion

MSC Nastran results are comparable with theoretical results.

Section 3: Rotordynamics Problems



Main Index

3.1 Flexible Rotor Critical Speed and Mode Shape

Problem Description

Complex eigenvalue analysis for Rotors.

Solution Number

SOL 107

Features Used

- The analysis is performed in fixed coordinate systems. For the purpose of comparison, same rotor is modeled using 1-D elements (ROTORG), axisymmetric harmonic elements (ROTORAX), and solid elements (ROTOR).

Reference

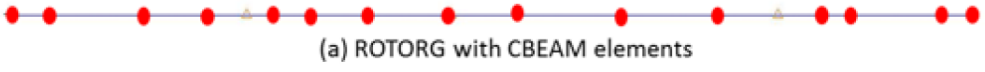
1. Geradin M. and Kill N., (1984), "A new Approach to Finite Element Modeling of Flexible Rotors." Engineering Computations, Vol. 1, 52-64.

Modeling Techniques Used

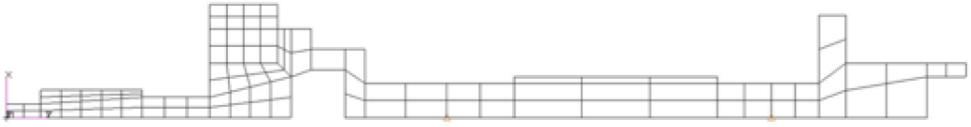
- Apex/Patran was used to create the Finite Element model. The same rotors are modeled by 1D element CBEAM for ROTORG, by 2D element CQUADX for ROTORAX, and by 3D element CHEXA/CPENTA for ROTOR. The bearing s are modeled by CBUSH:

```
CBUSH    201      211      50001                                        0
CBUSH    202      212      50001                                        1
$
PBUSH    211       K       2.627+7 2.627+7
CORD2R   1         0         0.       0.       0.       0.       0.         1.
          1.       1.        0.
PBUSH    212       K       1.751+7
```

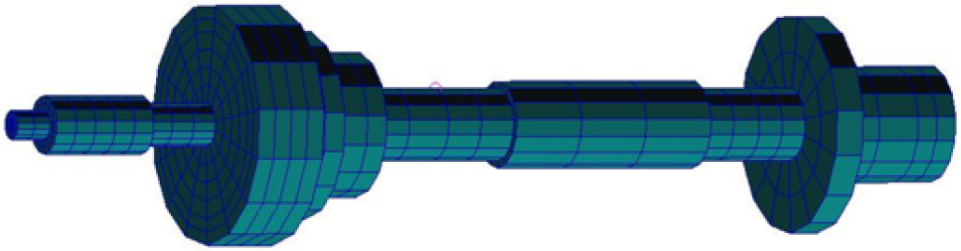
Illustrations of the Model



(a) ROTORG with CBEAM elements



(b) ROTORAX with CQUADX elements



(c) ROTOR with CHEXA elements

Model File Name and Location

Model File Name	Location
rug_7_9.dat (1D); rug_7_11.dat (2D); rug_7_12.dat (3D)	qa\nastran\tpl\verifman\rotor\

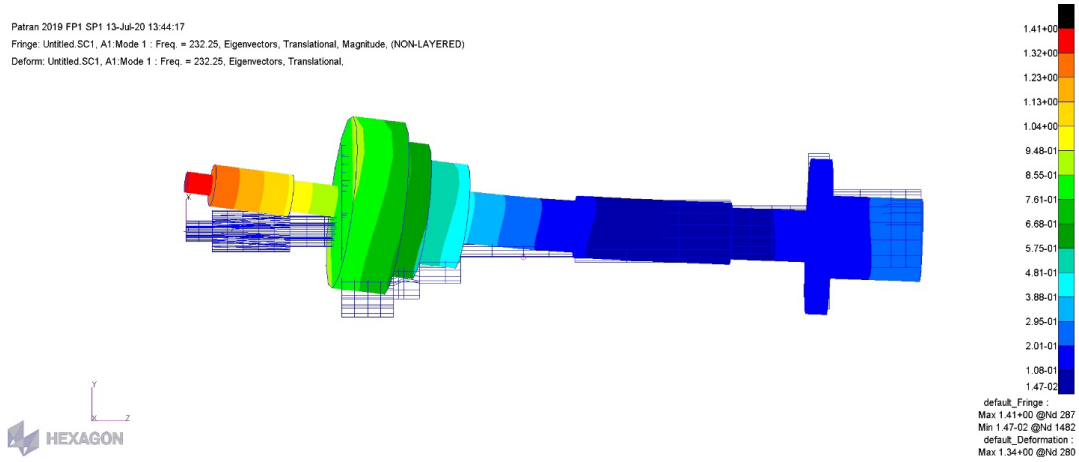
Results

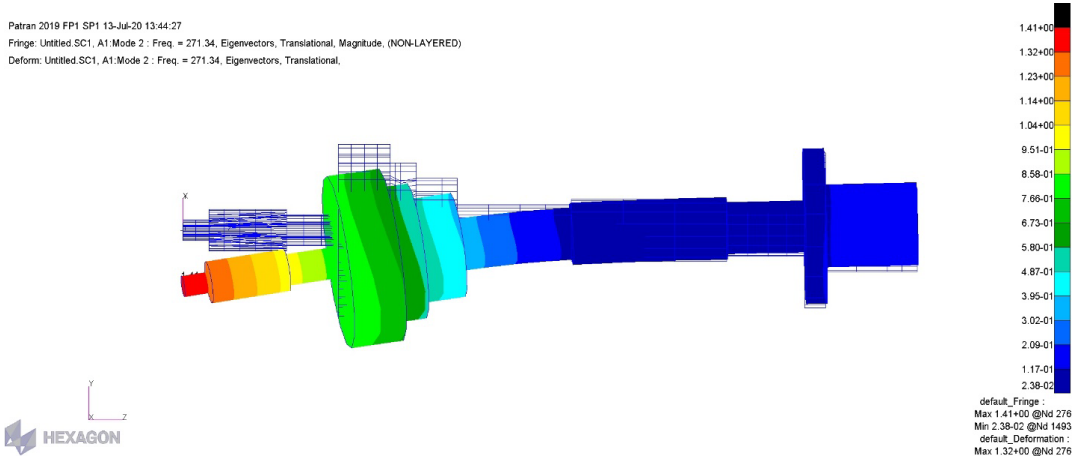
Eigenvalue comparison to the paper [1]:

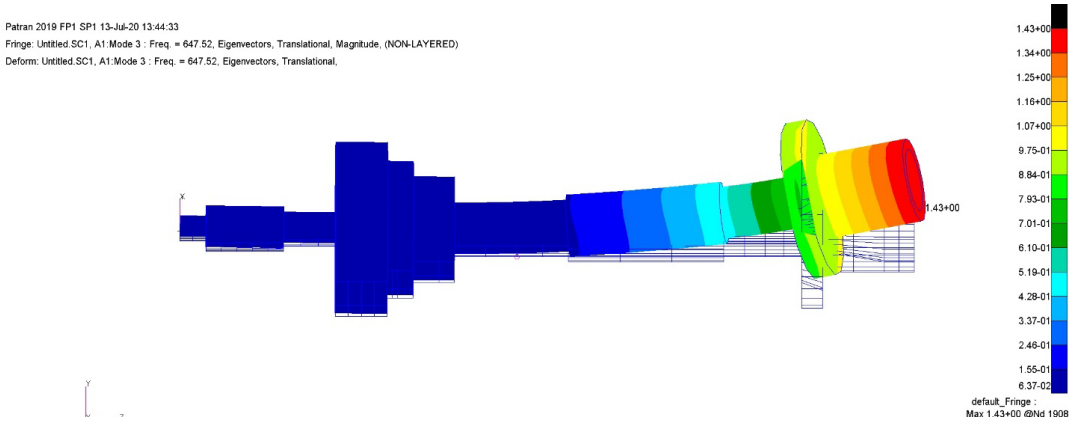
Mode Type	MSC Nastran			Paper [1]	
	Frequencies (Hz)			Frequencies (Hz)	
	Beam Model	Axi Model	Solid Model	Beam Model	Axi Model
1st Backward	231.6	231.1	232.3	231.3	228.7
1st Forward	274.1	270.8	271.3	275.8	272.3
2nd Backward	655.4	643.1	647.5	666.6	656.4
2nd Forward	793.6	772.7	776.0	806.3	794.1

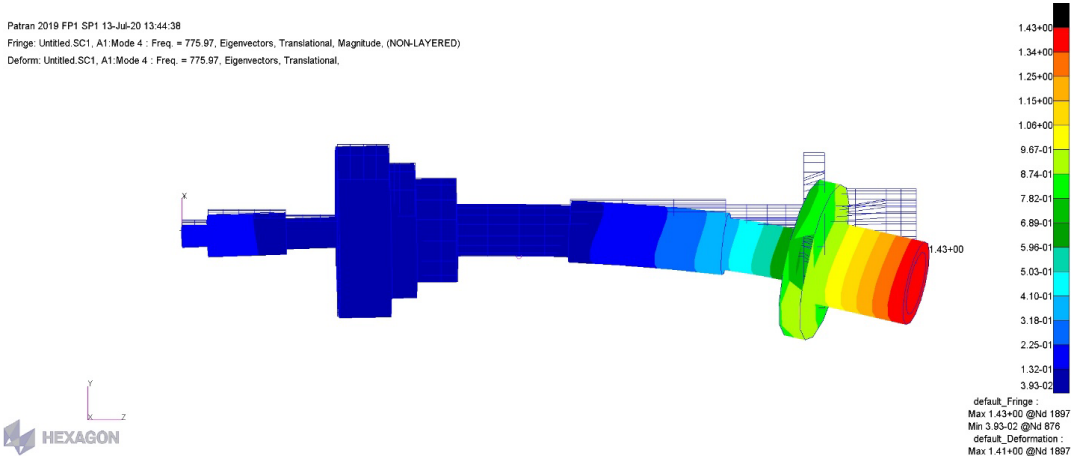
Mode Type	MSC Nastran			Paper [1]	
	Frequencies (Hz)			Frequencies (Hz)	
	Beam Model	Axi Model	Solid Model	Beam Model	Axi Model
Torsional	1006.1		1004.3		
3rd Backward	1025.4	1026.6	1037.2	995.3	963.6
3rd Forward	1580.2	1481.9	1456.0	1543.1	1479.4

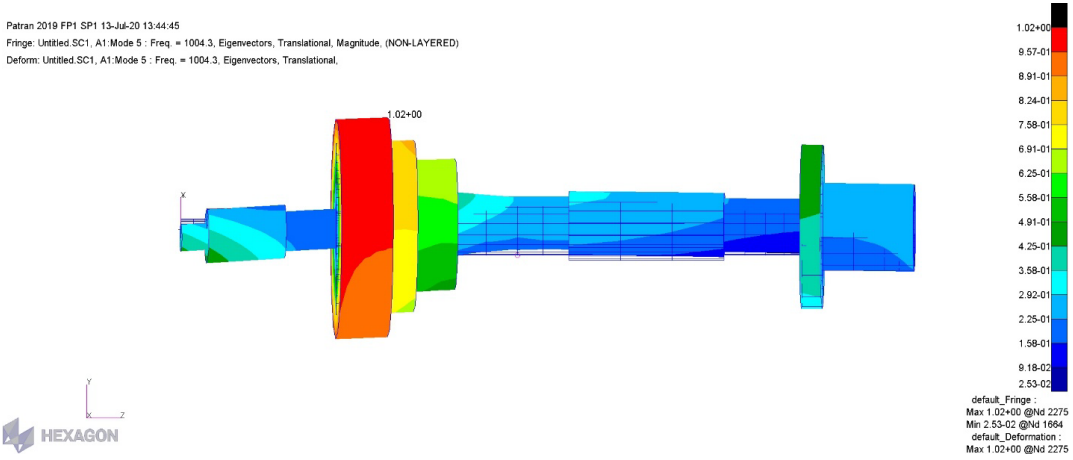
Mode shapes of solid rotor



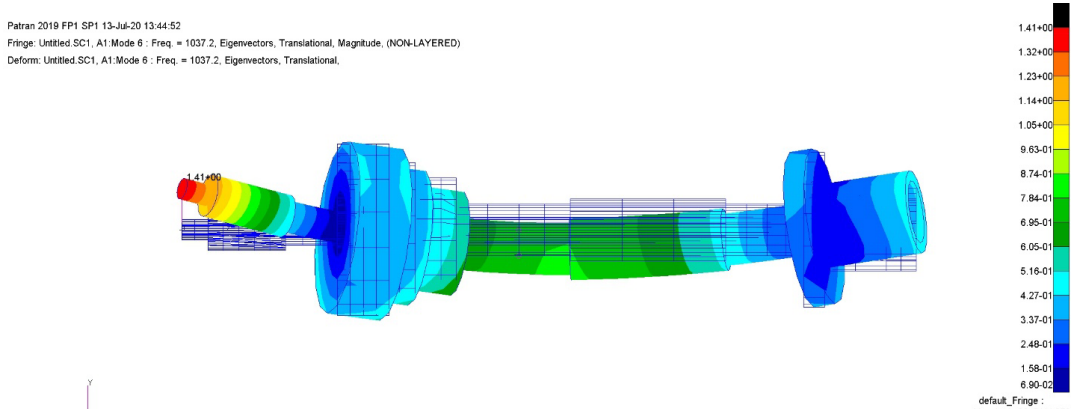


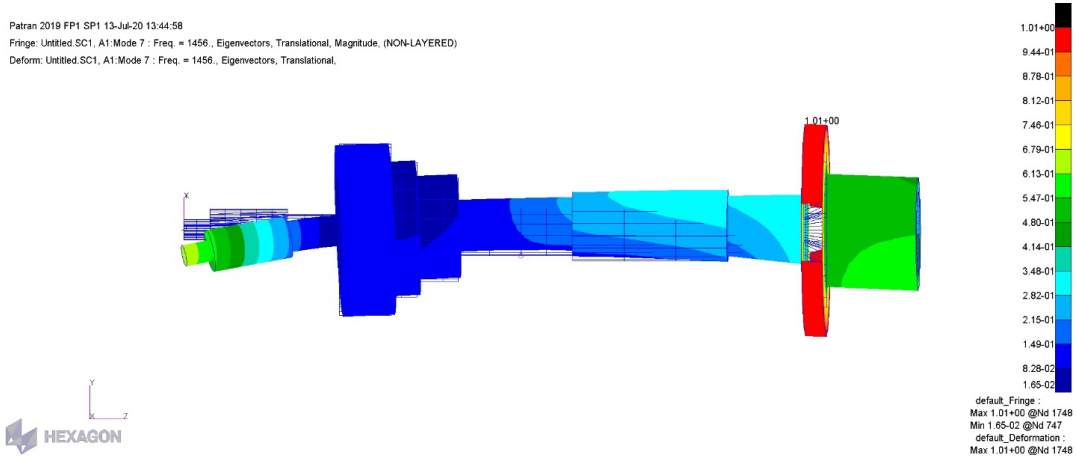

Patran 2019 FP1 SP1 13-Jul-20 13:44:33
Fringe: Untitled.SC1, A1:Mode 3 : Freq. = 647.52, Eigenvectors, Translational, Magnitude, (NON-LAYERED)
Deform: Untitled.SC1, A1:Mode 3 : Freq. = 647.52, Eigenvectors, Translational,
1.43+00
1.43+00
1.34+00
1.25+00
1.16+00
1.07+00
9.75-01
8.84-01
7.93-01
7.01-01
6.10-01
5.19-01
4.28-01
3.37-01
2.46-01
1.55-01
6.37-02
default_Fringe :
Max 1.43+00 @Nd 1908


Patran 2019 FP1 SP1 13-Jul-20 13:44:38
Fringe: Untitled.SC1, A1:Mode 4 : Freq. = 775.97, Eigenvectors, Translational, Magnitude, (NON-LAYERED)
Deform: Untitled.SC1, A1:Mode 4 : Freq. = 775.97, Eigenvectors, Translational,
1.43+00
1.43+00
1.34+00
1.25+00
1.15+00
1.06+00
9.67-01
8.74-01
7.82-01
6.89-01
5.96-01
5.03-01
4.10-01
3.18-01
2.25-01
1.32-01
3.93-02
default_Fringe :
Max 1.43+00 @Nd 1897
Min 3.93-02 @Nd 876
default_Deformation :
Max 1.41+00 @Nd 1897
HEXAGON


Patran 2019 FP1 SP1 13-Jul-20 13:44:45
Fringe: Untitled.SC1, A1:Mode 5 : Freq. = 1004.3, Eigenvectors, Translational, Magnitude, (NON-LAYERED)
Deform: Untitled.SC1, A1:Mode 5 : Freq. = 1004.3, Eigenvectors, Translational,
1.02+00
1.02+00
9.57-01
8.91-01
8.24-01
7.58-01
6.91-01
6.25-01
5.58-01
4.91-01
4.25-01
3.58-01
2.92-01
2.25-01
1.58-01
9.18-02
2.53-02
default_Fringe :
Max 1.02+00 @Nd 2275
Min 2.53-02 @Nd 1664
default_Deformation :
Max 1.02+00 @Nd 2275
HEXAGON

Main Index





Conclusion

The results obtained here compare well with those obtained by Geradin et al. (Geradin and Kill, 1984).

3.2 Jeffcott Rotor Eigenvalues

Problem Description

Solve a Jeffcott rotor eigenvalues and plot Campbell Diagram.

Solution Number

SOL 107

Features Used

- Massless shaft flexible rotor with lumped mass from disk - Jeffcott Rotor
- Rigid supported bearing - Pinned at bearing location
- Campbell Diagram - Spin speed vs Natural Frequency

Reference

1. Friswell, M. I., Penny, J. E. T., Garvey, S. D. and Lees, A. W. (2010). Dynamics of rotating machines. Cambridge University Press, P114~116.

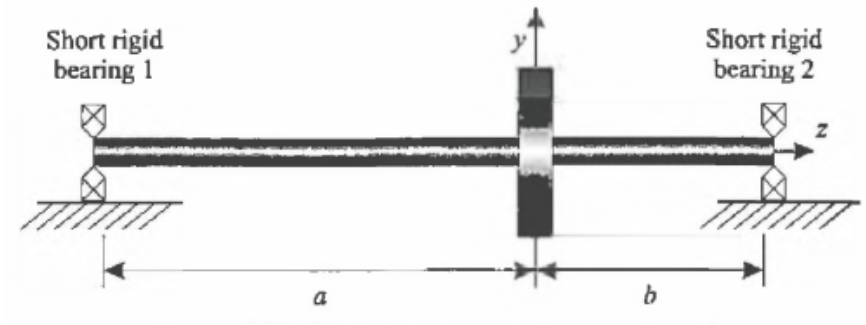
Modeling Techniques Used

- Apex/Patran was used to create the Finite Element 1D rotor model (ROTORG): Beam element for shaft (CBEAM), lumped mass/inertia for disk (CONM2); displacement constrains at two ends (SPC1).

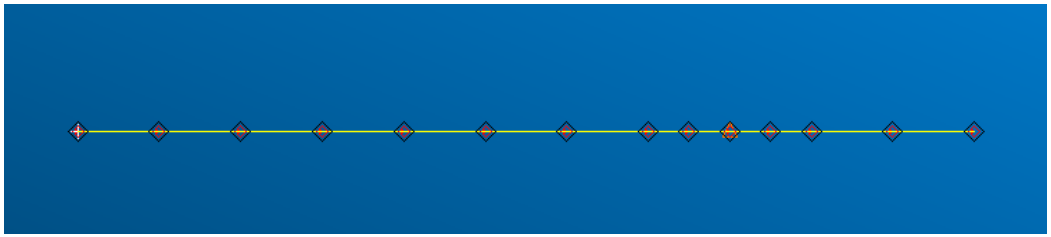
Illustrations of the Model

The flexible rotor with a single disk [1] is shown as following:

Example - A 38 mm diameter solid shaft is supported in self aligning bearings 1.1 m apart. A disk, 650 mm in diameter and 100 mm in thickness is shrunk onto the shaft 0.8 m from one bearing. The material properties of the shaft and disk are density ρ= 7810 kg/m^3 and modulus of elasticity E = 211 GPa.



The finite element models used in the analysis is shown in the following figure:



Massless shaft material:

```
$
$ Description of Material: rho=0
MAT1    1       2.11e11         .3      0.
$
```

Lumped mass added to Grid 10:

```
$ Elements and Element Properties: CONM2
$ Lumped mass and inertia
CONM2   100     10      0       259.2   0.      0.      0.
        7.06    0.      7.06    0.      0.      13.69
```

Pinned ends (bearings):

```
$ $ fix bearing node
SPC1     1       123      1
SPC1     1       12       14
```

Rotor define:

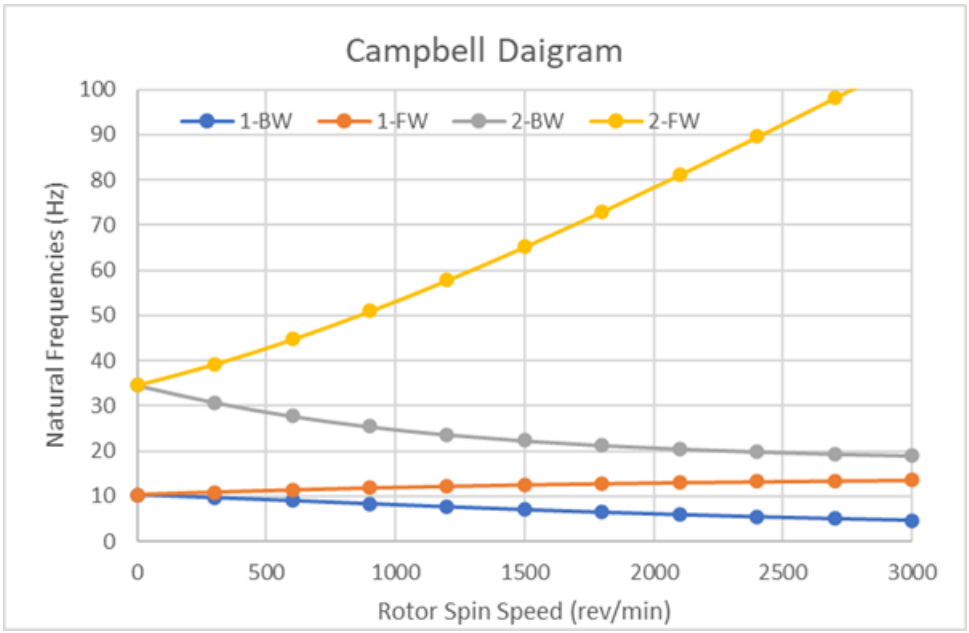
```
$ Rotor Dynamics data for RGYRO 1
ROTORG   4       1      THRU      14
RSPINR   4       9       10      RPM     1.
         0.
RGYRO    1       ASYNC      4      RPM                  1000.0
```

Model File Name and Location

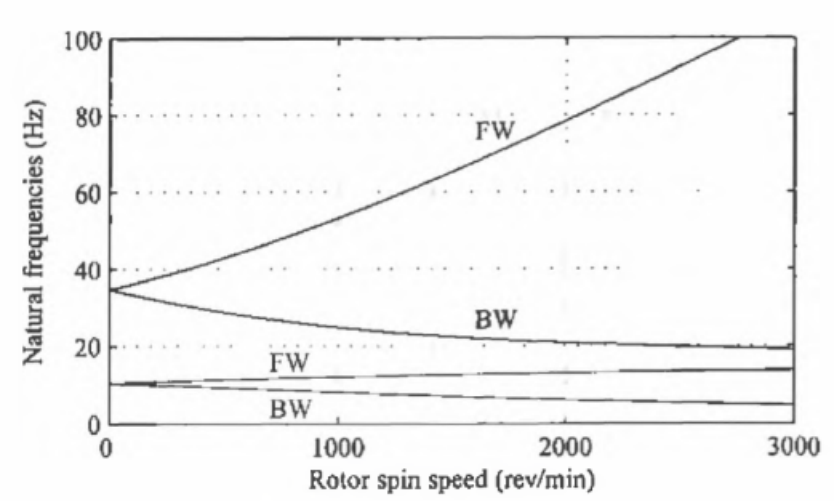
Model File Name	Location
friswell_3-9-1.dat	qa\nastran\tpl\verifman\rotor\

Results

The Campbell Diagram from MSC Nastran's output is shown as:



The analytical solution of Campbell Diagram from [1] is:



The comparison of MSC Nastran results and analytical solution from [1]:

	Speed, rpm	Freq, Hz	Freq, Hz	Freq, Hz	Freq, Hz
SOL 107	0	10.32	10.32	34.59	34.59
Analytical [1]	0	10.34	10.34	34.72	34.72
Error		0.15%	0.16%	0.39%	0.38%
SOL 107	900	8.30	11.86	25.42	50.95
Analytical [1]	900	8.31	11.88	25.55	51.06
Error		0.09%	0.19%	0.51%	0.22%

Conclusion

The Jeffcott rotor eigenvalues results from Nastran SOL 107 match the analytical results. The natural frequencies' maximum error is 0.39% at 0 rpm and 0.51% at 900 rpm.

3.3 Complex Eigenvalue solution for Jeffcott rotor with multiple disks

Problem Description

Solve a Jeffcott rotor with multiple disks for complex eigenvalues.

Solution Number

SOL 107

Features Used

- Massless shaft flexible rotor with lumped mass from disk - Jeffcott Rotor
- Rigid supported bearing - Pinned at bearing location.

Reference

1. Friswell, M. I., Penny, J. E. T., Garvey, S. D. and Lees, A. W. (2010). Dynamics of rotating machines. Cambridge University Press, P205-206.

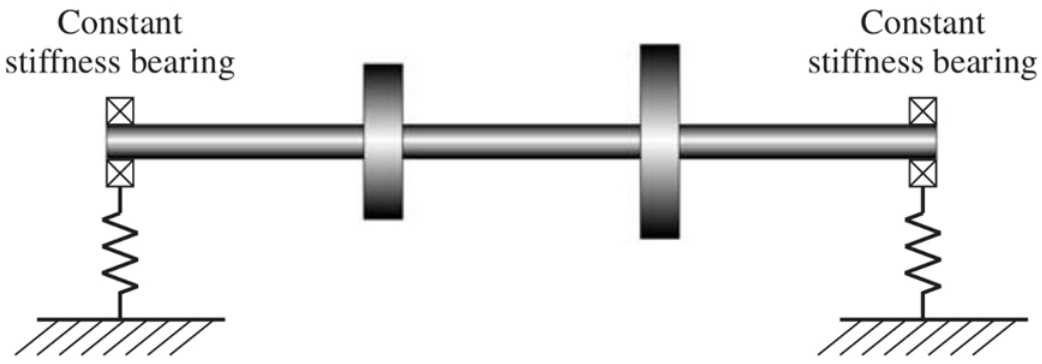
Modeling Techniques Used

- Apex/Patran was used to create the Finite Element 1D rotor model (ROTORG): Beam element for shaft (CBEAM), lumped mass/inertia for disk (CONM2); displacement constrains at two ends (SPC1).

Illustrations of the Model

The flexible rotor with a multiple disk [1] is shown as following:

Example - A 1.5m long shaft, has a diameter of 0.05m. The disks are keyed to the shaft at 0.5 and 1 m from one end. The left disk is 0.07m thick with a diameter of 0.28m; the right disk is 0.07m thick with a diameter of 0.35m. For the shaft, E = 211GN/m^2 and G=81.2GN/m^2. There is no internal shaft damping. For both the shaft and the disks, ρ = 7810 kg/m^3. The shaft is supported by identical bearings at its ends.



The finite element model used in the analysis is shown in the following figure.



Massless shaft material:

```
$ Description of Material: rho=0
MAT1     1        2.11e11  81.2e9          7810.
$
```

Pinned ends (bearings)

```
$ $ fix bearing node
SPC1     1        36       1        THRU     20
SPC1     1        36       1001     1002
SPC1     1        123456   100
```

Lumped mass added to Grids 7 & 10:

```
$ Elements and Element Properties: CONM2
$ Lumped mass and inertia
$.......2.......3.......4.......5.......6.......7.......8.......
CONM2   100     7       0       33.663170.      0.      0.
        .17869530.      .17869530.      0.      .329899
$
CONM2   101     14      0       52.5987 0.      0.      0.
        .42418660.      .42418660.      0.      .8054176
```

Rotor define

```
$ Rotor Dynamics data for RGYRO 1
ROTORG  4       1       THRU    20
RSPINR  4       9       10      RPM     1.
        0.
RGYRO   1       ASYNC   4       RPM                     0.0
```

Campbell define:

```
$ Parameter for Campbell Plot
CAMPBLL,15,SPEED,22,RPM
DDVAL,22,4000.0,THRU,0.0,BY,-4000.
```

Model File Name and Location

Model File Name	Description	Location
Friswell_5-9-1.dat	Jeffcott Rotor with Multiple Disks	\verifman\rotor\

Results

The comparison of MSC Nastran results and analytical solution from [1]:

Mode Type	Speed, rpm	Freq, Hz	Freq, Hz	Freq, Hz	Freq, Hz	Freq, Hz	Freq, Hz
SOL107	0	13.65	13.65	43.26	43.26	113.53	113.53
Analytical [1]	0	13.79	13.79	43.66	43.66	114.08	114.08
Error		1.02%	1.02%	0.92%	0.92%	0.48%	0.48%
SOL107	4000	13.45	13.83	39.75	46.44	95.24	130.63
Analytical [1]	4000	13.59	13.97	40.07	46.90	95.52	131.63
Error		1.03%	1.00%	0.80%	0.80%	0.29%	0.76%

Main Index

Conclusion

The Jeffcott rotor eigenvalues results from Nastran SOL 107 match the analytical results. The natural frequencies' maximum error is 1.02% at 0 rpm and 1.03% at 4000 rpm.

3.4 Complex Eigenvalue Solution for Jeffcott Rotor with Overhung Disk and Anisotropic Bearings

Problem Description

Solve a Jeffcott rotor with overhung disk supported by anisotropic bearings for complex eigenvalues.

Solution Number

SOL 107

Features Used

- Massless shaft flexible rotor with lumped mass from disk - Jeffcott Rotor
- Rigid anisotropic bearing - Pinned at bearing location.

Reference

1. Friswell, M. I., Penny, J. E. T., Garvey, S. D. and Lees, A. W. (2010). Dynamics of rotating machines. Cambridge University Press, P278-279.

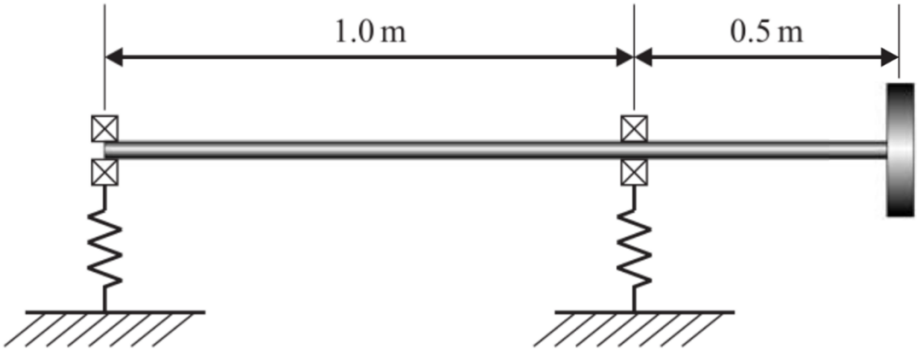
Modeling Techniques Used

- Apex/Patran was used to create the Finite Element 1D rotor model (ROTORG): Beam element for shaft (CBEAM), lumped mass/inertia for disk (CONM2); displacement constrains at two ends (SPC1).

Illustrations of the Model

The flexible rotor with a multiple disk [1] is shown as following:

Example - A 1.5m long shaft, has a diameter of 25mm. Shaft is supported with bearings at 0.0m & 1.0m. The bearings are short in that they present insignificant angular stiffness to the shaft, but they present finite translational stiffness. A disk is overhung is 250mm is diameter and 40mm thick. The shaft and disk are made of steel, and a mass density = 7810 kg/m^3, modulus of elasticity E = 211GN/m2 and Poisson's ratio of 0.3 are assumed.



The finite element model used in the analysis is shown in the following figure.



Several different variants of this system are considered in which the difference line in the bearing (and bearing support) properties as shown in below table.

Case	Stiffness (NM/m)		Damping (kNs/m)	
	K_{xx}	K_{yy}	C_{xx}	C_{yy}
1	10	10	0	0
2	10	20	0	0
3	10	20	60	60
4	10	20	400	400

Different bearing setups are as shown:

```
$ Pg.279 TABLE 6.3 Bearing support Characteristics 1
PBUSH   211     K       1.00+7  1.00+7
PBUSH   212     K       1.00+7  1.00+7
$
$ Pg.279 TABLE 6.3 Bearing support Characteristics 2
PBUSH   211     K       1.00+7  2.00+7
PBUSH   212     K       1.00+7  2.00+7
$
$ Pg.279 TABLE 6.3 Bearing support Characteristics 3
PBUSH   211     K       1.00+7  2.00+7
PBUSH   212     K       1.00+7  2.00+7
                B       60.0+3  60.0+3
$
$ Pg.279 TABLE 6.3 Bearing support Characteristics 4
PBUSH   211     K       1.00+7  2.00+7
PBUSH   212     K       1.00+7  2.00+7
                B       400.+3  400.+3
```

(or)

(or)

(or)

Massless shaft material

```
$ Description of Material: rho=0
MAT1     1        2.11e11          .3       7810.
```

Lumped mass added to Grids 7 & 10

```
$ Elements and Element Properties: CONM2
$ Lumped mass and inertia
$.......2.......3.......4.......5.......6.......7.......8.......
CONM2    100      7        0        15.334900.       0.       0.
         .06194660.        .06194660.       0.       .1198039
```

Pinned ends (bearings)

```
$ $ fix bearing node
SPC1     1        36       1        THRU     7
SPC1     1        36       1001     1002
SPC1     1        123456   100      2001     2002
$.......2.......3.......4.......5.......6.......
```

Rotor define

```
$ Rotor Dynamics data for RGYRO 1
ROTORG   4        1        THRU      7
RSPINR   4        1         2       RPM      1.0
         0.
RGYRO    1        ASYNC    4        RPM                        1000.0
```

Campbell define:

```
$ Parameter for Campbell Plot
CAMPBLL,15,SPEED,22,RPM
DDVAL,22,3000.0,THRU,0.0,BY,-1000.
```

Model File Name and Location

Model File Name	Description	Location
Friswell_6-8-3.dat	Jeffcott Rotor with Overhung Disk and Anisotropic Bearing Supports	\verifman\rotor\

Results

The comparison of MSC Nastran results and analytical solution from [1]:

Speed rev/min		Bearing Setup		Pair of Roots (rad/s)			
		Left	Right	First	Second	Third	Fourth
0	SOL107	1	1	0 ± 44.15j	0 ± 44.15j	0 ± 397.36j	0 ± 397.36j
0	Analytical [1]	1	1	0 ± 44.28j	0 ± 44.28j	0 ± 397.6j	0 ± 397.6j

Speed rev/min		Bearing Setup		Pair of Roots (rad/s)			
		Left	Right	First	Second	Third	Fourth
0	SOL107	2	2	0 ± 44.15j	0 ± 44.35j	0 ± 397.36j	0 ± 399.86j
0	Analytical [1]	2	2	0 ± 44.28j	0 ± 44.36j	0 ± 397.6j	0 ± 400.2j
0	SOL107	2	3	-0.037 ± 44.16j	-0.0099 ± 44.35j	-1.219 ± 400.4j	-0.850 ± 400.9j
0	Analytical [1]	2	3	-0.038 ± 44.29j	-0.010 ± 44.36j	-1.220 ± 400.6j	-0.851 ± 401.2j
0	SOL107	2	4	-0.065 ± 44.26j	-0.038 ± 44.26j	-0.211 ± 400.8j	-0.208 ± 401.5j
0	Analytical [1]	2	4	-0.066 ± 44.39j	-0.038 ± 44.40j	-0.211 ± 401.1j	-0.209 ± 401.9j
1000	SOL107	1	1	0 ± 42.10j	0 ± 46.21j	0 ± 389.8j	0 ± 402.1j
1000	Analytical [1]	1	1	0 ± 42.22j	0 ± 46.35j	0 ± 389.9j	0 ± 402.3j
1000	SOL107	2	3	-0.020 ± 42.13j	-0.028 ± 46.30j	-0.625 ± 391.9j	-1.352 ± 406.3j
1000	Analytical [1]	2	3	-0.020 ± 42.26j	-0.028 ± 46.40j	-0.619 ± 392.0j	-1.358 ± 406.6j
2000	SOL107	2	3	-0.016 ± 40.10j	-0.033 ± 48.33j	-0.201 ± 378.0j	-1.595 ± 410.1j
2000	Analytical [1]	2	3	-0.017 ± 40.22j	-0.033 ± 48.47j	-0.190 ± 377.9j	-1.605 ± 410.3j
3000	SOL107	2	3	-0.137 ± 38.12j	-0.038 ± 50.37j	-0.0001 ± 357.8j	-1.774 ± 412.8j
3000	Analytical [1]	2	3	-0.139 ± 38.23j	-0.039 ± 50.51j	-0.0004 ± 357.3j	-1.786 ± 413.0j

Conclusion

The Jeffcott rotor eigenvalues results from Nastran SOL 107 match the analytical results for different bearing stiffness & damping settings.

Section 4: SOL 200 Optimization Problems



Main Index

4.1 Optimization of a Three Bar Truss

Problem Description

Optimization of a three bar truss.

Solution Number

SOL 200

Features Used

- Sizing optimization with design constraints on displacements and stresses in the simple structure under two static load conditions.

Reference

1. DOT-Design Optimization Tools, User's Manual ,Vanderplaats Research and Development, Colorado Springs, CO, 1999.

Modeling Techniques Used

- The simple model can be constructed using bulk data entries directly, or from within any preprocessor such as PATRAN. Patran can also create the design task as outlined in Chapter 8 of the *MSC Nastran Design Sensitivity and Optimization User's Guide* (DSOUG). Figure 4.1-1 shows the finite element model which is made up of three rod elements loaded statically with two load cases. The design task is to minimize the weight of the structure while limiting the displacement in the y direction at node 4 to be between $-0.2 \le \delta \le 0.2$ inches and the stress in the bars to be $-15000 \le \sigma \le 20000$ psi. The loading for example of this manual differ from that of the DSOUG in that the 20000 lb loads are applied at a 45° angle.

Illustrations of the Model

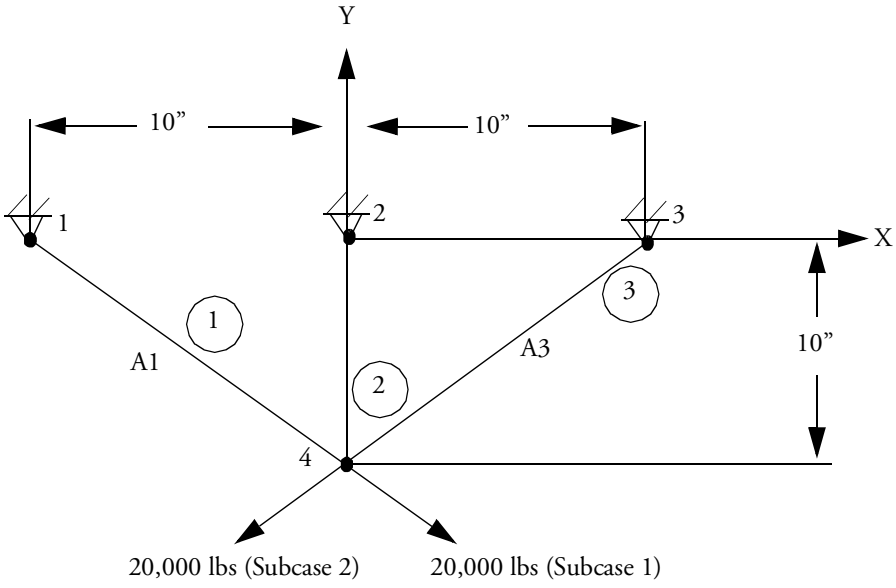


Figure 4.1-1 Three-Bar Truss

Model File Name and Location

Model File Name	Location
`vandy.dat`	tpl\verifman\SOL200

Results

The table below shows a comparison between the MSC Nastran result and the result in the reference.

Term	Reference Result	MSC Nastran Results
Weight	2.633	2.636
Design Variable 1	0.799	0.771
Design Variable 2	0.372	0.456
Design Variable 3	0.799	0.771
Maximum Constraint	.0028	.0028
Number of Function Evaluations	29	5

Conclusion

Slight differences are observed between the MSC Nastran result and the Referenced Results, which is typical for the nonlinear programming tasks involved with optimization. MSC Nastran employs sensitivities to make the optimization search process more efficient so that a converged result is achieved in 4 design cycles (5 function evaluations) while the reference uses gradients computed using finite differences, and therefore, many more function evaluations.

4.2 Design with Constraint on Minimum Frequency

Problem Description

Optimization of a cantilevered structure with a constraint on the first natural frequency.

Solution Number

SOL 200

Features Used

- Sizing optimization with a design constraint on a natural frequency.

Reference

1. M.J. Turner "Design of Minimum Mass Structures with Specified Natural Frequencies," AIAA Journal , Vol 5, No. 3, March 1967.

Modeling Techniques Used

- The simple model can be constructed using bulk data entries directly, or from within any preprocessor such as PATRAN. Patran can also create the design task as outlined in Chapter 8 of the *MSC Nastran Design Sensitivity and Optimization User's Guide* (DSOUG). The finite element model is made up of three shear webs with rod elements acting as spar caps on the top and bottom of the webs. Six CONM2 elements provide nonstructural mass and act to make the design task meaningful (without these masses, the structural mass and stiffness would be proportional and the design task would lead to design variables reaching their lower bounds). The design task is to minimize the weight of the structure while ensuring that the first natural frequency is greater than 20 Hz.

  Analysis Model

  - Web is modeled by QUAD4 elements
  - Caps are modeled by ROD elements
  - Material: E = 1.03E7 psi
  - Poisson ratio = 0.3
  - Nonstructural mass: Lumped masses at top and bottom nodes, 15 lbs each.

Illustrations of the Model

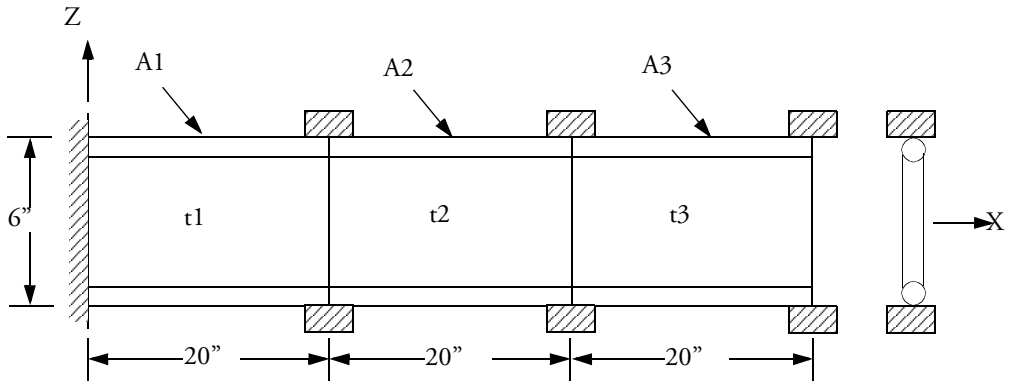


Figure 4.2-1 Cantilever Beam Vibration Model

Model File Name and Location

Model File Name	Location
`dsoug2.dat`	tpl\verifman\SOL200

Results

The referenced analysis is from 1967 and used innovative techniques of the time to attack the design task. The webs of the structure were taken to be piecewise linear with the design variables the thickness at four spanwise stations. MSC Nastran considered all the elements to be constant thickness so there are three design variables for the web and another three for the caps (top and bottom caps are taken to be equal). The table below averages the web thickness from the two ends to provide a direct comparison.

The table below shows a comparison between the MSC Nastran result and the result in the reference.

Term	Reference Result	MSC Nastran Results
Weight	6.91	6.97
Web 1	0.91	0.771
Web 2	0.485	0.456
Web 3	0.037	0.771
Cap 1	0.037	0.045
Cap 2	0.034	0.040
Cap 3	0.023	0.025
Maximum Constraint	0.0	.0026

Conclusion

It is seen that the two methods produce the same basic design but there are differences between them. There are significant differences in the modeling techniques used and the algorithms employed, so the strong qualitative agreement is considered acceptable.

4.3 Shape Design of a Culvert

Problem Description

Optimization of a Culvert Structure using Shape Variables to provide a least weight design with design constraints on element stresses.

Solution Number

SOL 200

Features Used

- Shape optimization with constraints on the stress in the elements.

Reference

1. Belegundu, A.D. and Rajan, S.P., "Shape Optimal Design Using Isoparametric Elements," Proceedings, 29th AIAA/ASME/ASEE/AMS/ASC Structures, Dynamics and Materials Conference, pp. 696-701, Williamsburg, VA, April 1988.

Modeling Techniques Used

- The 2-d culvert model is meshed using 27 CQUAD4 elements. A pressure load is applied to the "road" surface.
  The simple model can be constructed using bulk data entries directly, or from within any preprocessor such as PATRAN. Figure 4.3-1 shows the finite element model. The design task is to find the shape that minimizes the volume of the structure while modifying the shape of the initially circular hole. Grid points on the plane of symmetry can move in the y-direction while those at the base can move in the x-direction. A constraint is placed on the maximum allowable von Mises stress ≤31000 psi. A separate "auxiliary" model is used to create the candidate shape vectors that deforms the structure while separately applying a radial load to each of the seven grids on the cutout of the culvert. Stiff beams are added to cutout of this auxiliary model to create smooth deformation due to the applied point loads.

Illustrations of the Model

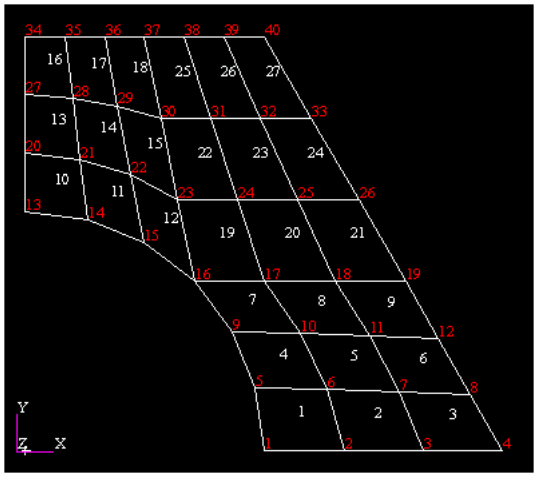


Figure 4.3-1 Initial Culvert Design

Model File Name and Location

Model File Name	Description	Location
`Dsoug5a.dat`	Auxiliary model for Culvert Example	\verifman\SOL200
`Dsoug5.dat`	Shape optimization of culvert	\verifman\SOL200

Results

The final design, including stress contours, is shown in Figure 4.3-2. An exact comparison with the reference cannot be made because

a. the number of elements used to mesh the surface differs between the referenced paper and the current analysis and

b. the reference does not contain data (such as grid locations) that can be compared directly.

However there is good qualitative agreement in that the reference indicates the final design has reduced the weight by 19% and MSC Nastran shows an 18.9% reduction.

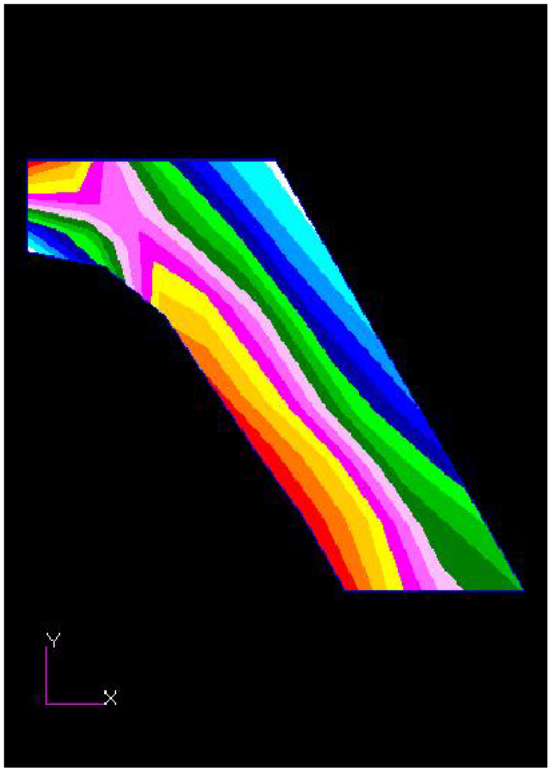


Figure 4.3-2 Final Culvert Design

4.4 Topology Design of the MBB Beam

Problem Description

A classic problem of topology optimization that seeks the optimal topology design of a simply supported beam under a point load.

Solution Number

SOL 200

Features Used

- Topology optimization with constraints on the weight of the structure and minimal compliance as the design objective

Reference

1. Sigmund Ole, Morphology-Based Black and White Filters for Topology Optimization. Structural and Multidisciplinary Optimization, 33(4-5) 401-424.https://doi.org/10.1007/s-158-006-0087-x.

Modeling Techniques Used

- This MBB example is widely used by academic and industrial researchers for Topology Optimization validation. A mesh of 4800 CQUAD4 elements represents the structure while a point load is applied vertically at the top left side of the model. The literature contains numerous representations of this model with differing dimensions and loading. However, they all have the same aspect ratio for the beam and apply the load at the same location and direction so that the topology results can be compared qualitatively regardless of the dimensions/loading/materials.

  Design Model Description

  Objective: Minimize compliance

  Topology design region: PSHELL

  Constraints: Mass target = 0.5 (i.e. mass savings 50%)

  Minimum member size control

  The simple model can be constructed from within any preprocessor such as PATRAN. The design model information is very terse in the case of topology optimization and can be generated manually or through the use of Patran.

Illustrations of the Model

Due to symmetry, a half span model is appropriate as shown in Figure 4.4-1

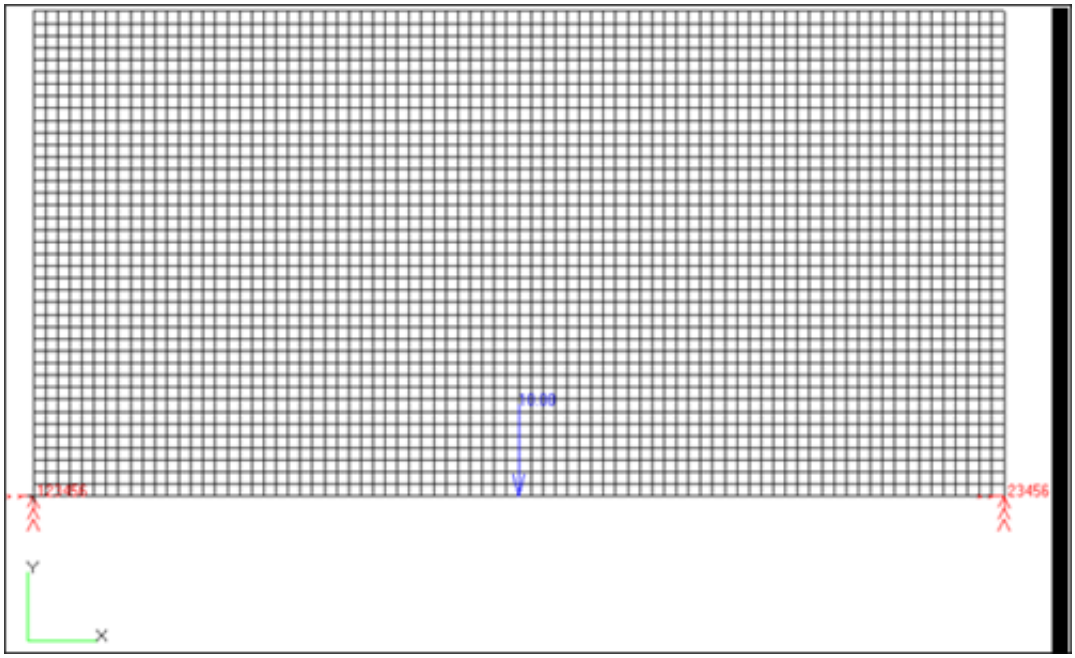


Figure 4.4-1 Creation and Selection of Subcase

Model File Name and Location

Model File Name	Location
mbb.dat	tpl\verifman\SOL200

Results

The final design is shown in Figure 4.4-2. The minimum thickness, which applies to the members generated by the design, is very important in constructing this figure as it prevents the possibility of very small members. An exact comparison with the reference cannot be made because of differences in dimensions in the two studies and wide number of variations applied to the design task. Nonetheless, many of the designs of the reference show a similar final design with the result of Figure 5g in the reference very similar to the one shown here.

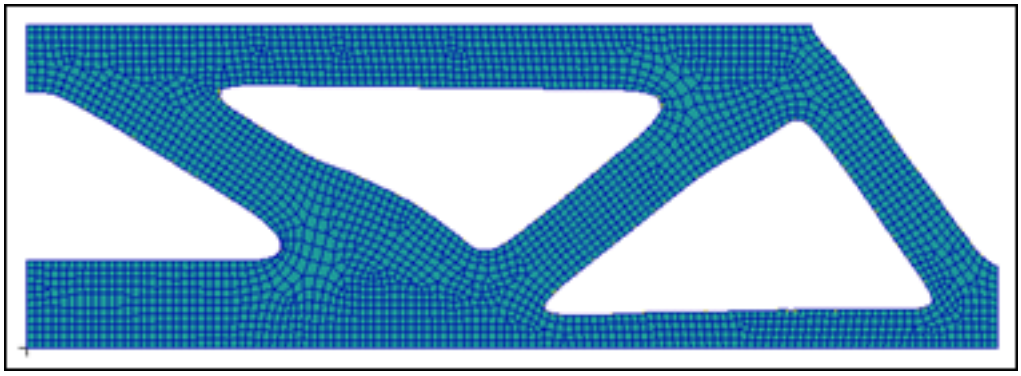


Figure 4.4-2 MBB Beam Topology Design with “Minimum Member Size”

Section 5: SOL 700 Explicit Dynamics Problems



Main Index



Main Index

5.1 Impulsively Loaded Strip

Problem Description

An aluminum strip clamped at its left and right edges is subjected to an impulsive initial velocity (v_z = -132 m/sec) over its central portion (one-fourth of the length).

The purpose is to investigate the sensitivity of the response to Poisson's ratio (ν =0.01, ν = 0.3) and to check results against data available from experiments performed at the Air Freight Flight Dynamics Laboratory.

This problem is also presented in the *Demonstration Problem Manual- Explicit Nonlinear (SOL 700)*.

Solution Number

SOL 700

Features Used

- Explicit dynamics.
- Elastic-plastic material.
- Impact (initial velocity)

Reference

1. Balmer, H. A. and Witmer, E. A. “Theoretical-experimental Correlation of Large Dynamic and Permanent Deformation of Impulsively Loaded Simple Structure,” 1964, Air Force Flight Dynamics Laboratory report FDRTDR-64-108.

Modeling Techniques Used

Due to the symmetry of the problem, only half of the strip needs to be modeled (see Figure 5.1-1). The left hand half strip is discretized by a regular mesh of (30x3) quadrilateral elements.

The MAT1 and MATEP entries are used to input the aluminum elastic-plastic material data.

Since plastic deformations occur, the PSHELL1 entry is used to specify five integration points across the thickness. The PSHELL1 entry defaults to three integration points.

The SPC entry is used to impose the zero displacement/rotations of the damped left-hand edge and zero out of-plane displacement/rotations of the right-hand edge (symmetry plane).

To better represent the actual experimental conditions, the initial velocity distribution is slightly modified so as to have a smooth transition to the remainder of the strip. The TIC1 entry is used to input initial velocities. Using the PARAM, INISTEP entry, the initial time step is set to 0.1e-6 sec according to the COURANT criterion.

length	L = 0.254m
width	W = 0.0305m
thickness	t = 0.00318m
density	$\rho = 2791 kg/m^3$
Young's modulus	$E = 7.17 \times 10^{10} N/m^2$
Poisson's ratio	$\nu = 0.3$
yield stress	$\sigma_y = 2.854 \times 10^8 N/m^2$
hardening modulus	$E_h = 0 N/m^2$

The ENDTIME Case Control entry is used to follow the dynamic behavior in the range up to 0.001 seconds. The Case Control entries TYPE, SAVE, GPOUT, GRIDS, and TIMES are used to build the z displacement time history of Grid Point 31 (in the plane of symmetry) by saving results every 0.01 msec. The experimental results are available for every 0.1 msec.

Illustrations of the Model

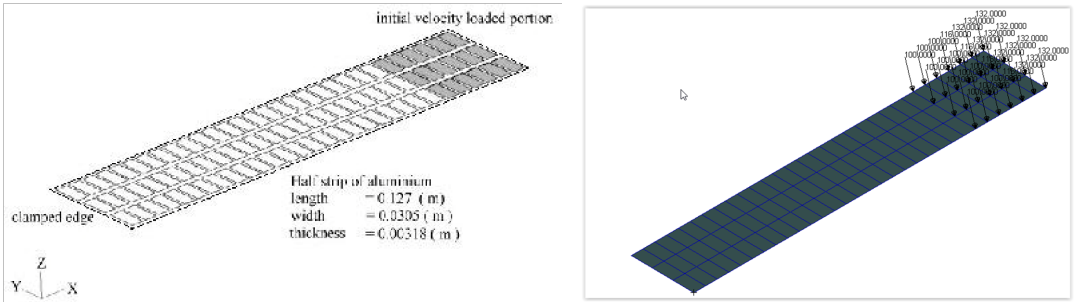


Figure 5.1-1 Numerical Model Layout

Model File Name and Location

Model File Name	Description	Location
`Impulse_a.dat`	ν = 0.01	Verif\sol700\impulsly_lded_strip doc\SOL700_input_files\Structural_Dynamics\impulsly_lded_strip
`Impulse_b.dat`	ν = 0.3	Verif\SOL700\impulsly_lded_strip doc\SOL700_input_files\Structural_Dynamics\impulsly_lded_strip

where the input file is in `tpl` directory.

Results

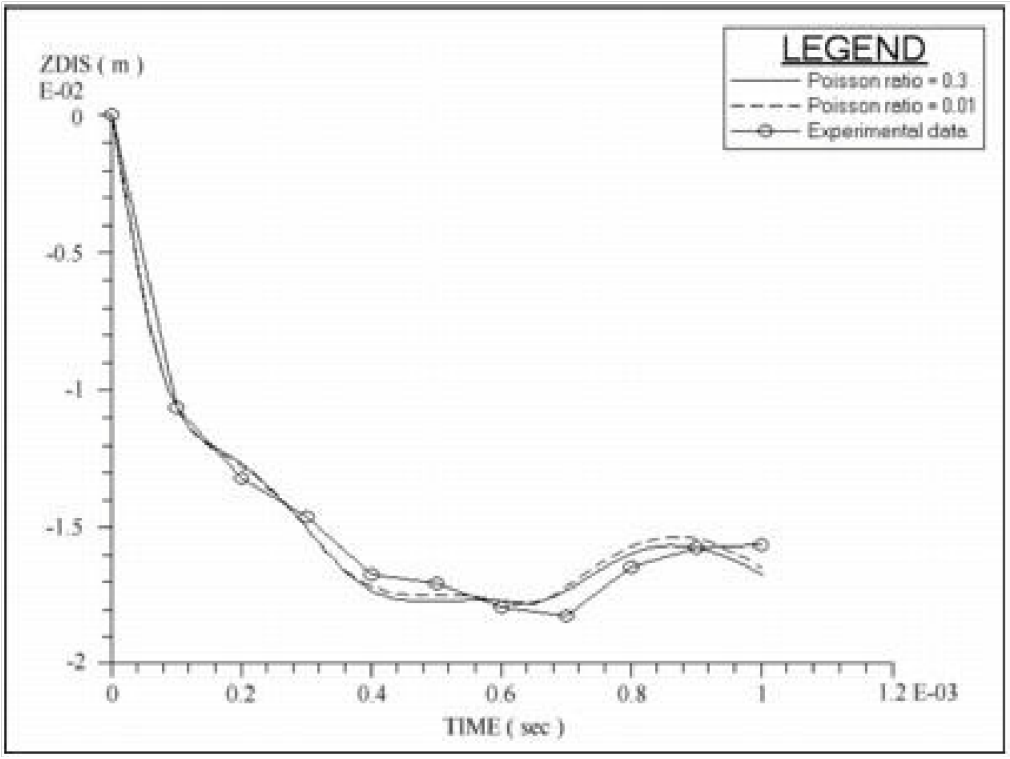


Figure 5.1-2 Vertical Z-displacement Time History of the (Free) Corner Grid Point (31) of the Half Strip

Conclusion

As viewed in the above figure, the analytical results agree well with the test results.

5.2 Basic Blast Analysis

Problem Description

The effect of a detonation on the environment can be simulated by assuming that the detonated material can be idealized by a sphere of hot gas with a homogeneous density and specific internal energy. This approach is suited for problems in which the processes inside of the explosive material are not to be investigated.

In this problem, the propagation of a blast wave will be simulated starting from the initial shock front radius R_0=0.05 m at the time t = 0 sec until it reaches a radius of R = 10 R_0. Both, the gas in the sphere and the surrounding environment behave as an ideal gas (Gas Constant = 1.4).

Solution Number

SOL 700

Features Used

- Transient explicit dynamic analysis.
- 8-node hex elements for Euler
- Two regions (high and low pressure) are defined with transient initial values

Reference

1. Baker, W. E., Explosions in Air, University of Texas Press, 1973, Austin and London.

Modeling Techniques Used

The motion of the gas is radial. Therefore, only a part of the area of interest need to be analyzed. Here, a mesh of 20 x 20 x 20 hexahedral Eulerian elements has been used for investigating the volume (0 < x < 0.3 m, 0 < y < 0.5 m, 0 < z < 0.5 m). The center of the blast wave is located at the origin of the global coordinate system. A higher level has to be used to indicate the higher priority of the spherical condition. The value for the parameter INISTEP has to be below the minimal time step that follows from the Courant Criterion:

$$\Delta t = S\frac{l}{c_0} \tag{5.2-1}$$

where l denotes the smallest element dimension, c_0 the initial speed of sound, and S is a safety factor Dytran's default value is 2/3).

With $l = 10R_0/20 = 0.025$ m and $c_0 = \sqrt{\gamma(\gamma-1)e_1}$ Equation (5.2-1) yields for the time step $\Delta t = 1.96\text{E-}7$ second. Therefore, the value 1.96E-7 sec is used for the INISTEP parameter.

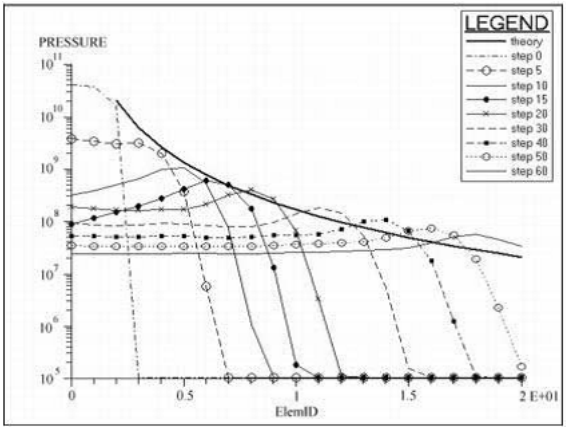


Figure 5.2-1 Radial Pressure Distribution for Several Time Steps and Theoretical Maximum Pressure

Illustrations of the Model

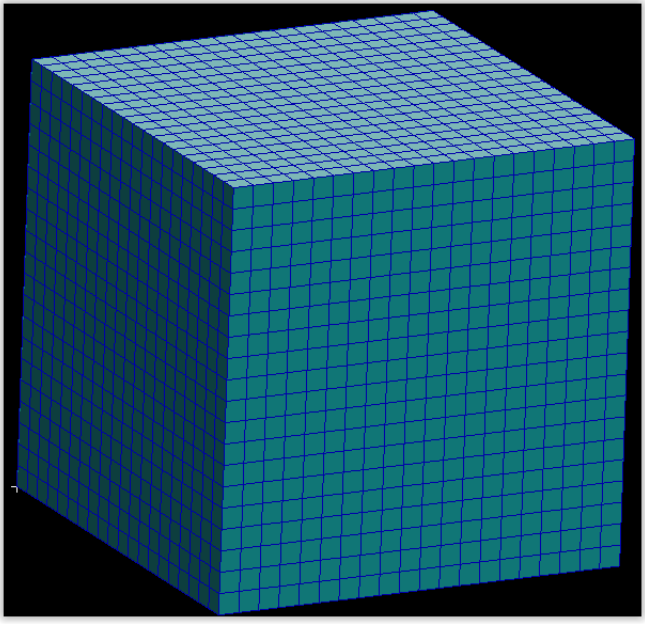


Figure 5.2-2 SOL 700 CTETRA Model - Impact of Cylinder - Quarter Model

Model File Name and Location

Model File Name	Description	Location
blast_sol700.dat	ν =0.1	Verifman\sol700\Basic_Blast doc\SOL700_input_files\Fluid_Dynamics\Basic_Blast

Results

Figure 5.2-3 shows pressure profiles in the elements 1 to 20, which are located along the edge y = z = 0 m of the control volume. The theoretical value of the shock front pressure is also included in this plot. Though this mesh is very coarse, the analysis results is a fairly good approximation of the theoretical values. Note that at t = 0, element 2 is not completely inside of the sphere of high energy gas. Therefore, its pressure is below that of element 1 at t = 0 sec.

Conclusion

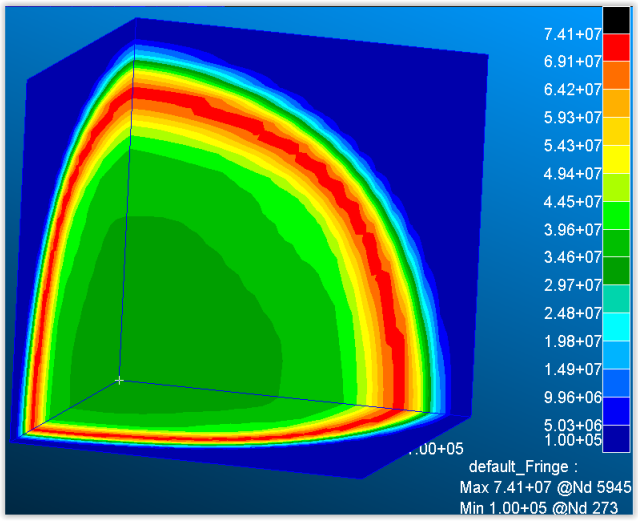


Figure 5.2-3 Pressure Contour Plot for Step 50

The contour plot of the pressure in Figure 5.2-3 shows an almost spherical shape of the shock front. Deviations are due to the fact that mass transport takes place along element faces only. Thus, the relief of pressure is hampered for the elements on the diagonal.

5.3 JWL Blast Analysis

Problem Description

This example demonstrates detonation of high explosives.

Solution Number

Transient explicit dynamic analysis (SOL 700)

Features Used

- Detonation of High Explosives (HE)
- Hydrodynamic Theory of Steady-state Plane Detonation
- Steady-state Detonation Modeling with SOL 700

Modeling Techniques Used

Due to symmetry, the reaction zone (detonation front) is a plane traveling along the length of the slab. Therefore, it is sufficient to model only a portion of the slab with all boundary faces closed to transport.

A mesh of 200 elements along the 50 cm of slab length is used (element thickness of 0.25 cm).

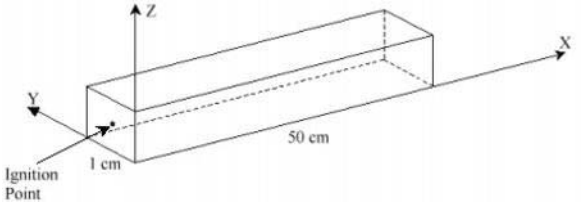


Figure 5.3-1 Geometrical Layout of Explosive and Detonation Point

At the start time, all the elements are filled with explosive material. Therefore, they all reference a EOSJWL equation of state defined by a DMAT entry.

The specific chemical energy q_0 is assigned as the initial specific internal energy of the explosive by using the TICEL entry.

The Chapman-Jouguet detonation velocity D_{cj}, the ignition point, and the ignition time are specified by the DETSPH entry. The ignition time is taken as the start time of the analysis and the ignition point is the center of the left face of the mesh. See Figure 5.3-1.

The duration of the analysis is set to 60 μsec necessary to burn the slab (50 cm) at a detonation speed of 7980 m/sec.

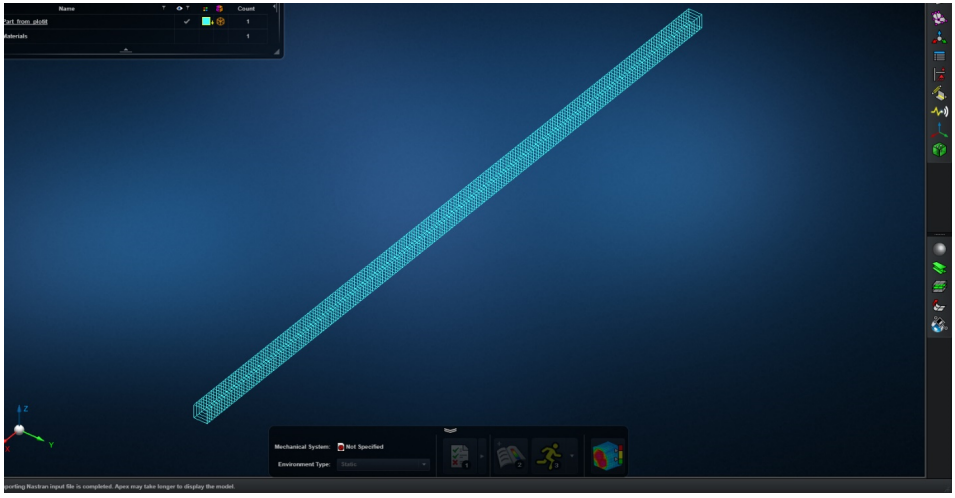
Edits of pressure profiles are requested every 10 μsec.

Boundary conditions: Two regions (high and low pressure) are defined with transient initial values

Element type: 8-node hex elements for Euler

FE results: Pressure distribution plot

Illustrations of the Model



Model File Name and Location

Model File Name	Description	Location
ep3_3.dat	ν =0.1	Verifman\sol700\JWL_Blast\ep3_3.dat doc\SOL700_input_files\Fluid_Dynamics\JWL_Blast\ep3_3.dat

Results

The solution of the detonation analysis is shown in Figure 5.3-2.

The peak pressure of each profile corresponding to the pressure behind the detonation front is depicted in Figure 5.3-3 in terms of the Chapman-Jouguet pressure fraction.

From the hydrodynamic theory of steady-state plane detonation for a γ-law gas, the CJ pressure is

$$P_{cj} = 2(\gamma_{cj} - 1)q_0\rho_0$$

with γ_{cj} = 2.706 for TNT, this leads to

$$P_{cj} = 2.9 * 10^{10} Pa$$

The code needs about 60 elements to build up the detonation front at which time the pressure reaches approximately 0.85 of P_{cj}. Subsequently, the front propagates with only a small increase of pressure.

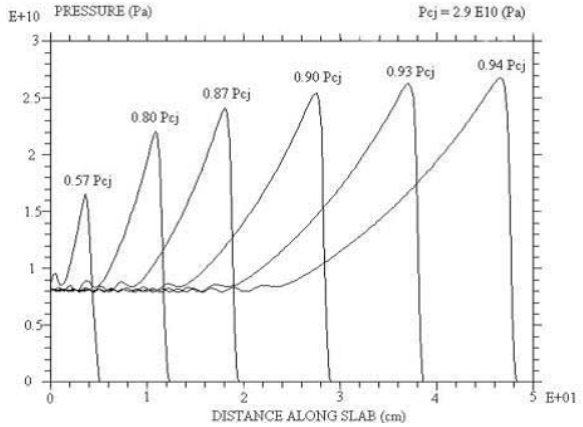


Figure 5.3-2 Peak Pressure Profiles at Different Distances

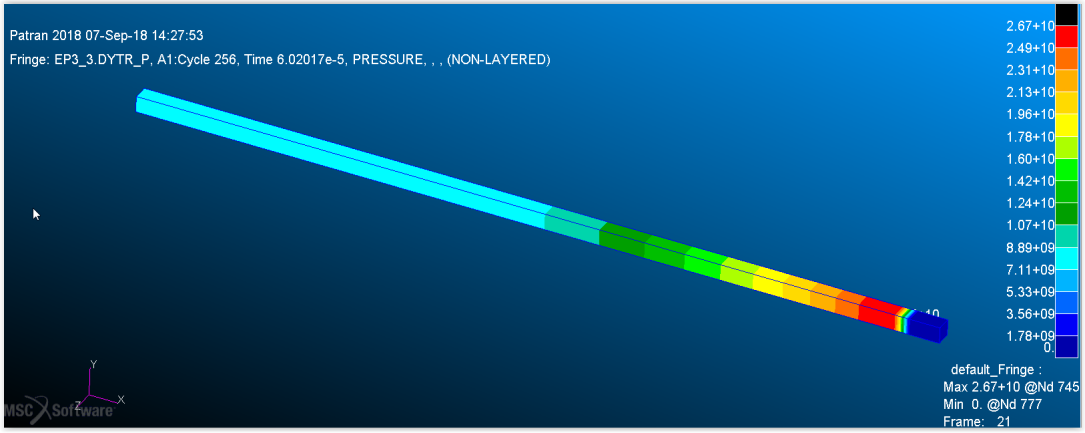


Figure 5.3-3 Pressure Distribution Plot

Conclusion

These results are acceptable if the fact is taken into account that SOL 700 is a first-order code that smears the shock front over a number of elements (always conserving momentum and energy). The result of that will be a reduction in peak pressure.

5.4 Taylor Test (a rod impacted against a rigid wall)

Problem Description

The Taylor "bar test" is an important laboratory test in the science of ballistics. It enables to determine an average value of the dynamic yield stress of a material. It consists of accelerating a cylindrical bar (the velocity being parallel to the axis of symmetry), and then let the bar hit a rigid target. As a result, the bar shortens and the impact side expands radially acquiring a mushroom like shape.

Much research work has been done on these impact tests. In this example, the experimental work done by Johnson/Cook is validated with a number of simulations in SOL 700 with different element types. On top of this work, the results are compared against the theoretical solution developed by Taylor.

Johnson and Cook developed a material model that represents a constitutive model for materials subjected to large strains, high strain rates and high temperatures. The SOL 700 implementation is validated for a set of constitutive constants presented in Reference 1.

Solution Number

SOL 700

Features Used

A list of the functionalities used in this problem.

References

1. Johnson, G.R. and Hook, W.H. "A constitutive model and data for metals subjected to large strains, high strain rates and high temperatures", April 1983, 7th Ballistic Symposium, The Hague, The Netherlands
2. Wilkins, L.M and Guinan M.W. "Impact of cylinders on a rigid boundary", August 1972, Lawrence Livermore Laboratory

Modeling Techniques Used

Johnson-Cook Material Model

This material model is described in the Dytran Reference and User manuals. For more detailed information and data sets for various materials, see Reference 1.

Theoretical Approach - Taylor

Taylor developed a simple expression for the final length of the bar, L_f, as function of the initial length L_0, the impact velocity V, the yield strength σ_y and the density ρ.

$$L_f L_0 = \exp(-\rho V^2 / 2\sigma_y)$$

The assumptions made in this expression are:

- There is a stationary yield front near the wall.
- A quasi-steady process is assumed.

Theoretical - Taylor

In Figure 5.4-9, Figure 5.4-10, and Figure 5.4-11 the theoretical reduction in length according to Taylor is plotted:

$L_f / L_0 = 0.78$

This theoretical reduction is based on an assumption of constant yield of 700 MPa.

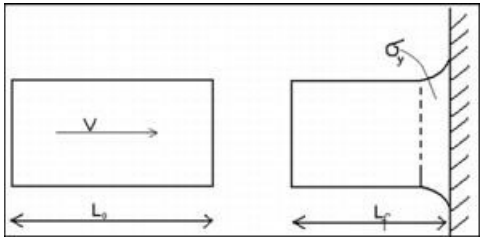


Figure 5.4-1 Schematic Overview of the Taylor Impact Test

In this Taylor test, the process stops when all the kinetic energy has converted to plastic work. It is interesting to note that the governing parameters of the process give a non-dimensional quantity, therefore, the result depends on this combination and not on the specific value of each parameter.

SOL 700 Model

A cylindrical rod with a length of 25.4 mm and a diameter of 3.82 impacts a rigid wall with a velocity of 190 mm/s. The material data:

- ρ = 8.96e-9 tonne/mm^3
- Bulk Modulus K = 143e3 MPa
- Shear Modulus G = 47.7e3 MPa
- Two different material models have been used:
  - Johnson/Cook
  - Constant von Mises: σ_y = 600 MPa (elastic-rigid-plastic)

The interface between the wall and the rod are assumed frictionless.

Illustrations of the Model

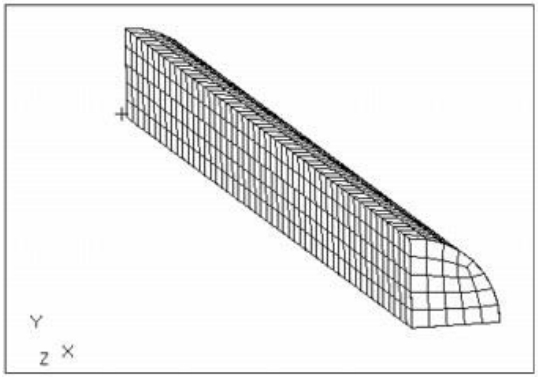


Figure 5.4-2 SOL 700 CHEXA Model - Impact of Cylinder - Quarter Model

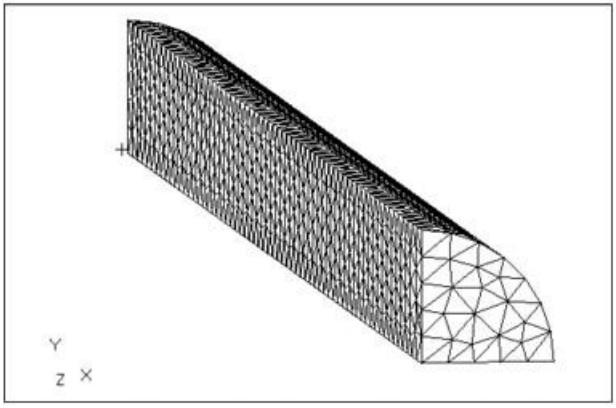


Figure 5.4-3 SOL 700 CTETRA Model - Impact of Cylinder - Quarter Model

Model File Name and Location

Model File Name	Description	Location

`sol700_hex2x2x2-full.dat`
and
`hexs-2x2x2full.bdf`
	INT=2, ISOP=FULL	Verif\sol700\ Taylor_test_rod_wall\Hexa2x2x2
		doc\SOL700_input_files\Structural_Dynamics\Taylor_test_rod_wall\Hexa2x2x2

`sol700_Hexs-JC_barimpact.dat`
and
`hexs-jc.bdf`
	INT=3	Verif\sol700\ Taylor_test_rod_wall\Hexs-EPM-default-JC
		doc\SOL700_input_files\Structural_Dynamics\Taylor_test_rod_wall\Hexs-EPM-default-JC

`sol700_Hexs-VM_2.dat`
and
`hexs-jc.bdf`
	INT=2	Verif\sol700\ Taylor_test_rod_wall\Hexs-EPM-default-JC
		doc\SOL700_input_files\Structural_Dynamics\Taylor_test_rod_wall\Hexs-EPM-default-JC

Results

The deformed shape is compared in Figure 5.4-4 through .

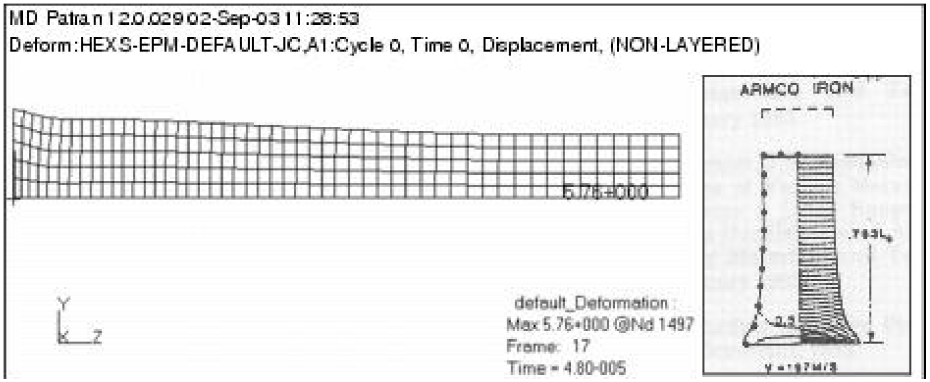


Figure 5.4-4 SOL 700 Simulation - Johnson Cook Material Model with CHEXA - One Point Gauss

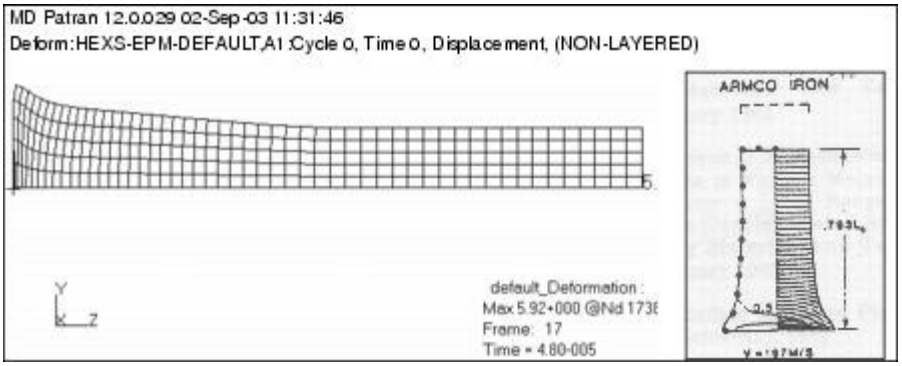


Figure 5.4-5 SOL 700 Simulation - von Mises Material Model with CHEXA - One Point Gauss

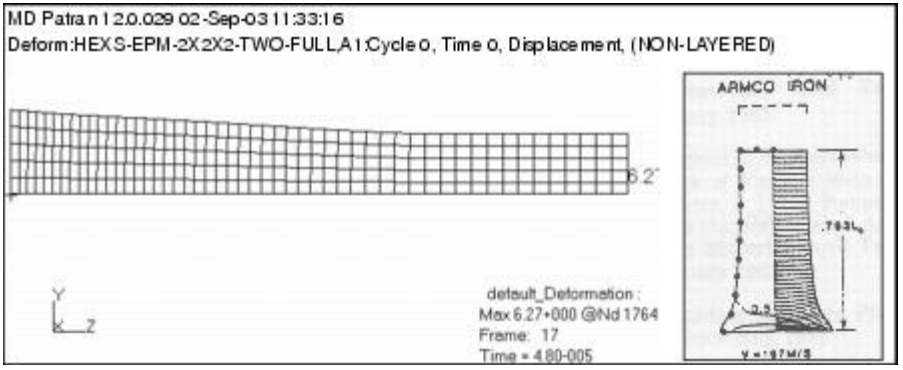


Figure 5.4-6 SOL 700 Simulation - von Mises Material Model with CHEXA - 2x2x2

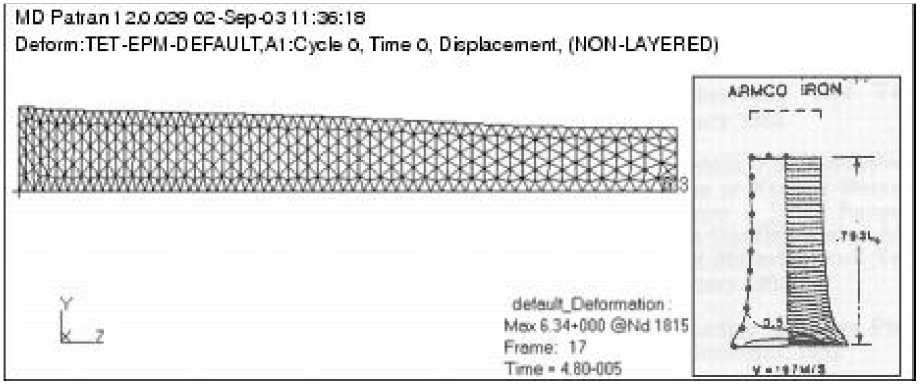


Figure 5.4-7 SOL 700 Simulation - von Mises Material Model with CTETRA - Old Element

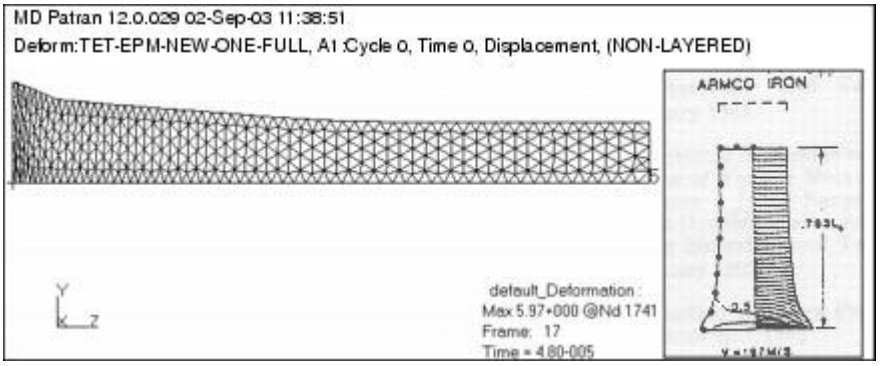


Figure 5.4-8 SOL 700 Simulation - von Mises Material Model with CTETRA - New Element

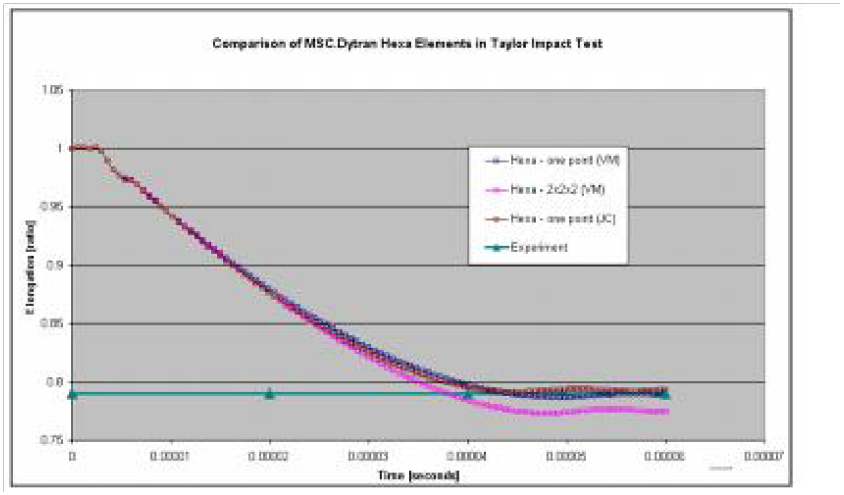


Figure 5.4-9 Comparison of SOL 700 CHEXA Elements

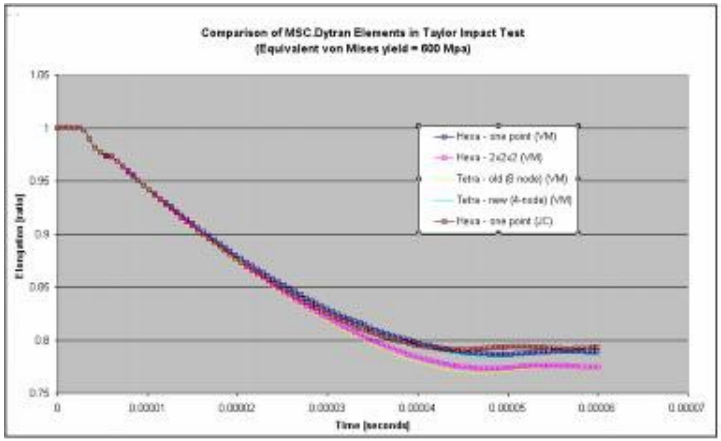


Figure 5.4-10 Comparison of Different Element Types in SOL 700

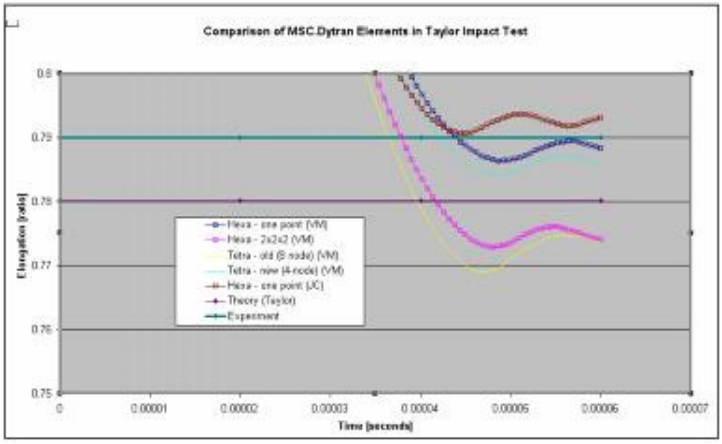


Figure 5.4-11 Detailed Comparison of Different Element Types in SOL 700

Conclusion

In Figure 5.4-11, a detailed section of Figure 5.4-10 is shown. The results all show an oscillating behavior that is common in all transient dynamic events and is physical. Furthermore, the new 2x2x2 CHEXA element is deviating from the experimental result. and further developments are targeted to improve the behavior of the new CHEXA element in impact events involving metal plasticity. The 2x2x2 CHEXA element has specifically been implemented and tested to reduce the hour-glassing phenomenon in rubber and foam modeling.

Section 6: NAFEMS Problems



Main Index

Linear Elastic

6.1 LE1: Elliptic Membrane

Problem Description

Find the tangential edge stress at D (refer below figure) of elliptical membrane which is uniformly loaded with outward pressure.

Solution Number

SOL 101

Features Used

- Linear Static Analysis.

Reference

1. Test LE1 from NAFEMS Publication NNB, Rev. 3, NAFEMS Linear Benchmarks, 5 Oct 1990.

Modeling Techniques Used

Elliptical membrane is modeled using plane stress elements CQUAD4. GPSTRESS is used to find out nodal stress output.

Loading

Uniform outward pressure of 10MPa at outer edge BC, Inner curved edge AD unloaded.

Boundary Condition

Edge AB, symmetry about Y axis, e.g. zero x displacement

Edge CD, symmetry about X axis, e.g. zero y displacement.

Engineering Data

E = 210E+03 MPa

ν = 0.3

P = 10 Mpa

Illustrations of the Model

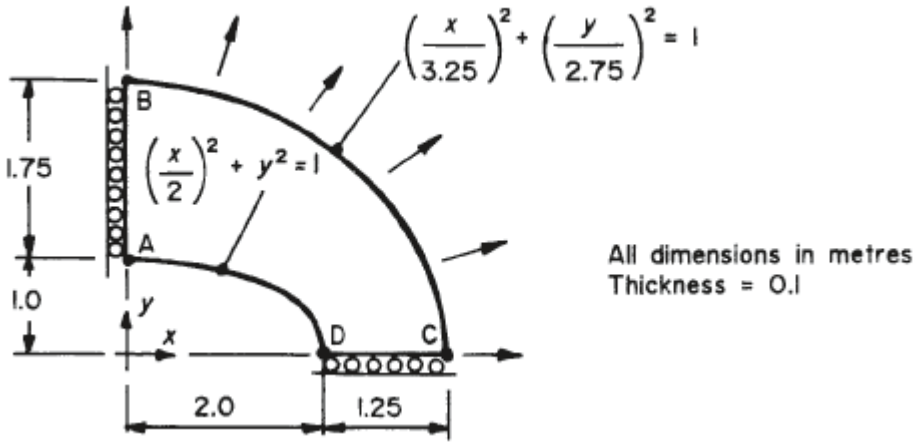


Figure 6.1-1 Elliptical membrane

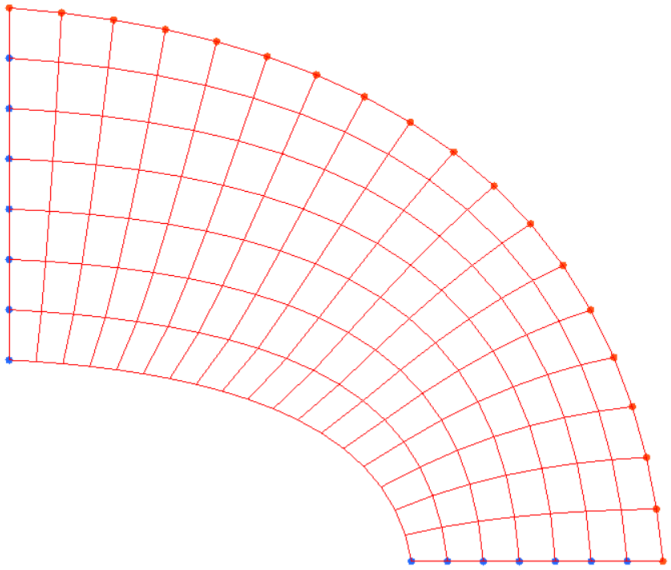


Figure 6.1-2 Elliptical membrane created in Patran using CQUAD4 elements

Model File Name and Location

Model File Name	Description	Location
`nafem_lin01.dat`	Linear Static Analysis of Elliptical membrane	\verifman\nafems

Results

Theoretical Results

Tangential edge stress (σ_{yy}) at D: 92.7 MPa

MSC Nastran results vs theoretical results are as follows:

Tangential Edge Stress (MPa)		
Theory	MSC Nastran	% Error
92.7	91.16	-1.661%

Conclusion

MSC Nastran results are comparable with theoretical results.

6.2 LE2: Cylindrical Shell Patch Test

Problem Description

Find the outer surface tangential (θ- θ) stress at E (refer below figure) of cylindrical shell with two load cases as follow.

1. Uniform normal edge moment, on DC of 1.0 kNm/m
2. Uniform outward normal pressure, at mid-surface ABCD, of 0.6 MPa and Tangential outward normal pressure, on edge DC, of 60.0 MPa.

Solution Number

SOL 101

Features Used

- Linear Static Analysis.

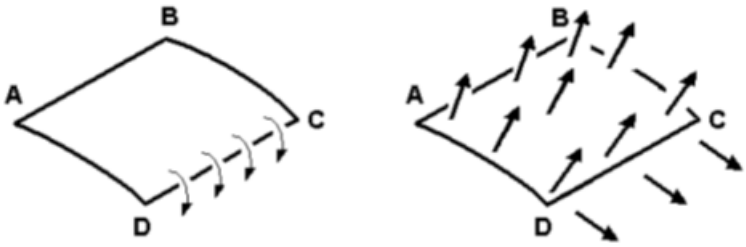
Reference

1. Test LE2 from NAFEMS report TSBM, Publication NNB, Rev. 3, NAFEMS Linear Benchmarks, 5 Oct 1990.

Modeling Techniques Used

Cylindrical shell is modeled using plane stress elements CQUAD4. GPSTRESS, STRESS and SURFACE are used to find out nodal stress output in cylindrical coordinate.

Loading



CASE1 (Left): Uniform normal edge moment, on DC, of 1.0 kNm/m

CASE2(Right): Uniform outward normal pressure, at mid-surface ABCD, of 0.6 MPa. Tangential outward normal pressure, on edge DC, of 60.0 MPa. The loading is shown in Figure 6.2-1.

Illustrations of the Model

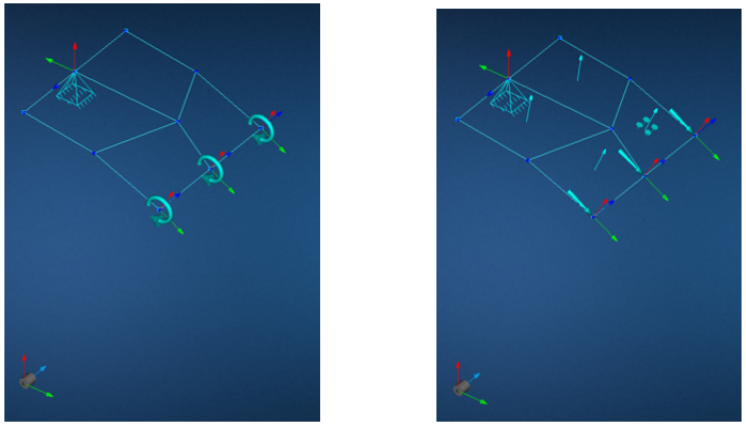


Figure 6.2-1 Finite element model for CASE1(left) and CASE2 (right)

Boundary Condition

Edge AB, all translations and rotations zero

Symmetric cylindrical shell about rθ plane, e.g. Z translations and normal rotations all zero. The boundary condition is shown in Figure 6.2-1.

Engineering Data

E = 210E+03 MPa and ν = 0.3

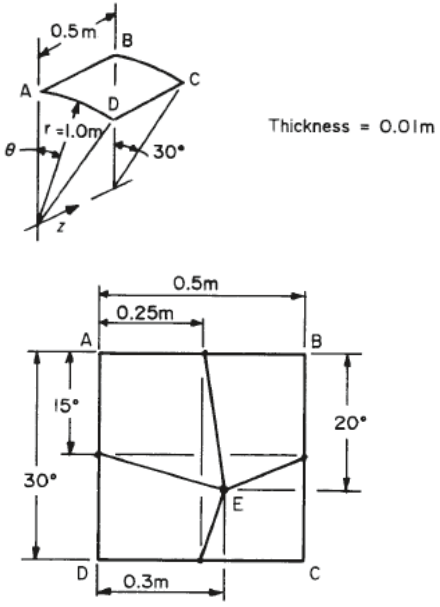


Figure 6.2-2 Cylindrical shell

Model File Name and Location

Model File Name	Description	Location
nafem_lin02.dat	Cylindrical Shell Patch Test	\verifman\nafems

Results

Theoretical Results

Outer Convex surface tangential stress at point E: 60 MPa

MSC Nastran results vs theoretical results are as follows:

	Tangential Edge Stress (MPa)		
	Theory	MSC Nastran	% Error
Case 1	60	59.58	-0.70%
Case 2	60	58.96	-1.73%

Conclusion

MSC Nastran results are comparable with theoretical results.

6.3 LE3: Hemisphere-Point Loads

Problem Description

Find the displacement in x- direction of point A of hemisphere shown in Figure 6.3-1.

Solution Number

SOL 101

Features Used

- Linear Static Analysis.

Reference

1. NAFEMS Finite Element Methods & Standards, The Standard NAFEMS Benchmarks, Test No. LE3. Glasgow: NAFEMS, Rev. 3, 1990.

Modeling Techniques Used

Hemisphere shell is modeled using plane stress elements CQUAD4.

Loading

Uniform normal pressure of 1 MPa on the upper surface of the plate.

Boundary Condition

Edge AB, all translations and rotations zero

Edge AD and edge BC, symmetry about r θ plane, e.g. Z translations and normal rotations all zero.

Engineering Data

E = 68.25E+03 MPa and $\nu = 0.3$

Illustrations of the Model

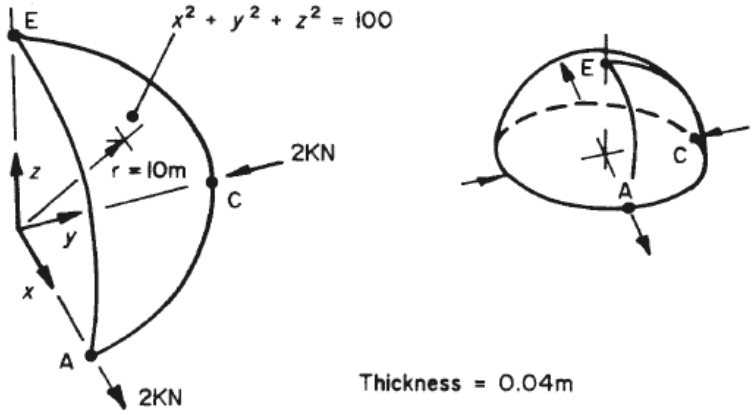


Figure 6.3-1 Hemisphere Shell

Model File Name and Location

Model File Name	Description	Location
`nafem_lin03.dat`	Hemisphere shell, point loads	\verifman\nafems

Results

Theoretical Results

X displacement at point A: 0.185 m

MSC Nastran results vs theoretical results are as follows:

X- displacement at Point A (m)		
Theory	MSC Nastran	% Error
0.185	0.17979	-2.81%

Conclusion

MSC Nastran results are comparable with theoretical results.

6.4 LE5: Z-section Cantilever

Problem Description

A z-section cantilever beam is subjected to a torsion load. Find the axial stress (X-X) at mid-surface at Point A shown in Figure 6.4-1.

Solution Number

SOL 101

Features Used

- Linear Static Analysis.

Reference

1. NAFEMS Finite Element Methods & Standards, The Standard NAFEMS Benchmarks, Test No. LE5. Glasgow: NAFEMS, Rev. 3, 1990.

Modeling Techniques Used

Z Shaped Cantilever Beam is modeled using plane stress elements CQUAD4.

Loading

Torque of 1.2 MNm applied at end x = 10 by two uniformly distributed edge shears, S = 0.6 at each flange.

Engineering Data

E = 210E+03 MPa

$\nu = 0.3$

Boundary Condition

Edge x = 0, all displacements are zero.

Illustrations of the Model

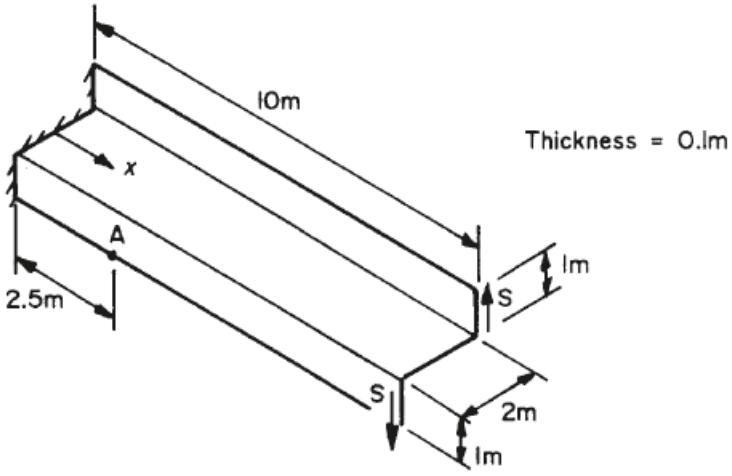


Figure 6.4-1 Z Shaped Cantilever Beam

Model File Name and Location

Model File Name	Description	Location
`nafem_lin05.dat`	Z- Section Cantilever, Torque	\verifman\nafems

Results

Theoretical Results

Axial (x-x) stress at mid-surface, point A: -108 MPa (Compression)

MSC Nastran results vs theoretical results are as follows:

Axial (x-x) stress at mid-surface, point A (MPa)		
Theory	MSC Nastran	% Error
-108	-103	-4.63%

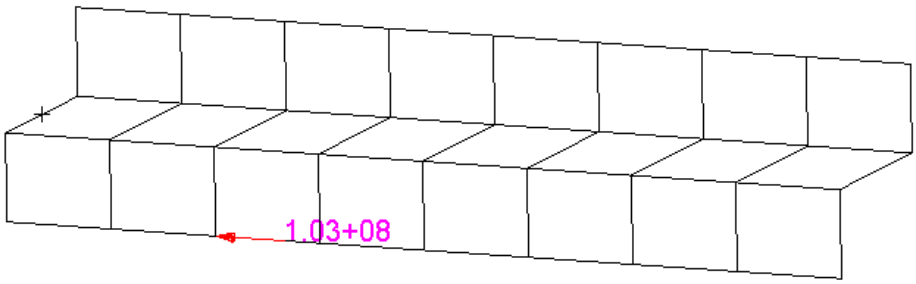


Figure 6.4-2 Axial Stress Result in X- direction of Z-section Cantilever Beam by Nastran

Conclusion

MSC Nastran results are comparable with theoretical results.

6.5 LE6: Skew Plate Normal Pressure

Problem Description

A skew plate is subjected uniform normal pressure in the vertical z- direction. Find the maximum principal at plate center.

Solution Number

SOL 400

Features Used

- Linear Static Analysis.

Reference

1. NAFEMS Finite Element Methods & Standards, The Standard NAFEMS Benchmarks, Test No. LE6. Glasgow: NAFEMS, Rev. 3, 1990.

Modeling Techniques Used

Skew plate is modeled using different shell elements. Refer to the Results section.

Loading

Normal pressure of -0.7KPa in vertical Z-direction.

Engineering Data

E = 210E+03 MPa and ν = 0.3

Boundary Condition

Simple supports (no Z-displacement) for all edges AB, BC, CD, DA.

Illustrations of the Model

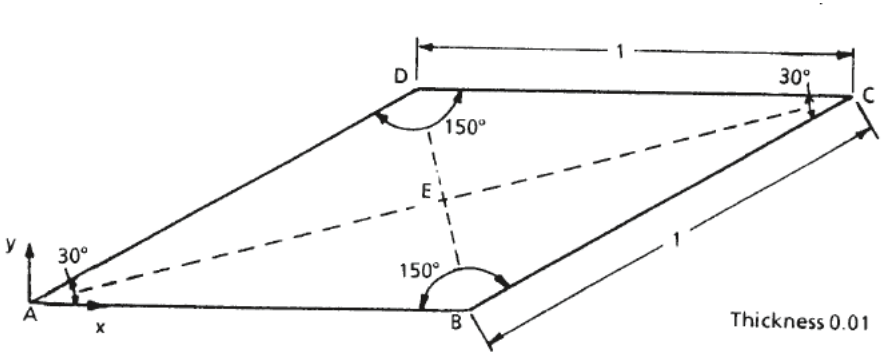


Figure 6.5-1 Skew Plate

Model File Name and Location

Model File Name	Description	Location
nafem_lin06q4.dat	Skew Plate, Pressure, CQUAD4	tpl\verifman\nafems
nafem_lin06q8.dat	Skew Plate, Pressure, CQUAD8	tpl\verifman\nafems
nafem_lin06q8a.dat	Skew Plate, Pressure, CQUAD8 Advance	tpl\verifman\nafems
nafem_lin06qr.dat	Skew Plate, Pressure, CQUADR	tpl\verifman\nafems

Results

Theoretical Results

Maximum principal stress on the lower surface at the plate center: 0.802 MPa

MSC Nastran results vs theoretical results are as follows:

Elements	Maximum Principal Stress, at point E (MPa)			
	CQUAD4	CQUAD8	CQUAD8 Advance	CQUADR
Theory	0.802	0.802	0.802	0.802
MSC Nastran	0.768	0.742	0.778	0.805
% Error	-4.30%	-7.50%	-3.0%	0.4%

Conclusion

MSC Nastran result Maximum principal stress at Center of plate for decks with different elements CQUAD4, CQUAD8, CQUAD8 with advance and CQUADR are comparable with Theoretical results.

6.6 LE7: Axisymmetric Cylinder/Sphere-Pressure

Problem Description

Axisymmetric cylinder-sphere shell is subjected uniform internal pressure. Find the axial stress σ_{yy} on the outer surface of the upper cylinder at D.

Solution Number

SOL 400

Features Used

- Linear Static Axisymmetric.

Reference

1. NAFEMS Finite Element Methods & Standards, The Standard NAFEMS Benchmarks, Test No. LE7. Glasgow: NAFEMS, Rev. 3, 1990.

Modeling Techniques Used

Cylindrical-Sphere Shell is modeled using axisymmetric elements CAXISYM.

Boundary Condition

Point A, zero radial displacement and zero rotation. Point F, zero y-displacement.

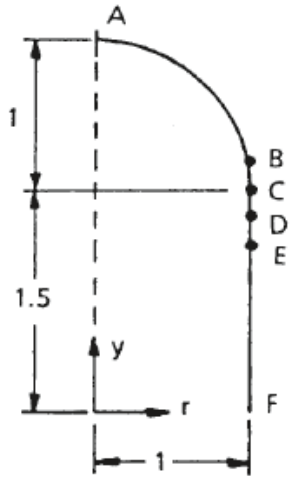
Loading

Uniform internal pressure of 1 MPa.

Engineering Data

E = 210E+03 MPa and $\nu = 0.3$

Illustrations of the Model



Units M, KN

Thickness = 0.025

Point	x	y
B	0.9814	1.6920
D	1.0	1.4034
E	1.0	1.1136

Figure 6.6-1 Z Shaped Cantilever Beam

Model File Name and Location

Model File Name	Description	Location
`nafem_lin07.dat`	Axisymmetric cylinder-sphere shell, uniform pressure	\verifman\nafems

Results

Theoretical Results

Axial (σ_{yy}) stress on the outer surface of the upper cylinder, at point D: 25.86 MPa

Nastran result is calculated using interpolation formula.

MSC Nastran results vs theoretical results are as follows:

Axial (x-x) stress at mid-surface, point A (MPa)		
Theory	MSC Nastran	% Error
25.86	25.6866	-0.67%

Conclusion

MSC Nastran results are comparable with theoretical results.

6.7 LE8: Axisymmetric Shell Pressure

Problem Description

Axisymmetric shell is subjected uniform internal pressure. Find the hoop stress on the outer surface of the shell at D.

Solution Number

SOL 400

Features Used

- Linear Static Axisymmetric.

Reference

1. NAFEMS Finite Element Methods & Standards, The Standard NAFEMS Benchmarks, Test No. LE8. Glasgow: NAFEMS, Rev. 3, 1990.

Modeling Techniques Used

Shell is modeled using axisymmetric elements CAXISYM. The model details are given below:

E = 210E+03 MPa; ν = 0.3

Loading

Uniform internal pressure of 1 MPa.

Boundary Condition

Point A, zero y displacement. Point E, zero radial displacement and zero rotation.

Illustrations of the Model

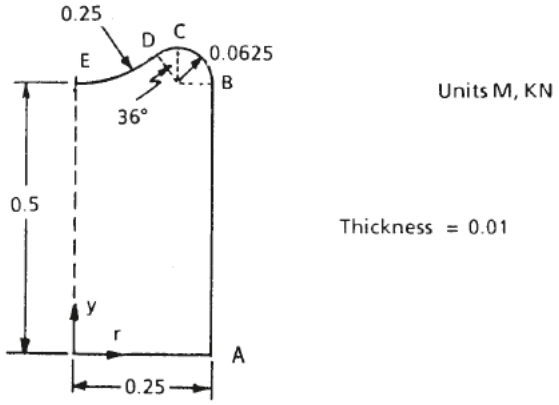


Figure 6.7-1 Axisymmetric Shell

Model File Name and Location

Model File Name	Description	Location
nafem_lin08.dat	Axisymmetric shell, uniform pressure	\verifman\nafems

Results

Theoretical Solution

Hoop stress (σ_{tt}) stress on the outer surface of shell, at point D: 94.55 MPa

Nastran result is calculated using extrapolation formula:

Hoop Stress at node 33: $\frac{(83.97 + 110.14)}{2} = 97.055$ MPa

Result comparison of theoretical solution and MSC Nastran results are as follows:

Hoop stress σ_{tt}, at point D (MPa)		
Theory	MSC Nastran	% Error
94.55	97.055	2.64%

Conclusion

MSC Nastran results are comparable with theoretical results.

6.8 LE9: Axisymmetric Branched Shell-Pressure

Problem Description

Axisymmetric branched shell is subjected uniform internal pressure over edge BCD shown below in figure. Find the axial stress σ_{yy} on the outer surface of the upper cylinder at C.

Solution Number

SOL 400

Features Used

- Linear Static Axisymmetric.

Reference

1. NAFEMS Finite Element Methods & Standards, The Standard NAFEMS Benchmarks, Test No. LE9. Glasgow: NAFEMS, Rev. 3, 1990.

Modeling Techniques Used

Cylindrical Shell is modelled using axisymmetric elements CAXISYM. The model details are given below:

E	=	210E+03 MPa
υ	=	0.3

Loading

Uniform internal pressure of 1 MPa over edge BCD

Boundary Condition

Point A, fully fixed

Illustrations of the Model

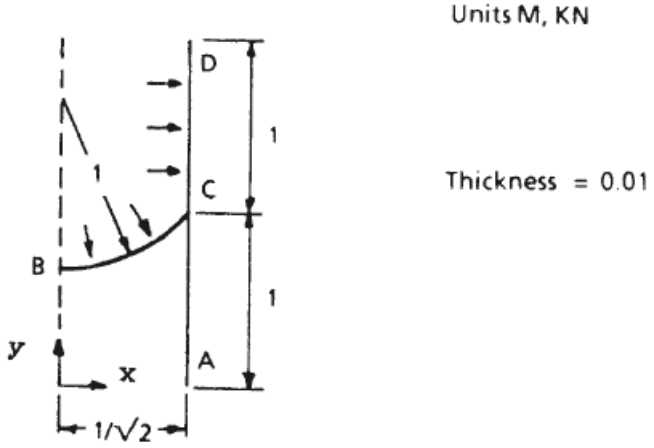


Figure 6.8-1 Axisymmetric Shell

Model File Name and Location

Model File Name	Description	Location
nafem_lin09.dat	Axisymmetric branched shell, uniform pressure	\verifman\nafems

Results

Theoretical Solution

Axial (σ_{yy}) stress on the outer surface of the upper cylinder, at point C: -319.9 MPa

Nastran result is calculated using extrapolation formula as follows:

Axial Stress at node 33: $139.515 + \left(\frac{0.0234375}{0.015625} \times (248.565 - 139.515)\right) = 303.09$ MPa

Result comparison of theoretical solution and MSC Nastran results are as follows:

Axial stress (σ_{yy}), at point C (MPa)		
Theory	MSC Nastran	% Error
-319.19	-303.09	-5.25%

Conclusion

MSC Nastran results are comparable with theoretical results.

6.9 LE10: Thick Plate Pressure

Problem Description

A elliptical thick plate is subjected uniform normal pressure on the upper surface of the plate. Find the direct stress σ_{yy} at Point D shown in Figure 6.9-1.

Solution Number

SOL 101

Features Used

- Linear Static Analysis.

Reference

1. NAFEMS Finite Element Methods & Standards, The Standard NAFEMS Benchmarks, Test No. LE10. Glasgow: NAFEMS, Rev. 3, 1990.

Modeling Techniques Used

Thick elliptical plate is modelled using solid elements CHEXA. The model details are given below:

E	=	210E+03 MPa
ν	=	0.3

Loading

Uniform normal pressure of 1 MPa on the upper surface of the Plate

Boundary Condition

Face DCD'C' zero y-displacement

Face ABA'B' zero x- displacement

Face BCB'C' x and y displacements fixed, z displacements fixed along mid-plane

Illustrations of the Model

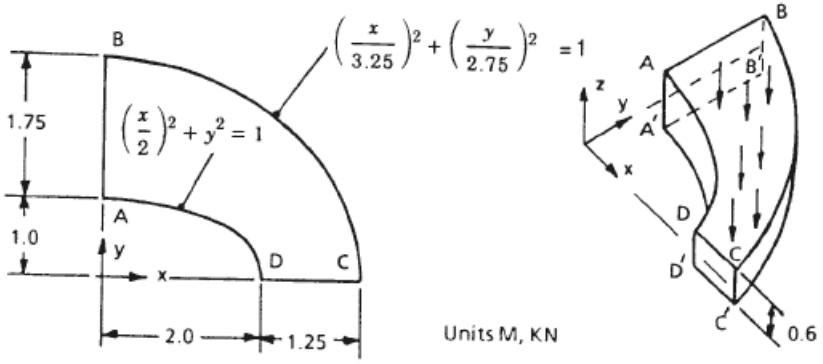


Figure 6.9-1 Axisymmetric Shell

Model File Name and Location

Model File Name	Description	Location
`nafem_lin10.dat`	Elliptical Thick Plate, Pressure	\verifman\nafems

Results

Theoretical Solution

Normal (σ_{yy}) stress at point D: 5.38 MPa

Result comparison of theoretical solution and MSC Nastran results are as follows:

Normal stress (σ_{yy}), at point D (MPa)			
Theory	MSC Nastran	% Error	Element
5.38	6	11.52%	HEXA 8
5.38	6.36	18.21%	HEXA 20

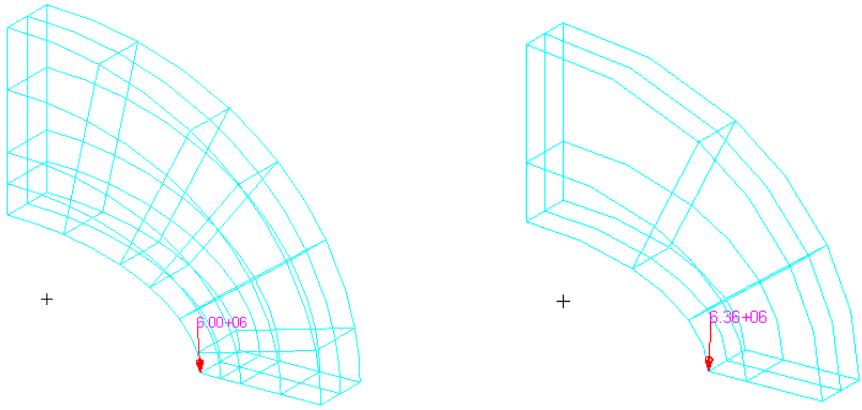


Figure 6.9-2 a) Model with CHEXA element with 8 node b) Model with CHEXA element with 20 node

Conclusion

MSC Nastran results are comparable with theoretical results.

6.10 LE11: Solid Cylinder/ Taper/ Sphere-Temperature

Problem Description

A solid spherical taper cylinder is subjected to temperature loading. Find the direct stress σ_{zz} at Point A shown in below figure.

Solution Number

SOL 101

Features Used

- Linear Static Analysis.

Reference

1. NAFEMS Finite Element Methods & Standards, The Standard NAFEMS Benchmarks, Test No. LE11. Glasgow: NAFEMS, Rev. 3, 1990.

Modeling Techniques Used

Solid cylinder is modeled using solid elements CHEXA. The model details are given below:

E	=	210E+03 MPa
ν	=	0.3
a	=	2.3E-04 /^{0}C

Loading

Linear temperature gradient in the radial and axial direction T ($^\circ$C) = $\sqrt{(x^2+y^2)}+z$

Boundary Condition

Symmetry on x-z plane i.e., zero Y-displacement

Symmetry on y-z plane i.e., zero X- displacement

Face on x-y plane zero Z- displacement

Face HIH'I' zero Z- displacement

Main Index

Illustrations of the Model

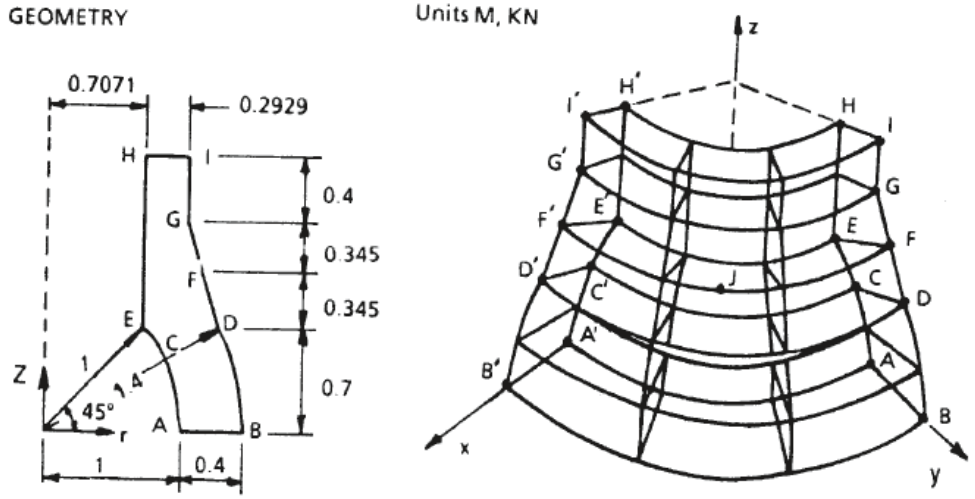


Figure 6.10-1 Spherical Taper Cylinder

Model File Name and Location

Model File Name	Description	Location
`nafem_lin11.dat`	Spherical taper cylinder, temperature	\verifman\nafems

Results

Theoretical Solution

Direct stress (σ_{zz}) at point A: -105 MPa

Result comparison of theoretical solution and MSC Nastran results are as follows:

Direct stress (σ_{zz}), at point A (MPa)		
Theory	MSC Nastran	% Error
-105	-99.477	-5.26%

Conclusion

MSC Nastran results are comparable with theoretical results.

Linear Thermo Elastic

6.11 T1: Membrane with Hot Spot

Problem Description

Find the direct stress in y- direction at point D in membrane as shown below figure which is subjected to thermal load (hot-spot) at center.

Solution Number

SOL 400

Features Used

- Heat Transfer

Reference

1. NAFEMS Finite Element Methods & Standards, The Standard NAFEMS Benchmarks, Test No. T1. Glasgow: NAFEMS, Rev. 3, 1990.

Modeling Techniques Used

Membrane is modeled using 2D element CQUAD4. The model details are given below:

Isotropic, E = 210E+03 MPa

ν = 0.3

Loading

Within hot-spot ($0 \le r \le 1.0$ mm)

Thermal strain (αT) = 1.0×10^{-3}

Outboard of hot-spot ($r > 1.0$ mm)

Thermal strain (αT) = 0

Boundary Condition

On quarter model,

at y=0, symmetric about x, e.g. y displacement zero

at x=0, symmetric about y, e.g. x displacement zero

Illustrations of the Model

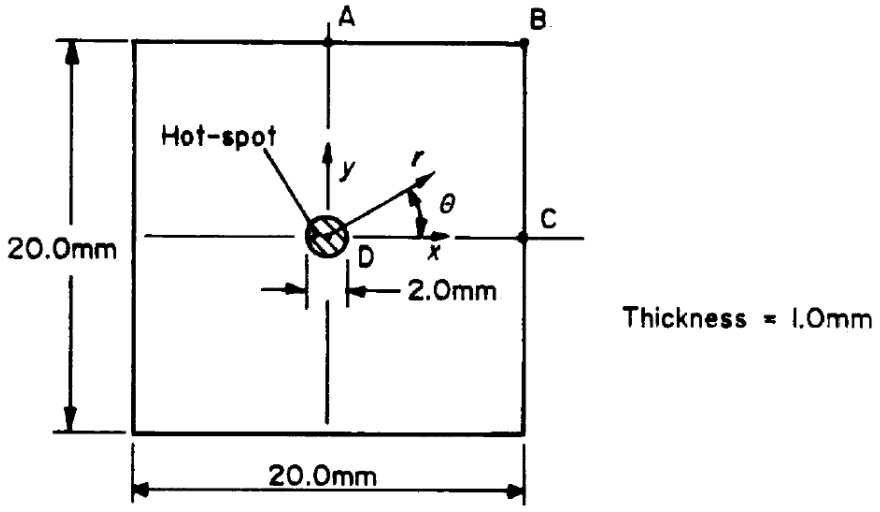
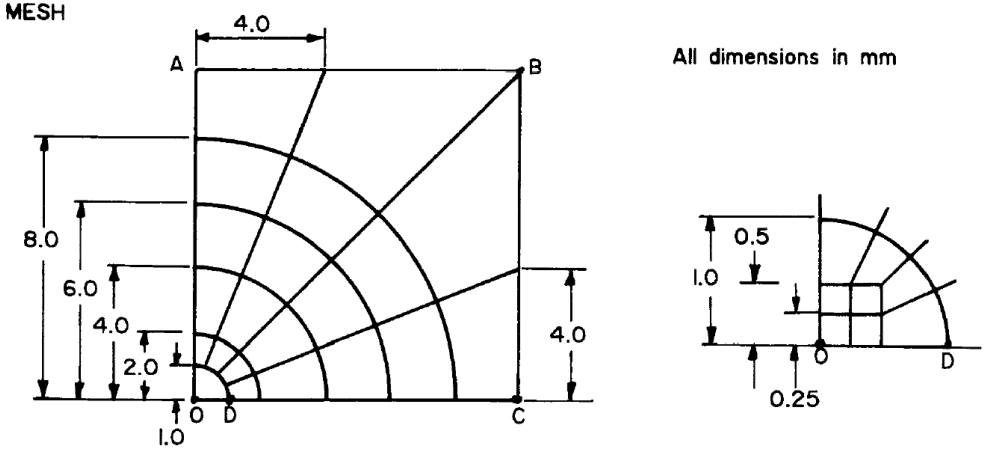


Figure 6.11-1 Membrane with Hot Spot



Model File Name and Location

Model File Name	Description	Location
nafem_te01.dat	Membrane with Hot Spot	\verifman\nafems

Results

Theoretical Solution

Direct stress at Point D in Y direction: 50 MPa

Result comparison of theoretical solution and MSC Nastran results are as follows:

Direct stress at D in Y- Direction (MPa)		
Theory	MSC Nastran	% Error
50	46.5	-7.0%

Conclusion

MSC Nastran results are comparable with theoretical results.

Heat Transfer

6.12 T2: One Dimensional Heat Transfer with Radiation

Problem Description

Find the material temperature at end of the bar when it is subjected to thermal load at other end.

Solution Number

SOL 400

Features Used

- Heat Transfer

Reference

1. NAFEMS Finite Element Methods & Standards, The Standard NAFEMS Benchmarks, Test No. T2. Glasgow: NAFEMS, Rev. 3, 1990.

Modeling Techniques Used

Bar is modeled using 2D element CBAR. The model details are given below:

Conductivity	=	55.6 W/m^0C
Specific Heat	=	460.0 J/Kg^0C
Density	=	7850 Kg/m^3

At boundary B

Emissivity	=	0.98
Stefan-Boltzman Constant	=	5.67 x 10-8 $Wm^2\ ^0K^{-4}$

Loading

Zero internal heat generation

Boundary Condition

At A, prescribed temperature of 1000 ^0K

At B, radiation to ambient temperature of 300 ^{0}K

No heat flux perpendicular to AB

Illustrations of the Model

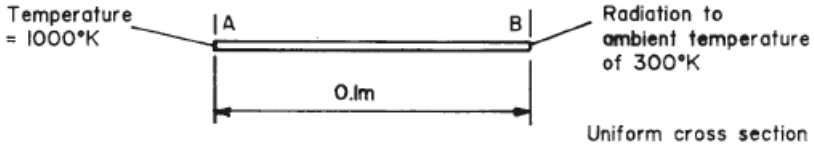


Figure 6.12-1 Bar

Model File Name and Location

Model File Name	Description	Location
nafem_te02.dat	One Dimensional Heat Transfer with Radiation	\verifman\nafems

Results

Theoretical Solution

Material temperature at B: 927 ^{o}K

Result comparison of theoretical solution and MSC Nastran results are as follows:

Temperature at B in K		
Theory	MSC Nastran	% Error
927	912.7	-1.54%

Conclusion

MSC Nastran results are comparable with theoretical results.

6.13 T3: One Dimensional Transient Heat Transfer

Problem Description

Find the Material Temperature at point C of the bar when it is subjected to transient thermal load at B. (Refer Figure 6.13-1)

Solution Number

SOL 400

Features Used

- Transient Heat Conduction

Reference

1. NAFEMS Finite Element Methods & Standards, The Standard NAFEMS Benchmarks, Test No. T3. Glasgow: NAFEMS, Rev. 3, 1990.

Modeling Techniques Used

Bar is modeled using 2D element CROD. The model details are given below:

Conductivity	=	35.0 W/m^0C
Specific Heat	=	440.5 J/Kg^0C
Density	=	7200 Kg/m^3

Loading

Zero internal heat generation

Boundary Condition

At time t = 0, All temperature = 0 ^0C

At time t > 0, All temperature = 0 ^0C

At B, temperature = $100 \sin \frac{nt}{40}$ ^0C

No heat flux perpendicular to AB

Illustrations of the Model

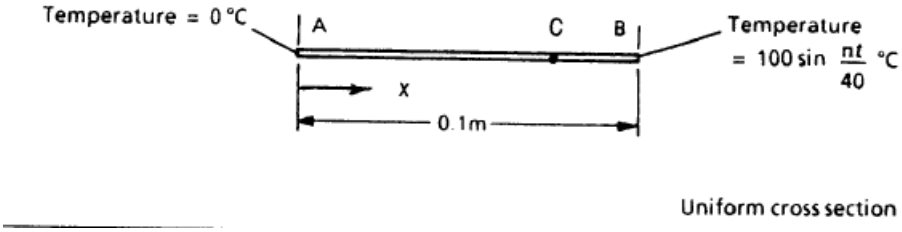


Figure 6.13-1 Bar Subjected to Transient Load

Model File Name and Location

Model File Name	Description	Location
`nafem_te03.dat`	One Dimensional Transient Heat Transfer	\verifman\nafems

Results

Theoretical Solution

Material temperature at C: 36.60 ^{o}C

Result comparison of theoretical solution and MSC Nastran results are as follows:

Material Temperature at Point C		
Theory	MSC Nastran	% Error
36.60	34.85	-4.78%

Conclusion

MSC Nastran results are comparable with theoretical results.

6.14 T4: Two Dimensional Heat Transfer with Convection

Problem Description

Natural Convection heat transfer occurs in one edge insulted plate which is subjected to thermal loading at one edge. Find the temperature at point E. (Refer Figure 6.14-1)

Solution Number

SOL 400

Features Used

- Heat Conduction

Reference

1. NAFEMS Finite Element Methods & Standards, The Standard NAFEMS Benchmarks, Test No. T4. Glasgow: NAFEMS, Rev. 3, 1990.

Modeling Techniques Used

Plate is modeled using 2D elements CQUAD4. The model details are given below:

Conductivity	=	52.0 W/m^0C
Surface convective heat transfer coefficient (edges BC, CD)	=	750.0 $W/m^{2\,0}C$

Loading

Zero internal heat generation

Boundary Condition

Edge AB, temperature = 100 ^0C

Edge DA, Zero heat flux

Edges BC, CD, convection to ambient temperature = 0 ^0C

Illustrations of the Model

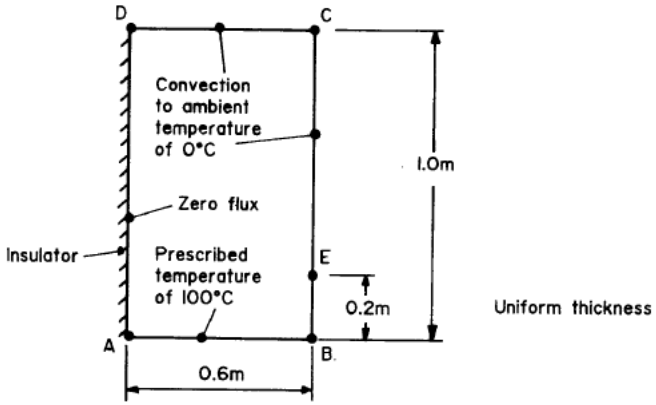


Figure 6.14-1 Insulated Plate

Model File Name and Location

Model File Name	Description	Location
nafem_te04.dat	Two Dimensional Transient Heat Transfer	\verifman\nafems

Results

Theoretical Solution

Material temperature at E: 18.3 ^{o}C

Result comparison of theoretical solution and MSC Nastran results are as follows:

Material Temperature at Point E		
Theory	MSC Nastran	% Error
18.3	19.3	5.46%

Conclusion

MSC Nastran results are comparable with theoretical results.

Free Vibration

6.15 FV2- Pin-Ended Double Cross: In-Plane Vibration

Problem Description

NAFEMS Benchmark FV2 - pin-ended double cross normal modes

Solution Number

SOL 103

Features Used

- Normal modes
- Beam and bar elements (Euler and considering shear flexibility)
- Coupling between flexural and extensional behavior
- Repeated eigenvalues

Reference

1. Problem FV2 from The Standard NAFEMS Benchmarks, Oct 5, 1990.

Modeling Techniques Used

- This is a simple model using beam or bar elements. Coordinate systems are used to simplify the modeling.
- Modeled using 2- and 3-noded beam elements.

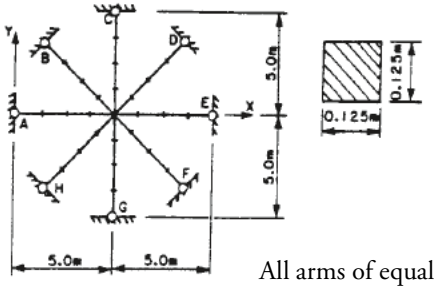
Illustrations of the Model

From the NAFEMS reference:

Geometry and Mesh

Exact beam: 4 elements per arm

Iso-parametric beam: 4 elements per arm



All arms of equal length

Boundary Conditions:

- Out-of plane motion is constrained at all nodes.
- Nodes A through H are constrained for in-plane translation.

Material Properties

$E = 200 \times 10^9 N/m^2$

$\rho = 8000 kg/m^3$

Model File Name and Location

Model File Name	Description	Location
fv2_2e.dat	2-noded Euler beams	\verifman\nafems
fv2_2ec	2-noded Euler beams with coupled mass	\verifman\nafems
fv2_2s.dat	2-noded beams with shear flexibility	\verifman\nafems
fv2_3e.dat	3-noded Euler beams	\verifman\nafems
fv2_3s.dat	3-noded beams with shear flexibility	\verifman\nafems

Results

Natural frequencies (Hz):

Mode	NAFEMS Value	2-Noded Euler Beams	2-Noded Euler Beams using Coupled Mass	2-Noded Beams with Shear Flexibility	3-Noded Euler Beams
1	11.336	11.333	11.339	11.323	11.337
2 and 3	17.687	17.662	17.692	17.621	17.685
4 through 8	17.715	17.69	17.720	17.650	17.715
9	45.477	45.016	45.524	45.841	45.856
10 and 11	57.364	56.058	57.427	55.682	58.043
12 through 16	57.683	56.343	57.748	55.961	58.387

Conclusion

Although not specifically mentioned, the NAFEMS values are obviously based on Euler beam theory and ignore the shear flexibility. If Euler beams are used, the results compare well with the NAFEMS expected values. If shear flexibility is included, then the natural frequencies are slightly lower.

6.16 FV4 - Cantilever with Off-Centre Point Masses

Problem Description

Calculation of modes of a cantilever beam with offset masses at the free end.

Solution Number

SOL 103 - normal modes

Features Used

- BAR elements (Euler beam and inclusion of shear flexibility)
- 3-noded Beam elements
- CONM2 with offset
- Normal modes
- Coupling between bending and torsion
- Close eigenvalues

Reference

1. NAFEMS Finite Element Methods & Standards. Abbassian, F., Dawswell, D. J., and Knowles, N. C.Selected Benchmarks for Natural Frequency Analysis, Test No. 4. Glasgow: NAFEMS, Nov., 1987.

Modeling Techniques Used

- Simple cantilever beam model.
- CONM2's with offsets used to model offset masses.

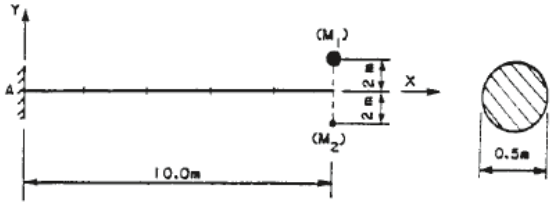
Illustrations of the Model

Geometry and Mesh

Exact beam: 5 elements along cantilever

Iso-parametric beam: 3 elements along cantilever

M_1 = 10000 kg (along X, Y, Z)
M_2 = 1000 kg (along X, Y, Z)



Model File Name and Location

Model File Name	Description	Location
fv4_2e.dat	2-noded Euler beams	\verifman\nafems
fv4_2ec	2-noded Euler beams with coupled mass	\verifman\nafems
fv4_2s.dat	2-noded beams with shear flexibility	\verifman\nafems
fv4_3e.dat	3-noded Euler beams	\verifman\nafems
fv2_3s.dat	3-noded beams with shear flexibility	\verifman\nafems

Results

Natural frequencies (Hz) of the first 6 modes:

Mode	NAFEMS Value	2-Noded Euler Beams	2-Noded Euler Beams using Coupled Mass	2-Noded Beams with Shear Flexibility	3-Noded Euler Beams
1	1.723	1.715	1.723	1.713	1.722
2	1.727	1.719	1.727	1.718	1.726
3	7.413	7.428	7.45	7.415	7.423
4	9.972	9.978	9.975	9.952	9.965
5	18.155	17.748	18.205	17.658	18.226
6	26.957	27.058	27.001	26.762	27.188

Conclusion

While not an exact match, the calculated natural frequencies agree well with the expected values.

6.17 FV5 - Deep Simply-supported Beam

Problem Description

Calculation of modes of a simply supported beam

Solution Number

SOL 103 - normal modes

Features Used

- BEAM elements (Timoshenko beam formulation)
- 2- and 3-noded Beam elements
- Normal modes - Lanczos method
- Shear deformation and torsion

Reference

1. NAFEMS Finite Element Methods & Standards. Abbassian, F., Dawswell, D. J., and Knowles, N. C.Selected Benchmarks for Natural Frequency Analysis, Test No. 5. Glasgow: NAFEMS, Nov., 1987.

Modeling Techniques Used

- Simply-supported beam model using BEAM elements.

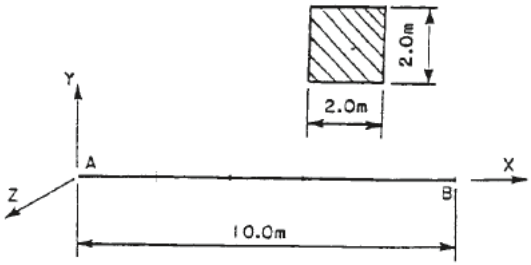
Illustrations of the Model

From the NAFEMS reference:

Geometry and Mesh

Exact beam: 5 elements

Iso-parametric beam: 5 elements



Model File Name and Location

Model File Name	Description	Location
`fv5_2.dat`	2-noded beams using lumped mass	\verifman\nafems
`fv5_2c`	2-noded beams using coupled mass	\verifman\nafems
`fv5_3.dat`	3-noded beams	\verifman\nafems

Results

Natural frequencies (Hz) of the first 9 modes

Mode	NAFEMS Value	2-Noded Beams using lumped mass	2-Noded Beams using Coupled Mass	3-Noded Euler Beams
1 and 2	42.649	43.111	43.226	38.635
3	77.542	77.204	77.842	77.524
4	125.0	124.487	125.515	125.002
5 and 6	148.31	149.393	154.519	172.888
7	233.1	224.056	241.242	232.811
8 and 9	284.55	269.578	305.392	253.929

Conclusion

While not an exact match, the calculated natural frequencies agree well with the expected values. While the bending modes for the 3-noded beam are different than those of the 2-noded beams, they agree well with the modes one obtains if the models using 2-noded beams are refined to have 10 elements.

6.18 FV12- Free Thin Square Plate

Problem Description

Out of plane free vibration of a square plate with in-plane motion constrained.

Solution Number

SOL 103 - normal modes

Features Used

- Rigid-body modes (3)
- Repeated eigenvalues

Reference

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., Selected Benchmarks for Natural Frequency Analysis, Test No. 12. Glasgow: NAFEMS, Nov., 1987.

Modeling Techniques Used

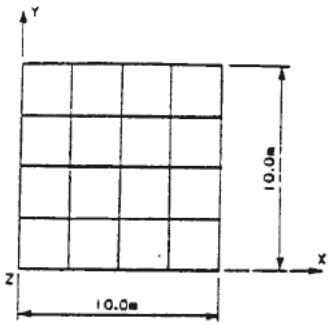
- A simple flat plate model is created using either QUAD4 or QUAD8 elements and the out-of plane modes are calculated.

Illustrations of the Model

Geometry and Mesh

H.O.E. 4 × 4 (as shown)

L.O.E. 8 × 8



t = 0.05 m

Boundary Conditions

x- and y-translation and z-rotation are constrained to 0.0

Material Properties

Young's modulus, $E = 200 \times 10^9 N/m^2$

Poisson's ratio, $\nu = 0.3$

Model File Name and Location

Model File Name	Description	Location
`fv12_q4.dat`	QUAD4 with lumped mass	\verifman\nafems
`fv12_q4c.dat`	QUAD4 with coupled mass	\verifman\nafems
`fv12_q8.dat`	QUAD8 with lumped mass	\verifman\nafems
`fv12_q8c.dat`	QUAD8 with coupled mass	\verifman\nafems

Results

Natural frequencies (Hz) of modes 4 through 10 (modes 1-3 are rigid-body modes).

QUAD4 results

Mode	NAFEMS Theoretical Value	NAFEMS Expected Value	QUAD4 Lumped Mass	QUAD4 Coupled Mass
4	1.622	1.532	1.578	1.624
5	2.36	2.356	2.241	2.389
6	2.922	2.831	2.804	2.979
7 and 8	4.233	4.122	3.931	4.237
9	7.416	7.363	6.823	7.79
10	n.a.	7.392	6.823	7.79

QUAD8 results

Mode	NAFEMS Theoretical Value	NAFEMS Expected Value	QUAD8 Lumped Mass	QUAD8 Coupled Mass
4	1.622	1.632	1.585	1.619
5	2.36	2.404	2.233	2.363
6	2.922	3.006	2.807	2.929
7 and 8	4.233	4.251	3.945	4.159
9	7.416	7.589	6.814	7.478
10	n.a.	8.027	6.814	7.478

Conclusion

As can be expected, the results using coupled mass are closer to the theoretical values than those using lumped mass.

6.19 FV15- Clamped Thin Rhombic Plate

Problem Description

Solution to find the first 6 modes of a clamped flat plate, which is skewed 45 degrees.

Solution Number

SOL 103 normal modes

Features Used

- Lanczos eigenvalue solution

Reference

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., Selected Benchmarks for Natural Frequency Analysis, Test No. 15. Glasgow: NAFEMS, Nov., 1987.

Modeling Techniques Used

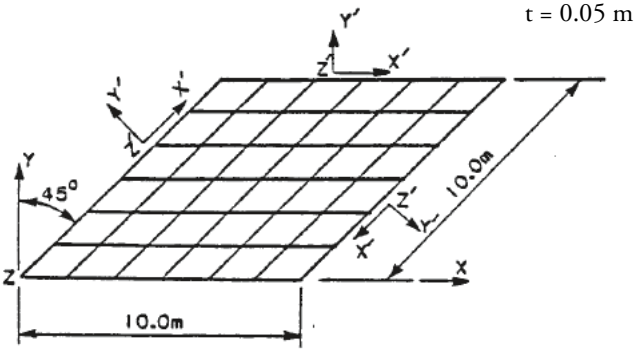
- Flat QUAD4 and QUAD8 elements modeling a clamped plate.

Illustrations of the Model

Geometry and Mesh

H.O.E. 6 × 6 (as shown)

L.O.E. 12 × 12



Boundary Conditions

Clamped edges

Model File Name and Location

Model File Name	Description	Location
`fv15_q4.dat`	QUAD4 with lumped mass	\verifman\nafems
`fv15_q4c.dat`	QUAD4 with coupled mass	\verifman\nafems
`fv15_q8.dat`	QUAD8 with lumped mass	\verifman\nafems
`fv15_q8c.dat`	QUAD8 with coupled mass	\verifman\nafems

Results

QUAD4 results

Mode	NAFEMS Theoretical Value	NAFEMS Expected Value	QUAD4 Lumped Mass	QUAD4 Coupled Mass
1	7.938	8.142	7.832	7.967
2	12.835	13.891	12.851	13.400
3	17.941	20.035	18.078	19.261
4	19.133	20.165	18.585	19.338
5	24.009	27.950	24.310	26.837
6	27.922	32.046	27.644	29.702

QUAD8 results

Mode	NAFEMS Theoretical Value	NAFEMS Expected Value	QUAD8 Lumped Mass	QUAD8 Coupled Mass
1	7.938	7.873	7.992	8.012
2	12.835	12.480	13.051	13.177
3	17.941	17.312	18.667	19.112
4	19.133	18.738	19.238	19.383
5	24.009	22.704	25.927	27.081
6	27.922	25.883	28.421	29.060

Conclusion

In all but one case (QUAD8, coupled mass, mode 5) the results are closer to the theoretical values than the expected results. It should be noted that a 45 degree skew angle is well beyond the recommended maximum skew of 30 degrees, but the elements still performed well.

6.20 FV16- Cantilevered Thin Plate

Problem Description

Normal modes of a cantilevered thin plate modeled using QUAD8 elements.

Solution Number

SOL 103 - normal modes using Lanczos method.

Features Used

- Normal modes calculation using QUAD8 elements, distorted in-plane.

Reference

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., Selected Benchmarks for Natural Frequency Analysis, Test No. 16. Glasgow: NAFEMS, Nov., 1987.

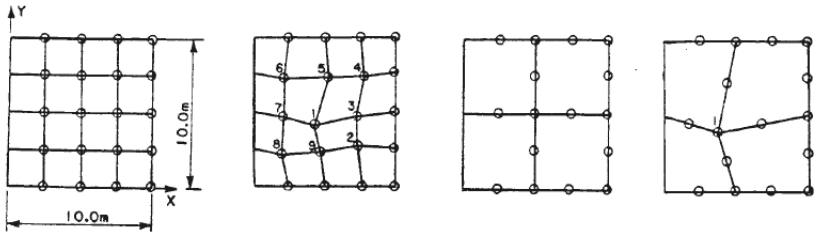
Modeling Techniques Used

- A simple flat plate model created using QUAD8 elements (NAFEMS lists 8-noded semi-loof elements, which do not exist in MSC Nastran). Four different versions are created, using different mesh refinement and distortion. The first 6 out-of-plane modes are calculated.

Illustrations of the Model

The numbering and locations shown for Test2 are not consistent with the axes and diagrams in the reference. As such, the locations have been applied to the closest GRID points to the offsets. Also, although it is not mentioned in the reference, straight lines are drawn between the offset GRID points, so the mid-side nodes will also be offset to maintain straight edges.

Illustrations of models from the reference:



Test 1 Test 2 Test 3 Test 4

Locations of Points from the reference (and corrections):

Node No.	Coordinates (m)	
	x	y
1	4.0	4.0
2	2.25	2.25
3	4.75	2.5
4	7.25	2.75
5	7.5	4.75
6	7.75	7.25
7	5.25	7.25
8	2.25	7.25
9	2.5	4.75

Table 6-1 Locations for Test 2

Node No.	Coordinates (m)	
	x	y
1	4.0	4.0

Table 6-2 Locations for Test 4

Correct locations in illustration for Test 2:

NAFEMS Node No.	Correct Node No.
1	1
2	8
3	9
4	2
5	3
6	4
7	5
8	6
9	7

Model File Name and Location

Model File Name	Description	Location
Fv16_t1.dat	Test 1 - CQUAD8, lumped mass	\verifman\nafems
Fv16_t1c.dat	Test 1 - CQUAD8, coupled mass	\verifman\nafems
Fv16_t2.dat	Test 2 - CQUAD8, lumped mass	\verifman\nafems
Fv16_t2c.dat	Test 2 - CQUAD8, coupled mass	\verifman\nafems
Fv16_t3.dat	Test 3 - CQUAD8, lumped mass	\verifman\nafems
Fv16_t3c.dat	Test 3 - CQUAD8, coupled mass	\verifman\nafems
Fv16_t4.dat	Test 4 - CQUAD8, lumped mass	\verifman\nafems
Fv16_t4c.dat	Test 4 - CQUAD8, coupled mass	\verifman\nafems

Results

Test 1 results:

Mode	NAFEMS Expected Value	QUAD8 Lumped Mass	QUAD8 Coupled Mass
1	0.421	0.4153	0.4183
2	1.029	1.005	1.023
3	2.582	2.485	2.579
4	3.306	3.132	3.298
5	3.753	3.622	3.765
6	6.555	6.292	6.719

Test 2 results:

Mode	NAFEMS Expected Value	QUAD8 Lumped Mass	QUAD8 Coupled Mass
1	0.421	0.4153	0.4193
2	1.029	1.007	1.026
3	2.582	2.509	2.604
4	3.306	3.164	3.321
5	3.753	3.664	3.815
6	6.555	6.327	6.801

Test 3 results:

Mode	NAFEMS Expected Value	QUAD8 Lumped Mass	QUAD8 Coupled Mass
1	0.421	0.4069	0.4186
2	1.029	0.9648	1.021
3	2.582	2.200	2.662
4	3.306	2.894	3.402
5	3.753	3.348	3.904
6	6.555	5.072	6.780

Test 4 results:

Mode	NAFEMS Expected Value	QUAD8 Lumped Mass	QUAD8 Coupled Mass
1	0.421	0.4067	0.4190
2	1.029	0.9554	1.026
3	2.582	2.200	2.673
4	3.306	2.800	3.437
5	3.753	3.387	4.006
6	6.555	4.941	6.892

Conclusion

Considering the number of elements in the models, the program gave excellent results. With square elements, the results show excellent correlation, considering the mesh and the results for distorted elements are acceptable, considering the mesh and element distortions. Using the recommendation of 5 GRID points per half sine-wave of the deformed shape, Test 1 and 2 might be expected to predict mode 1 and possibly mode 2 well and Test 3 and 4 might be expected to predict the first mode well.

6.21 FV22 - Clamped Thick Rhombic Plate

Problem Description

Solution to find the first 6 modes of a clamped flat plate, which is skewed 45 degrees.

Solution Number

SOL 103 - normal modes

Features Used

- Lanczos Eigenvalue Solution

Reference

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., Selected Benchmarks for Natural Frequency Analysis, Test No. 22. Glasgow: NAFEMS, Nov., 1987.

Modeling Techniques Used

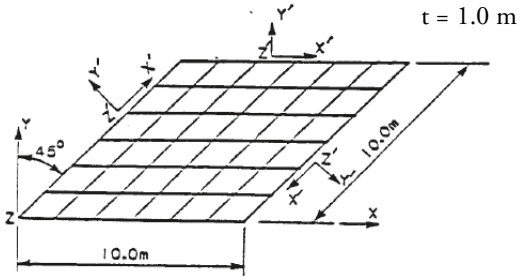
- Distorted Flat QUAD4 and QUAD8 elements modeling a thick clamped plate.

Illustrations of the Model

Geometry and Mesh

H.O.E. 6 × 6 (as shown)

L.O.E. 10 × 10



Boundary conditions

Clamped edges.

Model File Name and Location

Model File Name	Description	Location
`fv22_q4.dat`	QUAD4 with lumped mass	\verifman\nafems
`fv22_q4c.dat`	QUAD4 with coupled mass	\verifman\nafems
`fv22_q8.dat`	QUAD8 with lumped mass	\verifman\nafems
`fv22_q8c.dat`	QUAD8 with coupled mass	\verifman\nafems

Results

QUAD4

Mode	NAFEMS Theoretical Value	NAFEMS Expected Value	QUAD4 Lumped Mass	QUAD4 Coupled Mass
1	133.95	137.80	131.22	134.27
2	201.41	218.48	200.37	211.89
3	265.81	295.42	262.03	286.62
4	282.74	296.83	273.59	287.03
5	334.45	383.56	327.01	373.25
6	NA	426.59	372.22	410.58

QUAD8

Mode	NAFEMS Theoretical Value	NAFEMS Expected Value	QUAD4 Lumped Mass	QUAD4 Coupled Mass
1	133.95	133.86	134.91	135.24
2	201.41	203.34	204.42	206.27
3	265.81	271.38	270.27	276.39
4	282.74	283.68	286.89	289.07
5	334.45	346.41	374.45	353.77
6	NA	386.62	384.73	393.96

Conclusion

Distortion of elements to 45 degrees skew angle is well beyond the recommended maximum value of 30 degrees. However, the elements still performed well in this example. As the mesh is not that refined, the results are expected to be less accurate after the first mode or two.

6.22 FV32 - Cantilevered Tapered Membrane

Problem Description

Solution to find the first 6 modes of a tapered membrane plate.

Solution Number

SOL 103-normal modes

Features Used

- Lanczos eigenvalue solution

Reference

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., Selected Benchmarks for Natural Frequency Analysis, Test No. 32. Glasgow: NAFEMS, Nov., 1987.

Modeling Techniques Used

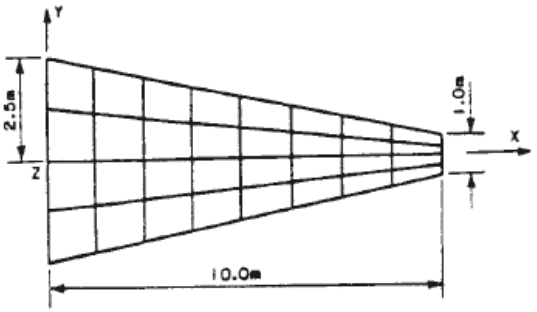
- Distorted Flat QUAD4 and QUAD8 elements modeling a tapered membrane. Note - the thickness is not provided, but for finite elements, any thickness will give the same answers.

Illustrations of the Model

Geometry and Mesh

H.O.E. 8 × 4 (as shown)

L.O.E. 16 × 8



Boundary Conditions

Clamped edges

Model File Name and Location

Model File Name	Description	Location
`fv32_q4.dat`	QUAD4 with lumped mass	\verifman\nafems
`fv32_q4c.dat`	QUAD4 with coupled mass	\verifman\nafems
`fv32_q8.dat`	QUAD8 with lumped mass	\verifman\nafems
`fv32_q8c.dat`	QUAD8 with coupled mass	\verifman\nafems

Results

QUAD4 results

Mode	NAFEMS Theoretical Value	NAFEMS Expected Value	QUAD4 Lumped Mass	QUAD4 Coupled Mass
1	44.623	44.636	44.52	44.65
2	130.03	130.14	129.55	131.04
3	162.70	162.72	162.56	162.80
4	246.05	246.63	244.13	250.33
5	379.90	382.02	374.46	391.54
6	391.44	391.55	389.60	393.10

QUAD8 results

Mode	NAFEMS Theoretical Value	NAFEMS Expected Value	QUAD4 Lumped Mass	QUAD4 Coupled Mass
1	44.623	44.905	44.54	44.63
2	130.03	132.12	129.71	130.11
3	162.70	162.83	162.66	162.71
4	246.05	252.99	245.14	246.44
5	379.90	393.31	377.87	381.41
6	391.44	396.26	390.92	391.52

Conclusion

Distortion of elements is a way to test their accuracy. In this case, the results are acceptably close to the predicted results.

6.23 FV41 - Axisymmetric Vibration

Problem Description

Free modes of a cylinder using axisymmetric elements.

Solution Number

SOL 103 - normal modes

Features Used

- Axisymmetric elements
- Normal modes - Lanczos method

Reference

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., Selected Benchmarks for Natural Frequency Analysis, Test No. 41. Glasgow: NAFEMS, Nov., 1987.

Modeling Techniques Used

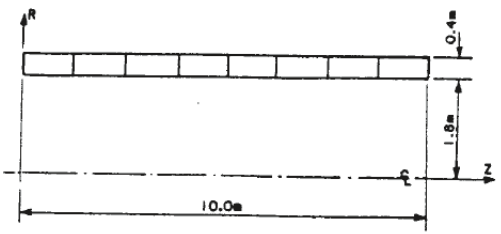
- Simply-supported beam model using BEAM elements.

Illustrations of the Model

Geometry and Mesh

H.O.E. 8 × 1 (as shown)

L.O.E. 16 × 3



Model File Name and Location

Model File Name	Description	Location
fv41_q4.dat	4-noded axisymmetric quad elements using lumped mass	\verifman\nafems
Fv41_q4c.dat	4-noded axisymmetric quad elements using coupled mass	\verifman\nafems
Fv41_q8.dat	8-noded axisymmetric quad elements using lumped mass	\verifman\nafems
Fv41_q4c.dat	8-noded axisymmetric quad elements using coupled mass	\verifman\nafems

Results

Natural frequencies (hz) of modes 2-6

Note: The first mode is a rigid-body mode.

4-noded CQUADX

Mode	NAFEMS Theoretical Value	NAFEMS Expected Value	4-noded QUADX Using Lumped Mass	4-noded QUADX Using Coupled Mass
2	243.53	244.01	243.23	244.01
3	377.41	379.41	374.47	379.42
4	394.11	395.41	389.47	395.44
5	397.72	397.94	389.49	401.38
6	405.28	421.87	396.10	421.89

8-noded CQUADX

Mode	NAFEMS Theoretical Value	NAFEMS Expected Value	8-noded QUADX Using Lumped Mass	8-noded QUADX Using Coupled Mass
2	243.53	243.50	243.24	243.57
3	377.41	377.46	356.49	377.46
4	394.11	394.28	356.88	394.30
5	397.72	397.94	375.85	397.97
6	405.28	406.41	393.65	406.44

Conclusion

While the results using lumped mass are not bad, the results using coupled mass fall within 1% of the expected values for all modes.

6.24 FV42 - Thick Hollow Sphere

Problem Description

Radial modes of a hollow sphere using axisymmetric elements.

Solution Number

SOL 103 - normal modes

Features Used

- Axisymmetric Elements
- Cylindrical Coordinate System
- MPC Equations
- Normal Modes - Lanczos Method

Reference

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., Selected Benchmarks for Natural Frequency Analysis, Test No. 42. Glasgow: NAFEMS, Nov., 1987.

Modeling Techniques Used

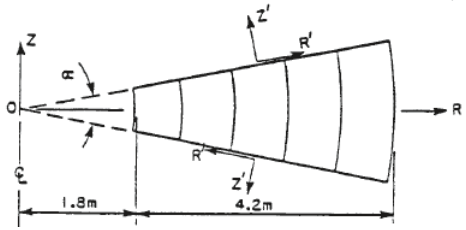
- Simply-supported beam model using BEAM elements.

Illustrations of the Model

Geometry and Mesh

H.O.E. 5 × 1 (as shown)
$\alpha = 10°$

L.O.E. 10 × 1
$\alpha = 5°$



Constraints:

z' displacements are constrained to 0.0 and GRIDs at the same radius are constrained to have the same radial displacement.

Main Index

Model File Name and Location

Model File Name	Description	Location
fv42_q4.dat	4-noded axisymmetric quad elements using lumped mass	\verifman\nafems
Fv42_q4c.dat	4-noded axisymmetric quad elements using coupled mass	\verifman\nafems
Fv42_q8.dat	8-noded axisymmetric quad elements using lumped mass	\verifman\nafems
Fv42_q8c.dat	8-noded axisymmetric quad elements using coupled mass	\verifman\nafems

Results

Natural frequencies (hz) of the first 5 modes

4-noded CQUADX

Mode	NAFEMS Theoretical Value	NAFEMS Expected Value	4-noded QUADX Using Lumped Mass	4-noded QUADX Using Coupled Mass
1	369.91	370.64	370.47	370.64
2	838.03	841.2	833.49	841.20
3	1451.2	1473.1	1423.77	1473.06
4	2117.0	2192.2	2033.83	2192.20
5	2795.8	2975.7	2050.78	2975.67

8-noded CQUADX

Mode	NAFEMS Theoretical Value	NAFEMS Expected Value	8-noded QUADX Using Lumped Mass	8-noded QUADX Using Coupled Mass
1	369.91	370.01	368.54	370.01
2	838.03	838.08	817.72	841.20
3	1451.2	1453.0	1387.84	1453.0
4	2117.0	2131.7	1475.40	1942.55
5	2795.8	2852.8	1974.77	2131.74

Conclusion

While the results using lumped mass are not bad, the results using coupled mass on the 4-noded elements matches the baseline values within a fraction of a percent. The 8-noded elements match the first 3 modes almost exactly, but fall off on the last two modes, indicating a need for a more refined mesh.

6.25 FV52 - Simply Supported Solid Square Plate

Problem Description

Calculate the first 10 modes of a plate which is supported in the Z-direction on its edges.

Solution Number

SOL 103 normal modes

Features Used

- Solid elements
- Normal modes - Lanczos method

Reference

1. NAFEMS Finite Element Methods & Standards. Abbassian, F., Dawswell, D. J., and Knowles, N. C.Selected Benchmarks for Natural Frequency Analysis, Test No. 51. Glasgow: NAFEMS, Nov., 1987.

Modeling Techniques Used

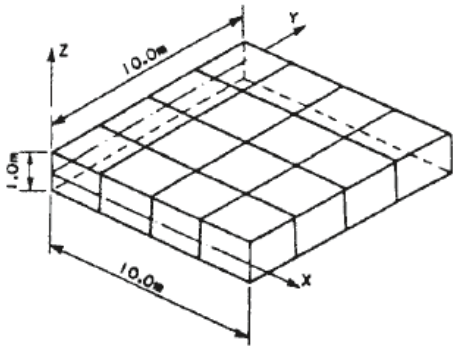
- Simply-supported beam model using BEAM elements.

Illustrations of the Model

Geometry and Mesh

H.O.E. 4 × 4 × 1 (as shown)

L.O.E. 8 × 8 × 3



Constraints

z' displacements are constrained to 0.0 along the bottom edges

Model File Name and Location

Model File Name	Description	Location
`fv52_h8.dat`	8-noded HEXA elements using lumped mass	\verifman\nafems
`fv52_h8c.dat`	8-noded HEXA elements using coupled mass	\verifman\nafems
`fv52_q20.dat`	20-noded HEXA elements using lumped mass	\verifman\nafems
`fv52_q20c.dat`	20-noded HEXA elements using coupled mass	\verifman\nafems

Results

Natural frequencies (Hz) of modes 4-10. Modes 1-3 are rigid-body modes.

8-noded HEXA

Mode	NAFEMS Theoretical Value	NAFEMS Expected Value	8-noded HEXA using Lumped Mass	8-noded HEXA using Coupled Mass
4	45.897	51.564	44.62	45.24
5, 6	109.44	132.73	107.83	113.70
7	167.89	194.37	157.55	172.25
8	193.59	197.18	193.64	196.76
9, 10	206.19	210.55	200.39	209.56

20 noded HEXA

Mode	NAFEMS Theoretical Value	NAFEMS Expected Value	20-noded HEXA using Lumped Mass	20-noded HEXA using Coupled Mass
4	45.897	44.762	43.81	44.16
5, 6	109.44	110.52	105.24	107.93
7	167.89	169.08	156.26	163.92
8	193.59	193.93	193.97	193.91
9, 10	206.19	206.64	193.52	206.60

Conclusion

In most cases, the values from MSC Nastran were closer to the theoretical values than the NAFEMS expected values. As expected with a coarse mesh model, the results using coupled mass were closer to the theoretical values than those using lumped mass.

6.26 FV73 - Cantilevered Thin Square Plate

Problem Description

Calculate the first 6 modes of a plate which is simply supported along the Y-axis.

Solution Number

SOL 103 normal modes

Features Used

- 8-noded QUAD elements (NAFEMS lists 8-noded semi-loof elements, which do not exist in MSC Nastran)
- Normal modes - Lanczos method

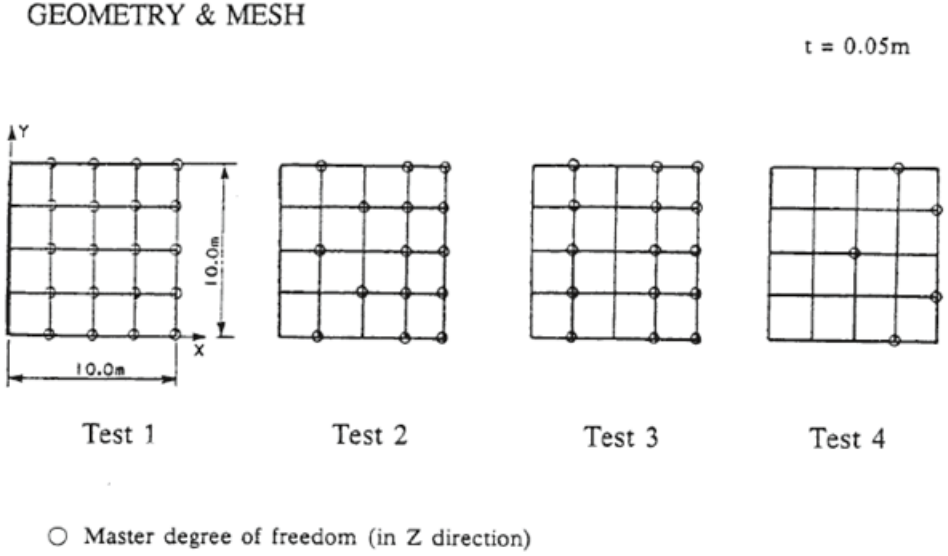
Reference

1. NAFEMS Finite Element Methods & Standards. Abbassian, F., Dawswell, D. J., and Knowles, N. C.Selected Benchmarks for Natural Frequency Analysis, Test No. 73. Glasgow: NAFEMS, Nov., 1987.

Modeling Techniques Used

- Simply-supported plate model using QUAD8 elements.

Illustrations of the Model



Constraints

X, Y, and Z translation and Y rotation are constrained to 0.0 on the GRID points along the Y-axis

Model File Name and Location

Model File Name	Description	Location
`fv73_q8_t1.dat`	Test 1 using Lumped Mass	\verifman\nafems
`fv73_q8_t1c.dat`	Test 1 using Coupled Mass	\verifman\nafems
`fv73_q8_t2.dat`	Test 2 using Lumped Mass	\verifman\nafems
`fv73_q8_t2c.dat`	Test 2 using Coupled Mass	\verifman\nafems
`fv73_q8_t3.dat`	Test 3 using Lumped Mass	\verifman\nafems
`fv73_q8_t3c.dat`	Test 3 using Coupled Mass	\verifman\nafems
`fv73_q8_t4.dat`	Test 4 using Lumped Mass	\verifman\nafems
`fv73_q8_t4c.dat`	Test 4 using Coupled Mass	\verifman\nafems

Results

Natural frequencies (Hz) of modes 1-6.

NAFEMS Values

	Mode 1	Mode 2	Mode 3	Mode 4	Mode 5	Mode 6
Reference Solution	0.421	0.129	2.582	3.306	3.753	6.555
Test 1	0.4174	1.020	2.564	3.302	3.769	6.805
Test 2	0.4174	1.020	2.597	3.345	3.888	7.517
Test 3	0.4175	1.021	2.677	3.365	4.035	7.495
Test 4	0.4184	1.032	2.850	3.571	5.466	-

Lumped Mass

	Mode 1	Mode 2	Mode 3	Mode 4	Mode 5	Mode 6
Test 1	0.415	1.005	2.485	3.150	3.622	6.292
Test 2	0.415	1.006	2.509	3.180	3.713	6.901
Test 3	0.415	1.007	2.563	3.196	3.828	6.879
Test 4	0.416	1.015	2.711	3.272	4.935	-

Coupled Mass

	Mode 1	Mode 2	Mode 3	Mode 4	Mode 5	Mode 6
Test 1	0.418	1.023	2.579	3.298	3.765	6.719
Test 2	0.418	1.023	2.605	3.332	3.862	7.399
Test 3	0.418	1.024	2.672	3.344	4.000	7.387
Test 4	0.419	1.034	2.836	3.535	5.360	-

Conclusion

While the results using lumped mass are good, the results using coupled mass is closer to the reference solution than the NAFEMS expected values in all cases.

Forced Vibration

6.27 Test 5 - Deep Simply Supported Beam

Problem Description

Normal modes of a simply supported beam model.

Solution Number

SOL 103 - normal modes using Lanczos method

Features Used

- Normal modes calculation using 3-noded BEAM3 elements

Reference

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C.Selected Benchmarks for Natural Frequency Analysis, Test No. 5. Glasgow: NAFEMS, Nov., 1987.

Modeling Techniques Used

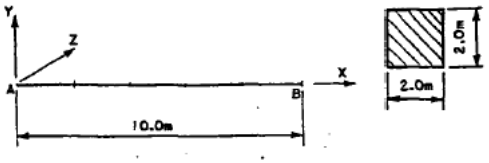
A simply supported beam model is solved for the first 9 modes.

Illustrations of the Model

Geometry and Mesh

Exact beam: 5 elements

Iso-parametric: 5 elements



Model File Name and Location

Model Name	Description	Location
Test5.dat	Simply supported Beam model - lumped mass	\verifman\nafems
Test5c.dat	Simply supported Beam model - coupled mass	\verifman\nafems

Results

Mode	NAFEMS Theoretical Value	NAFEMS Iso Value	BEAM3 Results
1 and 2	42.650	42.657	43.185
3	71.20	71.202	71.209
4	125.00	125.00	125.002
5 and 6	148.15	148.71	154.424
7	213.61	213.72	213.848
8 and 9	283.47	286.91	307.523

Conclusion

The normal recommendation is to have 8 elements per sine wave of the deformed shape. With only 5 elements in the model, the first mode is the only mode which can be expected to be accurate. Modes 1 through 4 are the first bending, axial, and torsional modes and are predicted well using this model. The other modes are not predicted as well, but mesh refinement would improve the accuracy of these modes.

6.28 Test 5H - Deep Simply Supported Beam Harmonic Forced Response

Problem Description

Forced harmonic vibration response of a simply supported beam model.

Solution Number

SOL 108 and 111 - Direct and Modal Frequency response analysis

Features Used

- Frequency response calculation using 3-noded BEAM3 elements

Reference

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C.Selected Benchmarks for Natural Frequency Analysis, Test No. 5H. Glasgow: NAFEMS, Nov., 1987.

Modeling Techniques Used

A simply supported beam model is solved for a harmonic loading. In the direct method, Rayleigh damping is used with a_0=5.35 and a_1=7.46 $\times 10^{-5}$. In the modal approach, modal damping of 2% of critical is used on the first 16 modes.

A uniform load of 10^6N/m is applied in the Y-direction over the frequency range from 0.0 to 60.0Hz.

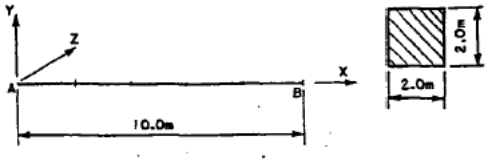
As the NAFEMS baseline does not use residual vectors, they are turned off in the modal solution. In a normal run, they would be enabled to improve the accuracy of the solution.

Illustrations of the Model

Geometry and Mesh

Exact beam: 5 elements

Iso-parametric: 5 elements



Model File Name and Location

Model Name	Description	Location
`Test5hd.dat`	Simply supported Beam model - Forced Harmonic Vibration, direct solution.	\verifman\nafems
`Test5hm.dat`	Simply supported Beam model - Forced Harmonic Vibration, modal solution.	\verifman\nafems

Results

Mode	Peak Disp (mm)	Peak Stress (N/mm^2)	Frequency (Hz)
Reference Solution	13.45	241.9	42.65
Target Solution (modal)	13.44	240.3	42.60
Target Solution (direct)	13.39	242.0	42.58
Modal Solution	13.14	238.4	43.185
Direct Solution	13.27	240.9	43.185

Conclusion

The maximum response occurs at the first natural frequency of the system, which for the 3-noded beam model is 43.185hz. The peak values are within engineering accuracy of the predicted values.

6.29 Test 5P - Deep Simply Supported Beam Periodic Forced Vibration Response

Problem Description

Forced periodic vibration response of a simply supported beam model.

Solution Number

SOL 108 and 111 - Direct and Modal Frequency response analysis

Features Used

- Frequency response calculation using 3-noded BEAM3 elements

Reference

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C.Selected Benchmarks for Natural Frequency Analysis, Test No. 5P. Glasgow: NAFEMS, Nov., 1987.

Modeling Techniques Used

A simply supported beam model is solved for a periodic loading. In the direct method, Rayleigh damping is used with a_0=5.35 and a_1=7.46 $\times 10^{-5}$. In the modal approach, modal damping of 2% of critical is used on the first 16 modes.

A uniform load of 10^6 N/m is applied in the Y-direction using the formula:

$F=F_0(\sin\omega t - \sin 3\omega t)$

where,

F_0 = 10^6 N/m

ω = $2\pi f$

f = 20 Hz

Although this appears to be a transient solution, it is really a frequency response solution, using Fourier transforms and inverse Fourier transforms.

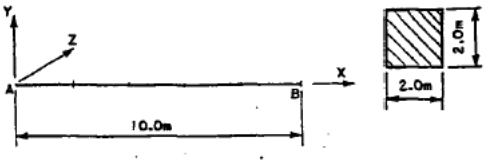
Illustrations of the Model

Illustration of the model from the reference:

Geometry and Mesh

Exact beam: 5 elements

Iso-parametric: 5 elements

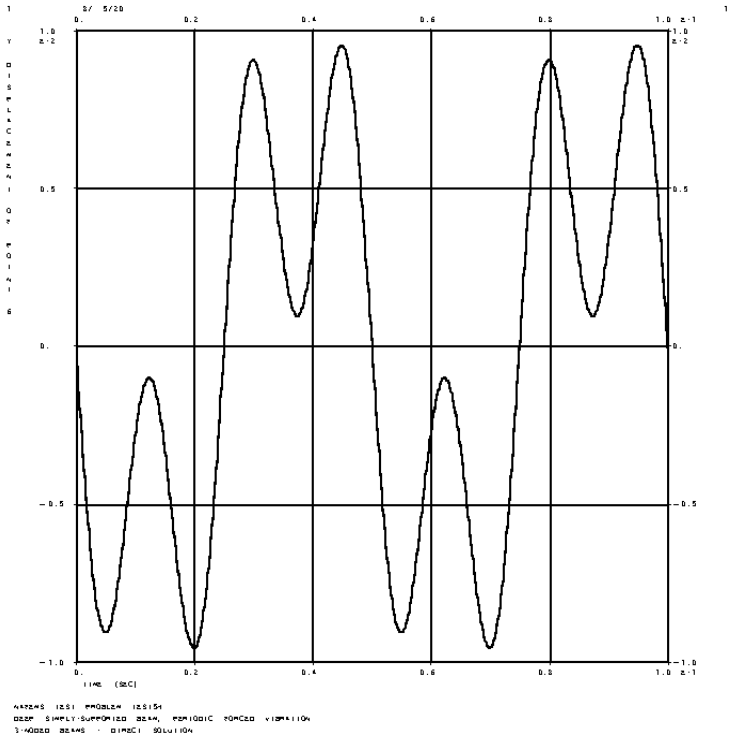


Model File Name and Location

Model Name	Description	Location
`Test5pd.dat`	Simply supported Beam model - Forced Periodic Vibration, direct solution.	\verifman\nafems
`Test5pm.dat`	Simply supported Beam model - Forced Periodic Vibration, modal solution.	\verifman\nafems

Results

Mode	Peak Disp (mm)	Peak Stress (N/mm^2)
Reference Solution	0.951	17.10
Target Solution (modal)	0.949	17.05
Target Solution (direct)	0.953	17.33
Modal Solution	0.953	17.38
Direct Solution	0.953	17.38



Conclusion

As residual vectors were used, the modal and direct methods gave the same results (to the number of significant digits shown), which were within engineering accuracy of the reference solution.

6.30 Test 5T-Deep Simply Supported Beam Transient Forced Vibration Response

Problem Description

Forced transient vibration response of a simply supported beam model.

Solution Number

SOL 109 and 112 - Direct and Modal transient response analysis

Features Used

- Transient response calculation using 3-noded BEAM3 elements

Reference

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C.Selected Benchmarks for Natural Frequency Analysis, Test No. 5T. Glasgow: NAFEMS, Nov., 1987.

Modeling Techniques Used

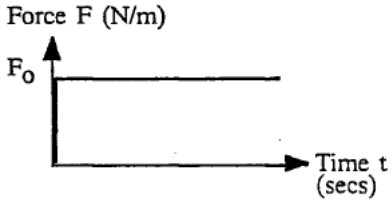
A simply supported beam model is solved for a periodic loading. In the direct method, Rayleigh damping is used with a_0=5.35 and a_1=7.46x10^{-5}. In the modal approach, modal damping of 2% of critical is used on the first 16 modes.

A load of 10^6N/m is suddenly applied in the Y-direction:

Forcing Function (Temporal)

Suddenly applied step load

$F_0 = 10^6$ N/m

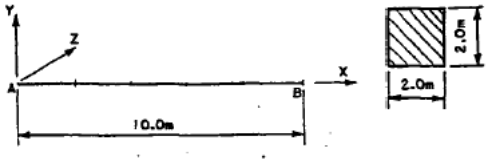


Illustrations of the Model

Geometry and Mesh

Exact beam: 5 elements

Iso-parametric: 5 elements



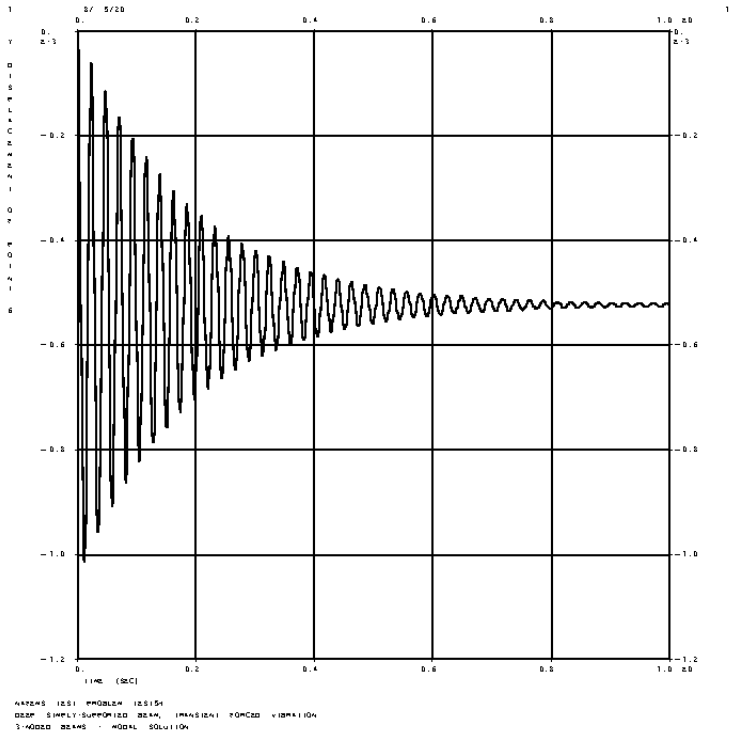
Model File Name and Location

Model Name	Description	Location
`Test5td.dat`	Simply supported Beam model - Forced Transient Vibration, direct solution	\verifman\nafems
`Test5tm.dat`	Simply supported Beam model - Forced Transient Vibration, modal solution	\verifman\nafems

Results

Solution	Peak Disp PSD (mm)	T_p (sec)	Peak Stress (N/mm2)	Static displacement (mm)
Reference Solution	1.043	0.0117	18.76	0.538
Target Solution (modal)	1.040	0.0116	18.71	0.536
Target Solution (direct)	1.057	0.0117	18.77	0.536
Modal Solution	1.016	0.0117	18.15	0.525
Direct Solution	1.016	0.0117	18.15	0.525

Plot of displacement vs time for the mid-point of the beam



Conclusion

As residual vectors were used, the modal and direct methods gave the same results (to the number of significant digits shown), which were within engineering accuracy of the reference solution.

6.31 Test 5R-Deep Simply Supported Beam Random Forced Response

Problem Description

Forced random vibration response of a simply supported beam model.

Solution Number

SOL 108 and 111 - Direct and Modal Frequency response analysis

Features Used

- Frequency response calculation using BEAM elements
- Random response analysis

Reference

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C.Selected Benchmarks for Natural Frequency Analysis, Test No. 5R. Glasgow: NAFEMS, Nov., 1987.

Modeling Techniques Used

A simply supported beam model is solved for a harmonic loading, followed by random response analysis. In the direct method, Rayleigh damping is used with a_0=5.35 and a_1=7.46x10^{-5}. In the modal approach, modal damping of 2% of critical is used on the first 16 modes.

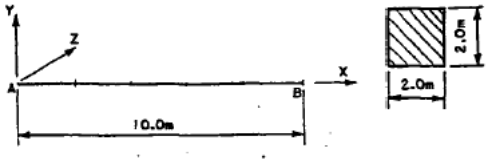
A uniform load of 10^6 N/m is applied in the Y-direction over the frequency range from 0.0 to 60.0 Hz to calculate the transfer functions, then the PSD provided is applied to obtain the random response.

Illustrations of the Model

Geometry and Mesh

Exact beam: 5 elements

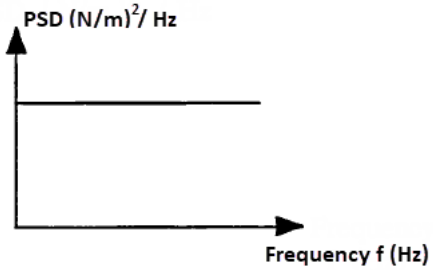
Iso-parametric: 5 elements



Forcing Function (Temporal)

Random forcing with uniform power spectral density (of force)

PSD = $(10^6$ N/m$)^2$ / Hz

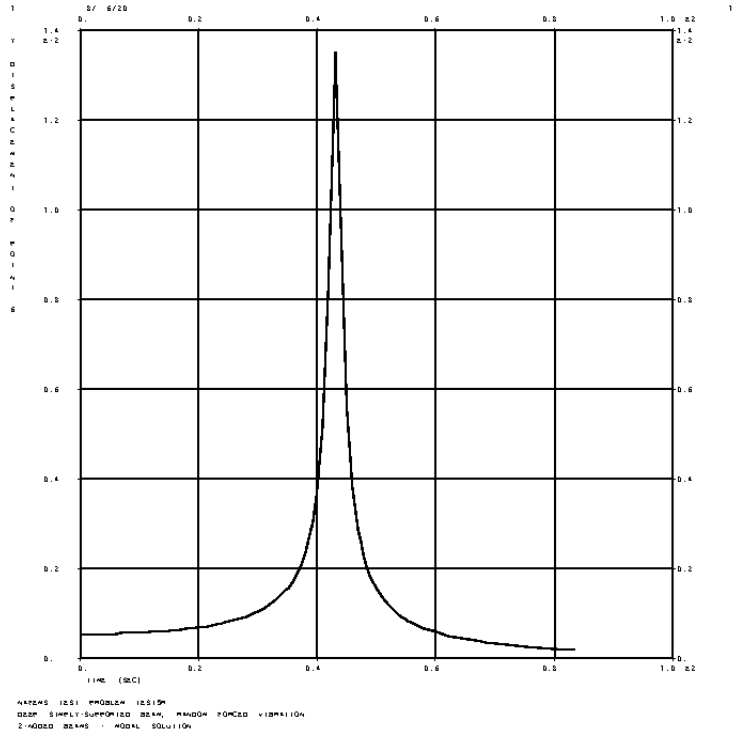


Model File Name and Location

Model Name	Description	Location
`Test5rd.dat`	Simply supported Beam model - Forced Random Vibration, direct solution	\verifman\nafems
`Test5rm.dat`	Simply supported Beam model - Forced Random Vibration, modal solution	\verifman\nafems

Results

Displacement PSD from Modal Run



Solution	Peak Disp PSD (mm^2/hz)	Peak Stress PSD $(N/mm^2)^2$/hz	Frequency (Hz)
Reference Solution	180.90	58516	42.65
Target Solution (modal)	179.35	53722	42.58

Solution	Peak Disp PSD (mm^2/hz)	Peak Stress PSD ($(N/mm^2)^2$/hz	Frequency (Hz)
Target Solution (direct)	179.29		42.53
Modal Solution	182.6	59388	43.166
Direct Solution	186.4	60623	43.166

Conclusion

The maximum response occurs at the first natural frequency of the system, which for this model is 43.166hz. The peak values are within engineering accuracy of the predicted values.

6.32 Test 13: Simply Supported Thin Square Plate: Frequency Extraction

Problem Description

Modal analysis of a thin square plate for extraction of frequencies.

Solution Number

SOL 103

Features Used

- Modal analysis using QUAD4 and QUAD8 shell elements

Reference

1. Test 13 from NAFEMS Selected Benchmarks for Forced Vibration, R0016, March 1993.

Modeling Techniques Used

A simply supported thin plate is solved for a modal. The model details are given below:

Material Properties

$E = 200 \times 10^9$ N/m^2 $\nu = 0.3$ $\rho = 8000$ kg/m^3

Loading

Steady state harmonic forcing function $F = F_0 \sin\omega t$

$F_0 = 100$ N/m^2 over whole plate with $\omega = 2\pi f$ and $f = 4.16$ Hz.

Boundary Conditions

$U_x = U_y = R_z = 0.0$ at all nodes and $U_z = 0$ along all 4 edges

$R_x = 0.0$ along edges x=0 and x=10m and $R_y = 0.0$ along edges y=0 and y=10m.

Illustrations of the Model

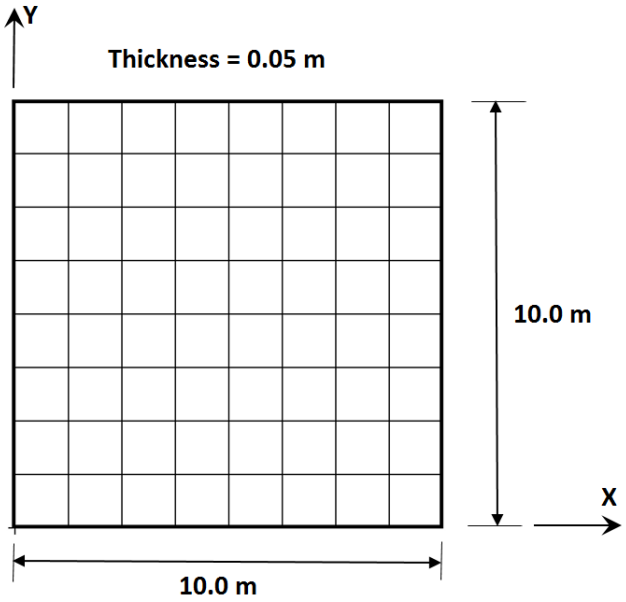


Figure 6.32-1 Simply Supported Thin Square Plate

Model File Name and Location

Model Name	Description	Location
test13_q4.dat	Test13: QUAD4 modal analysis	\verifman\nafems
test13_q8.dat	Test13: QUAD8 modal analysis	\verifman\nafems

Results

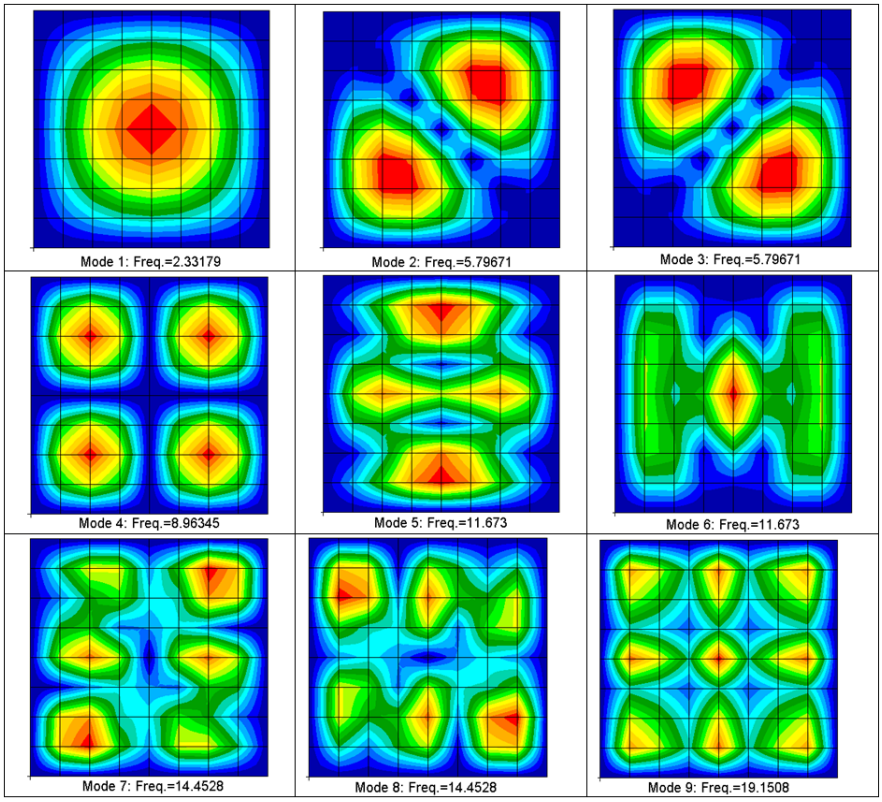


Figure 6.32-2 Mode Shapes of Thin Square Plate

Mode	NAFEMS	MSC Nastran (QUAD4)		MSC Nastran (QUAD8)	
	Hz	Hz	% Error	Hz	% Error
1	2.377	2.33179	-1.90%	2.375694	-0.05%
2 and 3	5.942	5.79671	-2.45%	5.938033	-0.07%
4	9.507	8.96345	-5.72%	9.747177	2.53%
5 and 6	11.884	11.67295	-1.78%	11.87156	-0.10%
7 and 8	15.449	14.45277	-6.45%	16.56407	7.22%

Conclusion

The results from MSC Nastran are compared with the NAFEMS results in the above tables.

6.33 Test 13H: Simply Supported Thin Square Plate: Harmonic Forced Vibration

Problem Description

Forced harmonic vibration response of a thin square plate with modal and direct solutions.

Solution Number

SOL 108 and SOL 111

Features Used

- Harmonic response calculation using QUAD4 and QUAD8 shell elements

Reference

1. Test 13H from NAFEMS Selected Benchmarks for Forced Vibration, R0016, March 1993.

Modeling Techniques Used

A simply supported thin plate is solved for a harmonic loading. In the direct method, Rayleigh damping is used with $a_0 = 0.299$ and $a_1 = 1.339 \times 10^{-3}$. In the modal approach, modal damping of 2% of critical is used on the first 16 modes. The model details are given below:

Material Properties

$E = 200 \times 10^9$ N/m^2 $\nu = 0.3$ $\rho = 8000$ kg/m^3

Loading

Steady state harmonic forcing function $F = F_0 \sin\omega t$

$F_0 = 100$ N/m^2 over whole plate with $\omega = 2\pi f$ and $f = 4.16$ Hz.

Boundary Conditions

$U_x = U_y = R_z = 0.0$ at all nodes and $U_z = 0$ along all 4 edges

$R_x = 0.0$ along edges x=0 and x=10m and $R_y = 0.0$ along edges y=0 and y=10m.

Main Index

Illustrations of the Model

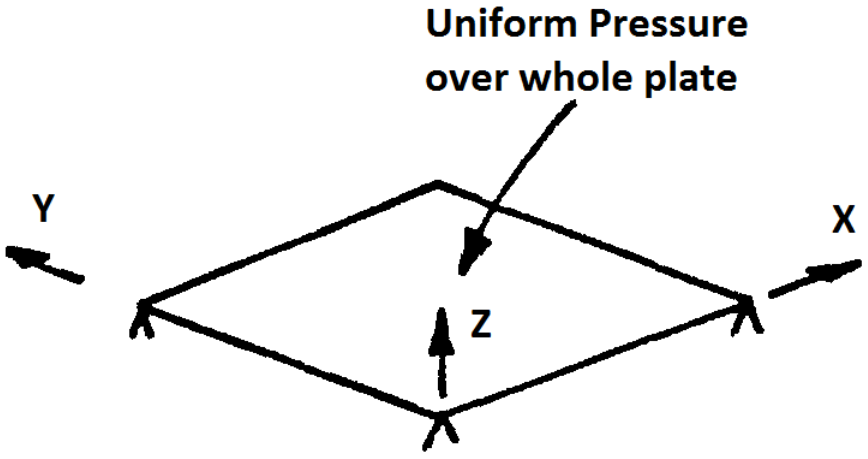


Figure 6.33-1 Simply Supported Thin Square Plate

Model File Name and Location

Model Name	Description	Location
test13hd_q4.dat	Test13H: QUAD4 direct solution	\verifman\nafems
test13hd_q8.dat	Test13H: QUAD8 direct solution	\verifman\nafems
test13hm_q4.dat	Test13H: QUAD4 modal solution	\verifman\nafems
test13hm_q8.dat	Test13H: QUAD8 modal solution	\verifman\nafems

Results

	Peak Displacement δ (mm)		
	NAFEMS	MSC Nastran	
		Values	% Error
QUAD4 (Modal)	45.42	45.9934	1.26%
QUAD4 (Direct)	45.42	45.9632	1.20%
QUAD8 (Modal)	45.42	45.2967	-0.27%
QUAD8 (Direct)	45.42	45.2756	-0.32%

	Peak Stress σ (N/mm^2)		
	NAFEMS	MSC Nastran	
		Values	% Error
QUAD4 (Modal)	30.03	31.0656	3.45%
QUAD4 (Direct)	30.03	31.0442	3.38%
QUAD8 (Modal)	30.03	35.0256	16.64%
QUAD8 (Direct)	30.03	35.0075	16.58%

	Frequency f (Hz)		
	NAFEMS	MSC Nastran	
		Values	% Error
QUAD4 (Modal)	2.377	2.3318	-1.90%

	Frequency f (Hz)		
	NAFEMS	MSC Nastran	
		Values	% Error
QUAD4 (Direct)	2.377	2.3318	-1.90%
QUAD8 (Modal)	2.377	2.3757	-0.05%
QUAD8 (Direct)	2.377	2.3757	-0.05%

Conclusion

The results from MSC Nastran are compared with the NAFEMS results in the above tables.

6.34 Test 13P: Simply Supported Thin Square Plate: Periodic Forced Vibration

Problem Description

Forced periodic vibration response of a thin square plate with modal and direct solutions.

Solution Number

SOL 108 and SOL 111

Features Used

- Periodic response calculation using QUAD4 and QUAD8 shell elements

Reference

1. Test 13P from NAFEMS Selected Benchmarks for Forced Vibration, R0016, March 1993.

Modeling Techniques Used

A simply supported thin plate is solved for a periodic loading. In the direct method, Rayleigh damping is used with $a_0 = 0.299$ and $a_1 = 1.339\times10^{-3}$. In the modal approach, modal damping of 2% of critical is used on the first 16 modes. The model details are given below:

Material Properties

$E = 200 \times 10^9$ N/m^2 $\quad \nu = 0.3$ $\quad \rho = 8000$ kg/m^3

Loading

Steady state periodic forcing function $F = F_0 (\sin\omega t - \sin 3\omega t)$

$F_0 = 100$ N/m^2 over whole plate with $\omega = 2\pi f$ and $f = 1.2$ Hz.

Boundary Conditions

$U_x = U_y = R_z = 0.0$ at all nodes and $U_z = 0$ along all 4 edges

$R_x = 0.0$ along edges x=0 and x=10m and $R_y = 0.0$ along edges y=0 and y=10m.

Main Index

Illustrations of the Model

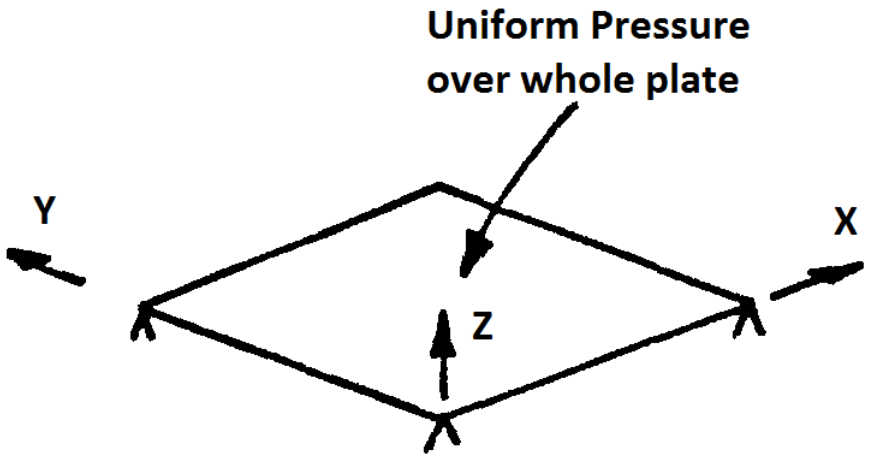


Figure 6.34-1 Simply Supported Thin Square Plate

Model File Name and Location

Model Name	Description	Location
`test13pd_q4.dat`	Test13P: QUAD4 direct solution	\verifman\nafems
`test13pd_q8.dat`	Test13P: QUAD8 direct solution	\verifman\nafems
`test13pm_q4.dat`	Test13P: QUAD4 modal solution	\verifman\nafems
`test13pm_q8.dat`	Test13P: QUAD8 modal solution	\verifman\nafems

Results

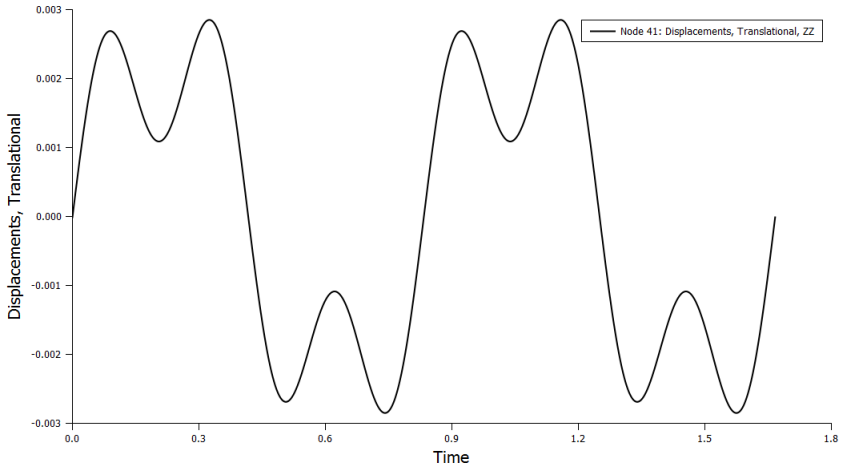


Figure 6.34-2 Displacement versus Time plot at node 41

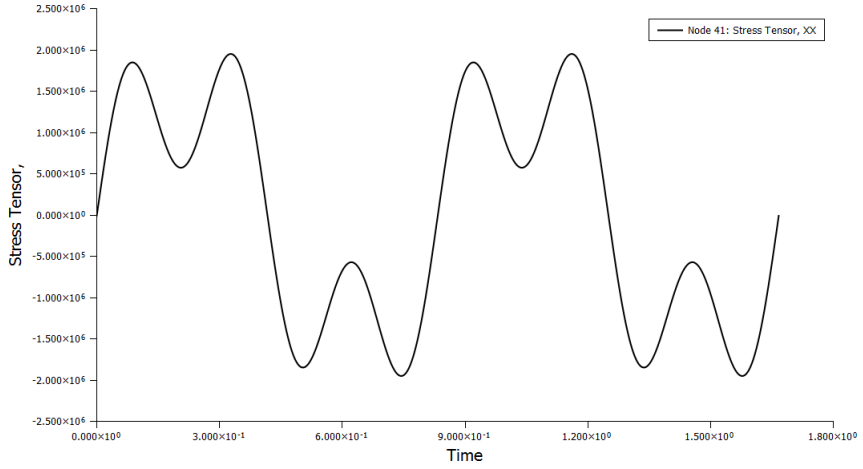


Figure 6.34-3 Stress versus Time plot at node 41

	Peak Displacement δ (mm)			Peak Stress σ (N/mm^2)		
	NAFEMS	MSC Nastran		NAFEMS	MSC Nastran	
		Values	% Error		Values	% Error
QUAD4 (Modal)	2.863	2.8519	-0.39%	2.018	1.9526	-3.24%
QUAD4 (Direct)	2.863	2.8513	-0.41%	2.018	1.9510	-3.32%
QUAD8 (Modal)	2.863	2.8682	0.18%	2.018	2.2714	12.56%
QUAD8 (Direct)	2.863	2.8675	0.16%	2.018	2.2690	12.44%

Conclusion

The displacement and stress results from the direct solution with QUAD4 elements are shown in Figure 6.34-2 and Figure 6.34-3. The results from MSC Nastran are compared with the NAFEMS results in the above table.

6.35 Test 13T: Simply Supported Thin Square Plate: Transient Forced Vibration

Problem Description

Forced transient vibration response of a thin square plate with modal and direct solutions.

Solution Number

SOL 109 and SOL 112

Features Used

- Transient response calculation using QUAD4 and QUAD8 shell elements.

Reference

1. Test 13T from NAFEMS Selected Benchmarks for Forced Vibration, R0016, March 1993.

Modeling Techniques Used

A simply supported thin plate is solved for a transient loading. In the direct method, Rayleigh damping is used with $a_0 = 0.299$ and $a_1 = 1.339 \times 10^{-3}$. In the modal approach, modal damping of 2% of critical is used on the first 16 modes. The model details are given below:

Material Properties

$E = 200 \times 10^9$ N/m^2 $\nu = 0.3$ $\rho = 8000$ kg/m^3

Loading

Suddenly applied step load $F_0 = 100$ N/m^2 over whole plate.

Boundary Conditions

$U_x = U_y = R_z = 0.0$ at all nodes and $U_z = 0$ along all 4 edges

$R_x = 0.0$ along edges x=0 and x=10m and $R_y = 0.0$ along edges y=0 and y=10m.

Illustrations of the Model

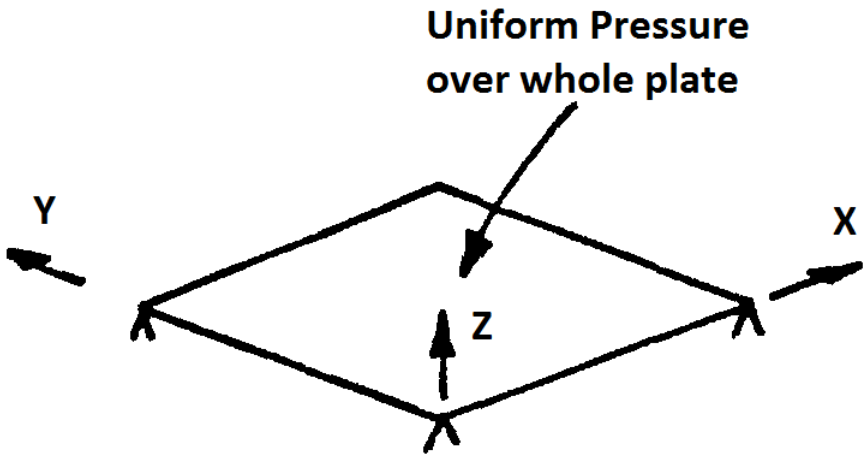


Figure 6.35-1 Simply Supported Thin Square Plate

Model File Name and Location

Model Name	Description	Location
`test13td_q4.dat`	Test13T: QUAD4 direct solution	\verifman\nafems
`test13td_q8.dat`	Test13T: QUAD8 direct solution	\verifman\nafems
`test13tm_q4.dat`	Test13T: QUAD4 modal solution	\verifman\nafems
`test13tm_q8.dat`	Test13T: QUAD8 modal solution	\verifman\nafems

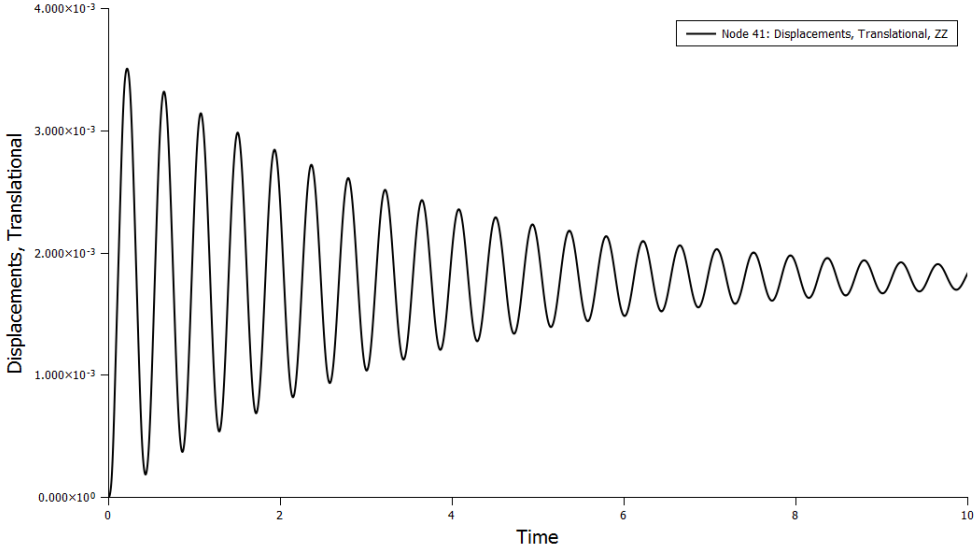


Figure 6.35-2 Displacement versus Time plot at node 41

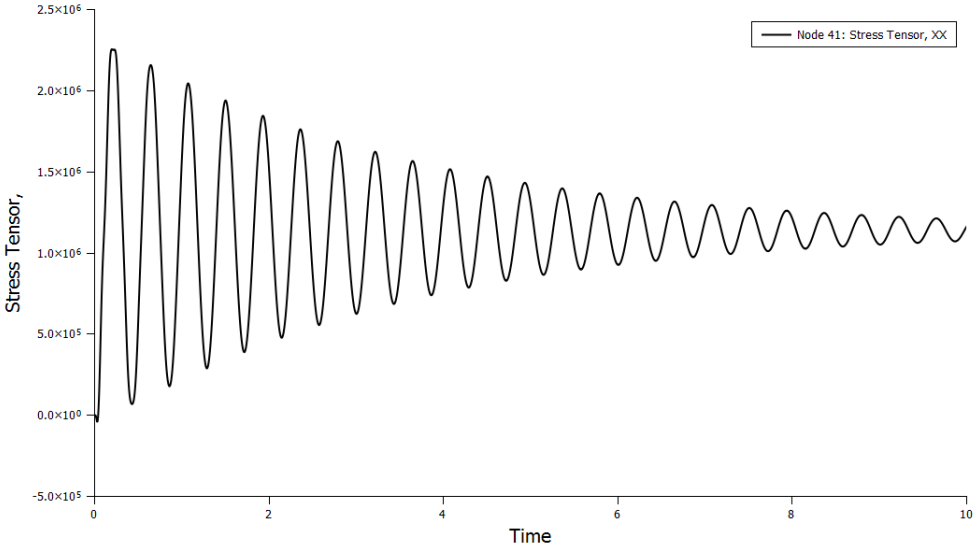


Figure 6.35-3 Stress versus Time plot at node 41

Results

	Peak Displacement δ_p (mm)			Peak Displacement t_p (sec)		
	NAFEMS	MSC Nastran		NAFEMS	MSC Nastran	
		Values	% Error		Values	% Error
QUAD4 (Modal)	3.523	3.4951	-0.79%	0.21	0.2150	2.38%
QUAD4 (Direct)	3.523	3.5082	-0.42%	0.21	0.2150	2.38%
QUAD8 (Modal)	3.523	3.4329	-2.56%	0.21	0.2110	0.48%
QUAD8 (Direct)	3.523	3.4479	-2.13%	0.21	0.2110	0.48%

	Peak Stress σ (N/mm^2)			Static Displacement δ_s (mm)		
	NAFEMS	MSC Nastran		NAFEMS	MSC Nastran	
		Values	% Error		Values	% Error
QUAD4 (Modal)	2.484	2.2601	-9.01%	1.817	1.8038	-0.73%
QUAD4 (Direct)	2.484	2.2534	-9.28%	1.817	1.8038	-0.73%
QUAD8 (Modal)	2.484	2.4977	0.55%	1.817	1.7647	-2.88%
QUAD8 (Direct)	2.484	2.4727	-0.45%	1.817	1.7647	-2.88%

Conclusion

The displacement and stress results from the direct solution with QUAD4 elements are shown in Figure 6.35-2 and Figure 6.35-3. The results from MSC Nastran are compared with the NAFEMS results in the above tables.

6.36 Test 13R: Simply Supported Thin Square Plate: Random Forced Vibration

Problem Description

Forced random vibration response of a thin square plate with modal and direct solutions.

Solution Number

SOL 108 and SOL 111

Features Used

- Random response calculation using QUAD4 and QUAD8 shell elements

Reference

1. Test 13R from NAFEMS Selected Benchmarks for Forced Vibration, R0016, March 1993.

Modeling Techniques Used

A simply supported thin plate is solved for a random loading. In the direct method, Rayleigh damping is used with $a_0 = 0.299$ and $a_1 = 1.339 \times 10^{-3}$. In the modal approach, modal damping of 2% of critical is used on the first 16 modes. The model details are given below:

Material Properties

$E = 200 \times 10^9$ N/m^2 $\nu = 0.3$ $\rho = 8000$ kg/m^3

Loading

Random forcing with uniform power spectral density (of force) PSD = (100 N/m^2)^2 / Hz over whole plate.

Boundary Conditions

$U_x = U_y = R_z = 0.0$ at all nodes and $U_z = 0$ along all 4 edges

$R_x = 0.0$ along edges x=0 and x=10m and $R_y = 0.0$ along edges y=0 and y=10m.

Illustrations of the Model

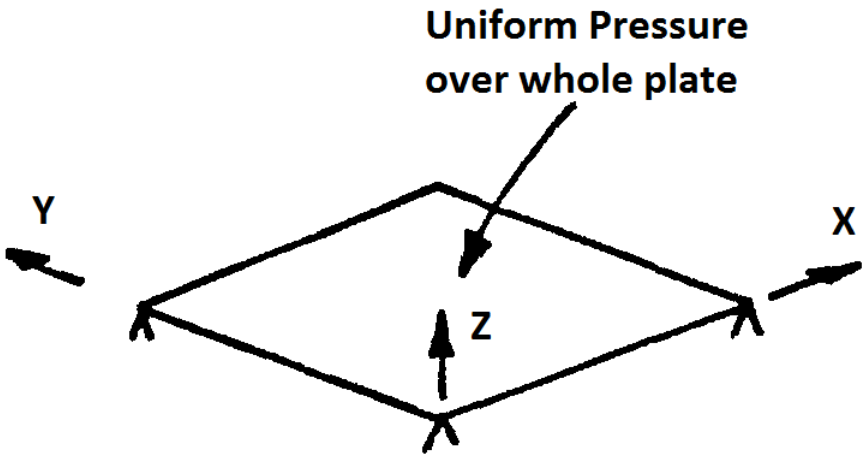


Figure 6.36-1 Simply Supported Thin Square Plate

Model File Name and Location

Model Name	Description	Location
test13rd_q4.dat	Test13R: QUAD4 direct solution	\verifman\nafems
test13rd_q8.dat	Test13R: QUAD8 direct solution	\verifman\nafems
test13rm_q4.dat	Test13R: QUAD4 modal solution	\verifman\nafems
test13rm_q8.dat	Test13R: QUAD8 modal solution	\verifman\nafems

Results

	Peak Displacement δ PSD (mm^2/Hz)		
	NAFEMS	MSC Nastran	
		Values	% Error
QUAD4 (Modal)	2063.2	2115.3920	2.53%
QUAD4 (Direct)	2063.2	2112.6130	2.39%
QUAD8 (Modal)	2063.2	2051.7860	-0.55%
QUAD8 (Direct)	2063.2	2049.8790	-0.65%

	Peak Stress σ PSD ($((N/mm^2)^2/Hz$)		
	NAFEMS	MSC Nastran	
		Values	% Error
QUAD4 (Modal)	1025.44	913.7688	-10.89%
QUAD4 (Direct)	1025.44	912.5110	-11.01%
QUAD8 (Modal)	1025.44	1226.8230	19.64%
QUAD8 (Direct)	1025.44	1225.5580	19.52%

	Frequency f (Hz)		
	NAFEMS	MSC Nastran	
		Values	% Error
QUAD4 (Modal)	2.377	2.3318	-1.90%
QUAD4 (Direct)	2.377	2.3318	-1.90%
QUAD8 (Modal)	2.377	2.3757	-0.05%
QUAD8 (Direct)	2.377	2.3757	-0.05%

Conclusion

The results from MSC Nastran are compared with the NAFEMS results in the above tables.

6.37 Test 21: Simply Supported Thick Square Plate: Frequency Extraction

Problem Description

Modal analysis of a thick square plate for extraction of frequencies.

Solution Number

SOL 103

Features Used

- Modal analysis using QUAD4 and QUAD8 shell elements.

Reference

1. Test 21 from NAFEMS Selected Benchmarks for Forced Vibration, R0016, March 1993.

Modeling Techniques Used

A simply supported thick plate is solved for a modal. The model details are given below:

Material Properties

$E = 200 \times 10^9$ N/m^2 $\nu = 0.3$ $\rho = 8000$ kg/m^3

Boundary Conditions

$U_x = U_y = R_z = 0.0$ at all nodes and $U_z = 0$ along all 4 edges

$R_x = 0.0$ along edges x=0 and x=10m and $R_y = 0.0$ along edges y=0 and y=10m.

Illustrations of the Model

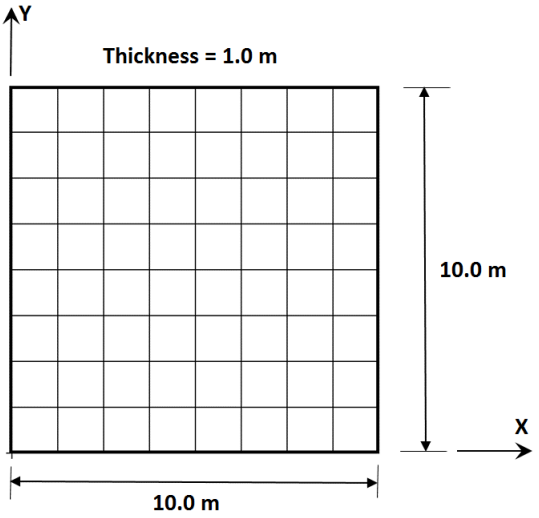


Figure 6.37-1 Simply Supported Thick Square Plate

Model File Name and Location

Model Name	Description	Location
`test21_q4.dat`	Test21: QUAD4 modal analysis	\verifman\nafems
`test21_q8.dat`	Test21: QUAD8 modal analysis	\verifman\nafems

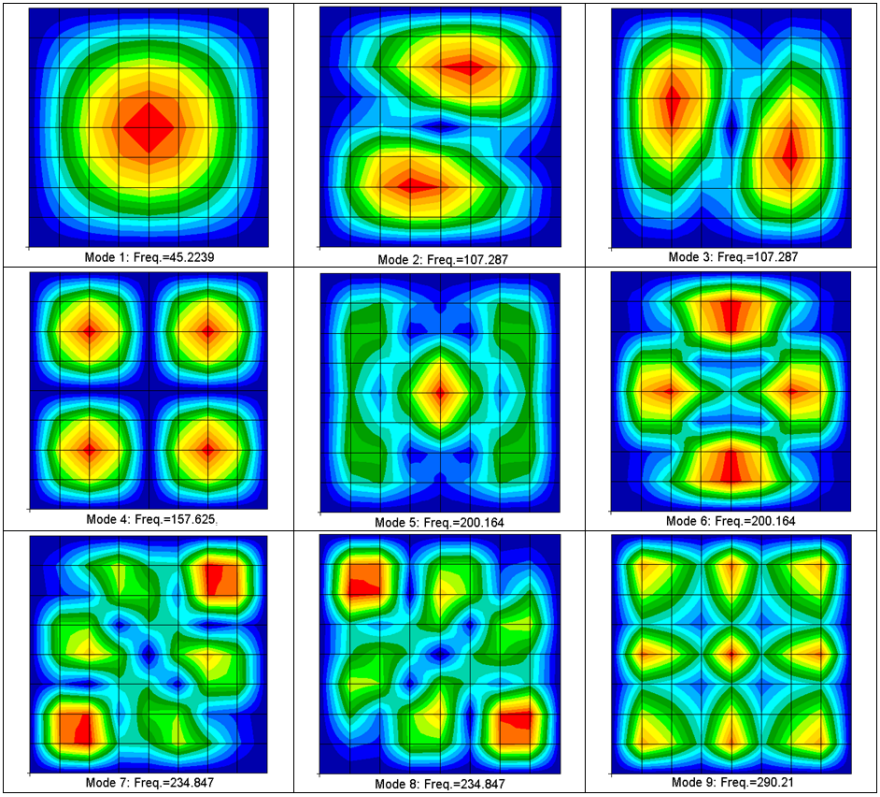


Figure 6.37-2 Mode Shapes of Thick Square Plate

Results

Mode	NAFEMS	MSC Nastran (QUAD4)		MSC Nastran (QUAD8)	
	Hz	Hz	% Error	Hz	% Error
1	45.897	45.22394	-1.47%	46.14619	0.54%
2 and 3	109.44	107.28680	-1.97%	110.25390	0.74%
4	167.89	157.62500	-6.11%	166.90790	-0.58%
5 and 6	204.51	200.16360	-2.13%	204.93380	0.21%
7 and 8	256.5	234.84710	-8.44%	248.16460	-3.25%

Conclusion

The results from MSC Nastran are compared with the NAFEMS results in the above table.

6.38 Test 21H: Simply Supported Thick Square Plate: Harmonic Forced Vibration

Problem Description

Forced harmonic vibration response of a thick square plate with modal and direct solutions.

Solution Number

SOL 108 and SOL 111

Features Used

- Harmonic response calculation using QUAD4 and QUAD8 shell elements.

Reference

1. Test 21H from NAFEMS Selected Benchmarks for Forced Vibration, R0016, March 1993.

Modeling Techniques Used

A simply supported thin plate is solved for a harmonic loading. In the direct method, Rayleigh damping is used with $a_0 = 5.772$ and $a_1 = 6.929 \times 10^{-5}$. In the modal approach, modal damping of 2% of critical is used on the first 16 modes. The model details are given below:

Material Properties

$E = 200 \times 10^9$ N/m^2 $\quad \nu = 0.3$ $\quad \rho = 8000$ kg/m^3

Loading

Steady state harmonic forcing function $F = F_0 \sin\omega t$

$F_0 = 10^6$ N/m^2 over whole plate with $\omega = 2\pi f$ and f = 0 to 78.17 Hz.

Boundary Conditions

$U_x = U_y = R_z = 0.0$ at all nodes and $U_z = 0$ along all 4 edges

$R_x = 0.0$ along edges x=0 and x=10m and $R_y = 0.0$ along edges y=0 and y=10m.

Main Index

Illustrations of the Model

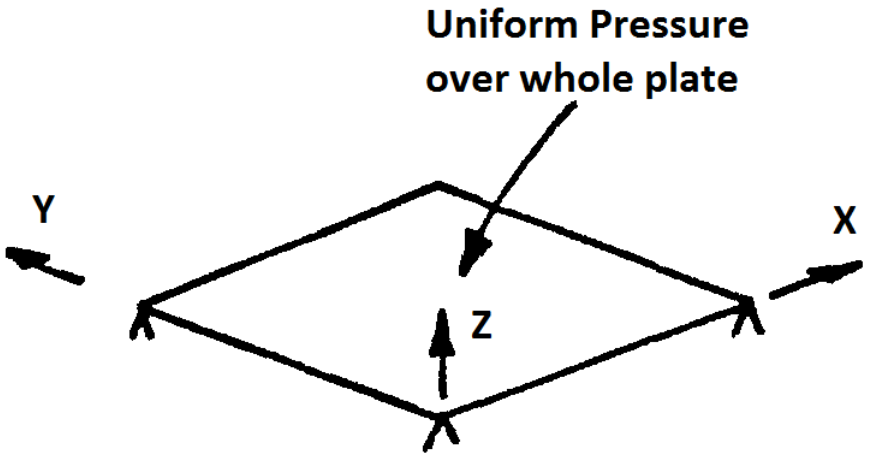


Figure 6.38-1 Simply Supported Thick Square Plate

Model File Name and Location

Model Name	Description	Location
`Test21hd_q4.dat`	Test21H: QUAD4 direct solution	\verifman\nafems
`Test21hd_q8.dat`	Test21H: QUAD8 direct solution	\verifman\nafems
`Test21hm_q4.dat`	Test21H: QUAD4 modal solution	\verifman\nafems
`Test21hm_q8.dat`	Test21H: QUAD8 modal solution	\verifman\nafems

Results

	Peak Displacement δ (mm)		
	NAFEMS	MSC Nastran	
		Values	% Error
QUAD4 (Modal)	58.33	61.1372	4.81%
QUAD4 (Direct)	58.33	61.1334	4.81%
QUAD8 (Modal)	58.33	60.0149	2.89%
QUAD8 (Direct)	58.33	60.0176	2.89%

	Peak Stress σ (N/mm^2)		
	NAFEMS	MSC Nastran	
		Values	% Error
QUAD4 (Modal)	800.8	776.6374	-3.02%
QUAD4 (Direct)	800.8	776.5670	-3.03%
QUAD8 (Modal)	800.8	877.5689	9.59%
QUAD8 (Direct)	800.8	877.5689	9.59%

	Frequency f (Hz)		
	NAFEMS	MSC Nastran	
		Values	% Error
QUAD4 (Modal)	45.9	45.2339	-1.47%
QUAD4 (Direct)	45.9	45.2339	-1.47%
QUAD8 (Modal)	45.9	46.1462	0.54%
QUAD8 (Direct)	45.9	46.1462	0.54%

Conclusion

The results from MSC Nastran are compared with the NAFEMS results in the above tables.

6.39 Test 21P: Simply Supported Thick Square Plate: Periodic Forced Vibration

Problem Description

Forced periodic vibration response of a thick square plate with modal and direct solutions.

Solution Number

SOL 108 and SOL 111

Features Used

- Harmonic response calculation using QUAD4 and QUAD8 shell elements.

Reference

1. Test 21P from NAFEMS Selected Benchmarks for Forced Vibration, R0016, March 1993.

Modeling Techniques Used

A simply supported thin plate is solved for a harmonic loading. In the direct method, Rayleigh damping is used with $a_0 = 5.772$ and $a_1 = 6.929 \times 10^{-5}$. In the modal approach, modal damping of 2% of critical is used on the first 16 modes. The model details are given below:

Material Properties

$E = 200 \times 10^9$ N/m^2 $\nu = 0.3$ $\rho = 8000$ kg/m^3

Loading

Steady state harmonic forcing function $F = F_0 (\sin\omega t - \sin 3\omega t)$

$F_0 = 10^6$ N/m^2 over whole plate with $\omega = 2\pi f$ and $f = 20$ Hz.

Boundary Conditions

$U_x = U_y = R_z = 0.0$ at all nodes and $U_z = 0$ along all 4 edges

$R_x = 0.0$ along edges x=0 and x=10m and $R_y = 0.0$ along edges y=0 and y=10m.

Illustrations of the Model

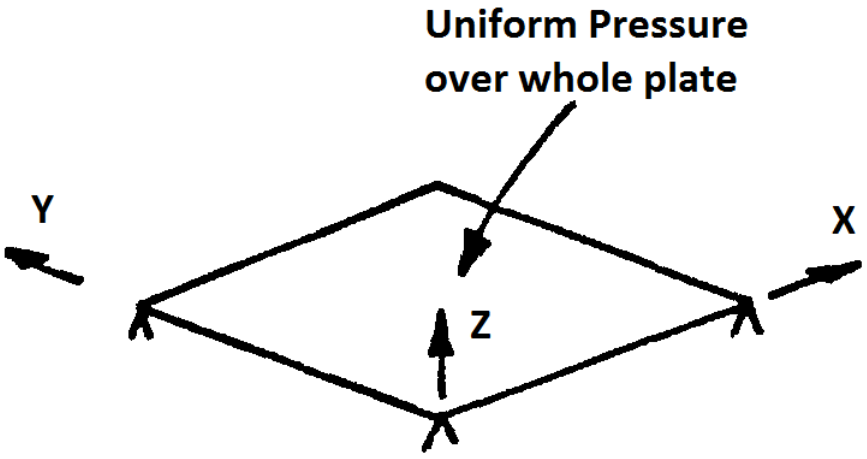


Figure 6.39-1 Simply Supported Thick Square Plate

Model File Name and Location

Model Name	Description	Location
`Test21pd_q4.dat`	Test21H: QUAD4 direct solution	\verifman\nafems
`Test21pd_q8.dat`	Test21H: QUAD8 direct solution	\verifman\nafems
`Test21pm_q4.dat`	Test21H: QUAD4 modal solution	\verifman\nafems
`Test21pm_q8.dat`	Test21H: QUAD8 modal solution	\verifman\nafems

Results

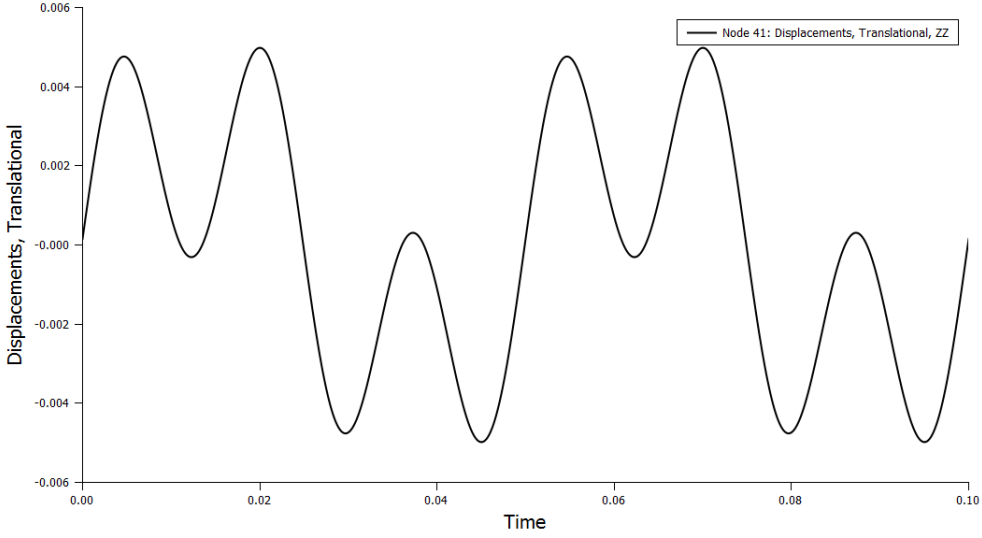


Figure 6.39-2 Displacement versus Time plot at node 41

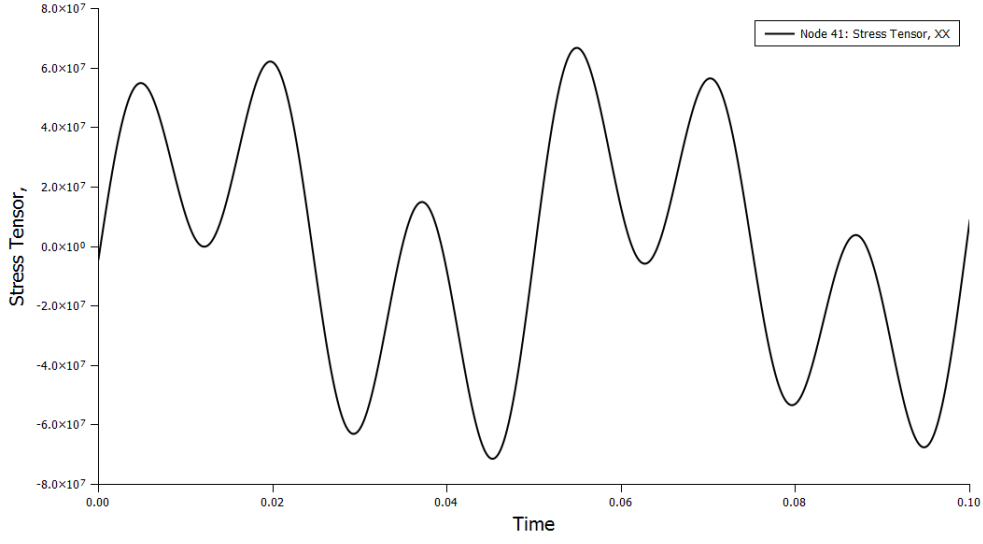


Figure 6.39-3 Stress versus Time plot at node 41

Results

	Peak Displacement δ (mm)			Peak Stress σ (N/mm^2)		
		MSC Nastran			MSC Nastran	
	NAFEMS	Values	% Error	NAFEMS	Values	% Error
QUAD4 (Modal)	4.929	4.9828	1.09%	67.67	64.1107	-5.26%
QUAD4 (Direct)	4.929	4.8454	-1.70%	67.67	62.2009	-8.08%
QUAD8 (Modal)	4.929	5.1885	5.27%	67.67	77.4392	14.44%
QUAD8 (Direct)	4.929	4.9322	0.06%	67.67	73.5257	8.65%

Conclusion

The displacement and stress results from the direct solution with QUAD4 elements are shown in Figure 6.39-2 and Figure 6.39-3. The results from MSC Nastran are compared with the NAFEMS results in the above table.

6.40 Test 21T: Simply Supported Thick Square Plate: Transient Forced Vibration

Problem Description

Forced transient vibration response of a thick square plate with modal and direct solutions.

Solution Number

SOL 109 and 112

Features Used

- Random transient calculation using QUAD4 and QUAD8 shell elements

Reference

1. Test 21T from NAFEMS Selected Benchmarks for Forced Vibration, R0016, March 1993.

Modeling Techniques Used

A simply supported thick plate is solved for a transient loading. In the direct method, Rayleigh damping is used with $a_0 = 5.772$ and $a_1 = 6.929 \times 10^{-5}$. In the modal approach, modal damping of 2% of critical is used on the first 16 modes. The model details are given below:

Material Properties

E	=	200×10^9 N/m^2
ν	=	0.3
ρ	=	8000 kg/m^3

Loading

Suddenly applied step load $F_0 = 10^6$ N/m^2 over whole plate

Boundary Condition

$u_x = u_y = R_z = 0.0$ at all nodes and $U_z = 0$ along all 4 edges

R_x = 0.0 along edges x=0 and x=10m and R_y = 0.0 along edges y=0 and y=10m

Illustrations of the Model

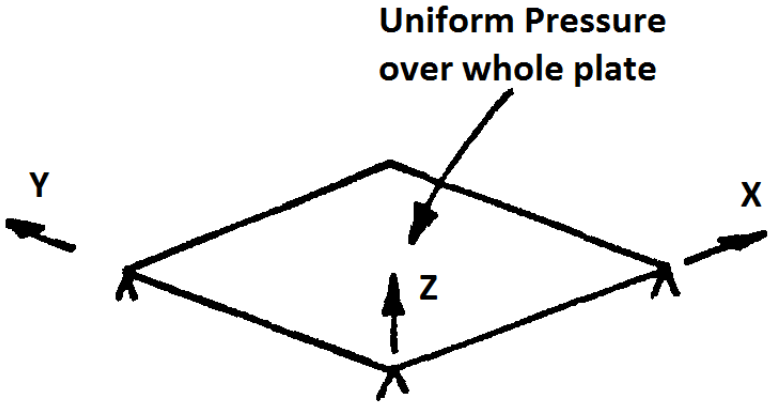


Figure 6.40-1 Simply Supported Thick Square Plate

Model File Name and Location

Model Name	Description	Location
Test21td_q4.dat	Test21T: QUAD4 direct solution	tpl\verifman\nafems
Test21td_q8.dat	Test21T: QUAD8 direct solution	
Test21tm_q4.dat	Test21T: QUAD4 modal solution	
Test21tm_q8.dat	Test21T: QUAD8 modal solution	

Results

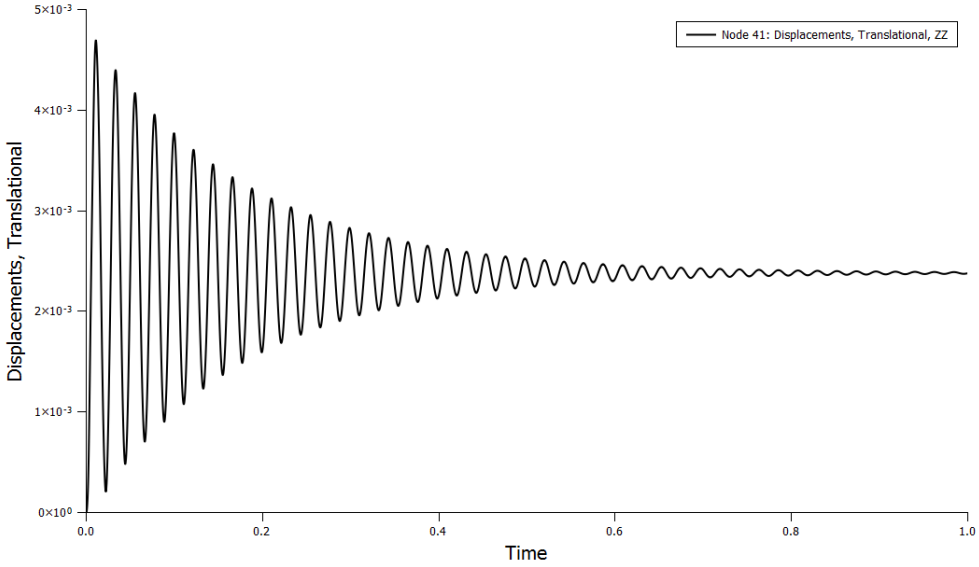


Figure 6.40-2 Displacement versus Time plot at node 41

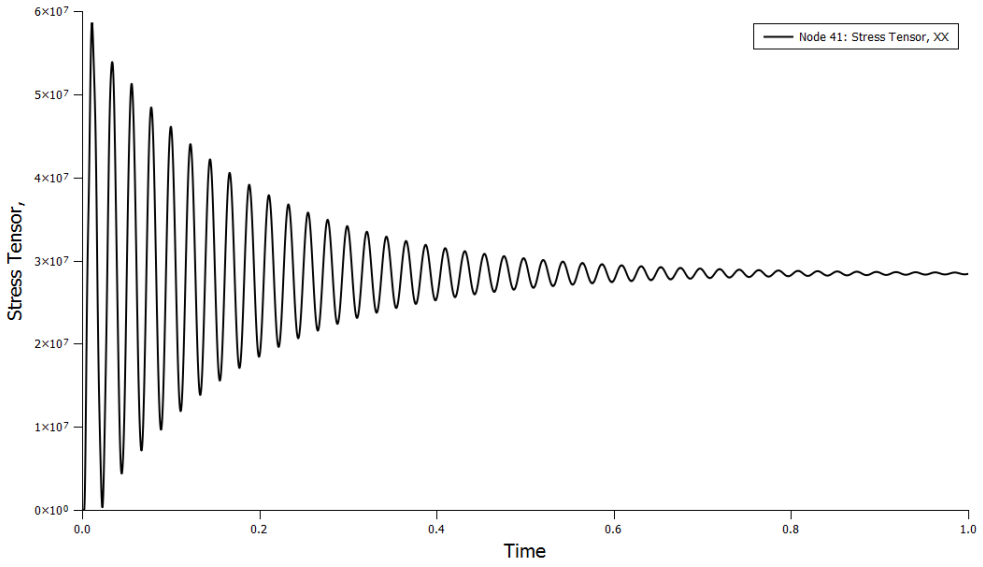


Figure 6.40-3 Stress versus Time plot at node 41

	Peak Displacement δ_p (mm)		
		MSC Nastran	
	NAFEMS	Values	% Error
QUAD4 (Modal)	4.524	4.7026	3.95%
QUAD4 (Direct)	4.524	4.6947	3.77%
QUAD8 (Modal)	4.524	4.6047	1.78%
QUAD8 (Direct)	4.524	4.5977	1.63%

	Peak Stress σ (N/mm^2)		
		MSC Nastran	
	NAFEMS	Values	% Error
QUAD4 (Modal)	62.11	59.1492	-4.77%
QUAD4 (Direct)	62.11	58.6496	-5.57%
QUAD8 (Modal)	62.11	66.3350	6.80%
QUAD8 (Direct)	62.11	65.7299	5.83%

	Peak Response Time (t_p)		
		MSC Nastran	
	NAFEMS	Values	% Error
QUAD4 (Modal)	0.0108	0.0108	0.00%
QUAD4 (Direct)	0.0108	0.0109	0.93%
QUAD8 (Modal)	0.0108	0.0105	-2.78%
QUAD8 (Direct)	0.0108	0.0107	-0.93%

	Static Displacement δ_s (mm)		
		MSC Nastran	
	NAFEMS	Values	% Error
QUAD4 (Modal)	2.333	2.3816	2.08%
QUAD4 (Direct)	2.333	2.3816	2.08%
QUAD8 (Modal)	2.333	2.3273	-0.25%
QUAD8 (Direct)	2.333	2.3274	-0.24%

Conclusion

The displacement and stress results from the direct solution with QUAD4 elements are shown in Figure 6.40-2 and Figure 6.40-3. The results from MSC Nastran are compared with the NAFEMS results in the above tables.

6.41 Test 21R: Simply Supported Thick Square Plate: Random Forced Vibration

Problem Description

Forced random vibration response of a thick square plate with modal and direct solutions.

Solution Number

SOL 108 and 111

Features Used

- Random response calculation using QUAD4 and QUAD8 shell elements

Reference

1. Test 21R from NAFEMS Selected Benchmarks for Forced Vibration, R0016, March 1993.

Modeling Techniques Used

A simply supported thick plate is solved for a random loading. In the direct method, Rayleigh damping is used with $a_0 = 5.772$ and $a_1 = 6.929 \times 10^{-5}$. In the modal approach, modal damping of 2% of critical is used on the first 16 modes. The model details are given below:

Material Properties

$E = 200 \times 10^9\ N/m^2$

$\nu = 0.3$

$\rho = 8000\ kg/m^3$

Loading

Random forcing with uniform power spectral density (of force) PSD = $(10^6\ N/m^2)^2$ / Hz over the whole plate.

Boundary Condition

$u_x = u_y = R_z = 0.0$ at all nodes and $U_z = 0$ along all 4 edges

R_x = 0.0 along edges x=0 and x=10m and R_y = 0.0 along edges y=0 and y=10m

Illustrations of the Model

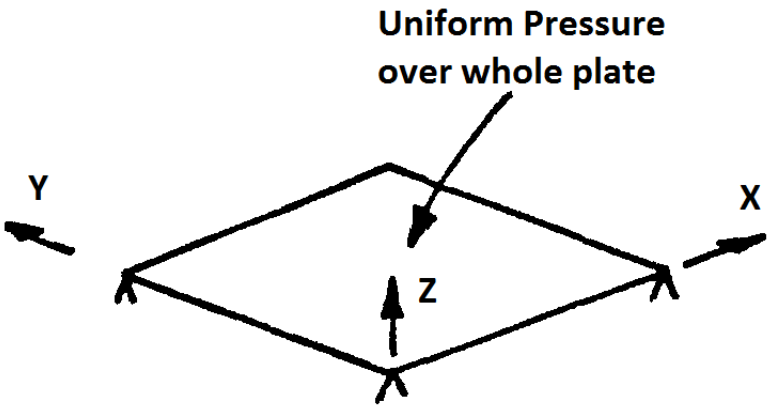


Figure 6.41-1 Simply Supported Thick Square Plate

Model File Name and Location

Model Name	Description	Location
Test21rd_q4.dat	Test21R: QUAD4 direct solution	tpl\verifman\nafems
Test21rd_q8.dat	Test21R: QUAD8 direct solution	
Test21rm_q4.dat	Test21R: QUAD4 modal solution	
Test21rm_q8.dat	Test21R: QUAD8 modal solution	

Results

	Peak δ PSD (mm^2/Hz)		
	NAFEMS	MSC Nastran	
		Values	% Error
QUAD4 (Modal)	3401.81	3737.7530	9.88%
QUAD4 (Direct)	3401.81	3737.2870	9.86%
QUAD8 (Modal)	3401.81	3601.7850	5.88%
QUAD8 (Direct)	3401.81	3602.1140	5.89%

	Peak σ PSD ($((N/mm^2)^2/Hz)$)		
	NAFEMS	MSC Nastran	
		Values	% Error
QUAD4 (Modal)	641200	603173.1	-5.93%
QUAD4 (Direct)	641200	603063.7	-5.95%
QUAD8 (Modal)	641200	770148.1	20.11%
QUAD8 (Direct)	641200	770148.2	20.11%

	Frequency f (Hz)		
	NAFEMS	MSC Nastran	
		Values	% Error
QUAD4 (Modal)	45.9	45.2239	-1.47%
QUAD4 (Direct)	45.9	45.2239	-1.47%
QUAD8 (Modal)	45.9	46.1462	0.54%
QUAD8 (Direct)	45.9	46.1462	0.54%

Conclusion

The results from MSC Nastran are compared with the NAFEMS results in the above tables.

Nonlinear

6.42 NL4: Snap-back Under Displacement Control

Problem Description

This is a NAFEMS nonlinear benchmark to demonstrate a geometric nonlinearity solution procedure test of snap-back under displacement control. It is modeled using the modified Riks-Ramm arc length procedure available in MSC Nastran.

Solution Number

SOL 400

Features Used

- Implicit nonlinear analysis.
- Modified Riks/Ramm arc length method.

Reference

1. Test NL4 from NAFEMS Publication NNB, Rev. 1, NAFEMS Non-Linear Benchmarks, October 1989 from a previous report Benchmark Tests for Solution Procedures for Geometric Non-Linearity by Crisfield, Duxbury & Hunt. NAFEMS Report SPGNL, October 1987.

Modeling Techniques Used

The spring and element assembly are modeled using CELAS1 and CROD elements respectively. The model details are given below:

Material Properties

$AE = 5.0x10^7$, $L = 2500$, $\alpha L = 25$, $K_1 = 1.5$, $K_2 = AE/L(1 + \alpha^2)^{1/2} = 19999.0$, $K_3 = 0.25$, $K_4 = 1.0$

Loading

Incremental load P=10000.0 is applied to node A in the X-direction using arc length method.

Boundary Condition

The displacements $u_y = 0.0$ at node A and B, along with the displacement $u_x = 0.0$ at node C.

Illustrations of the Model

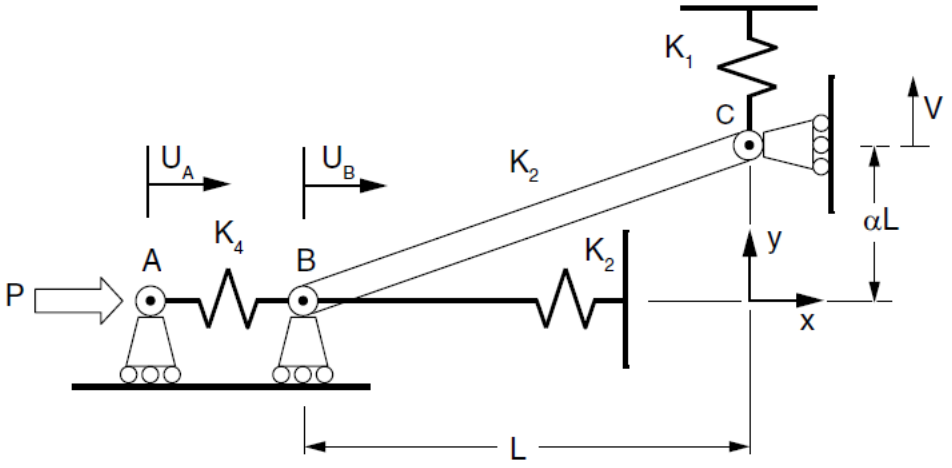


Figure 6.42-1 Snap-back under displacement control

Model File Name and Location

Model File Name	Description	Location
nafemnl4.dat	NL4: Snap-back Under Displacement Control	\verifman\nafems

Results

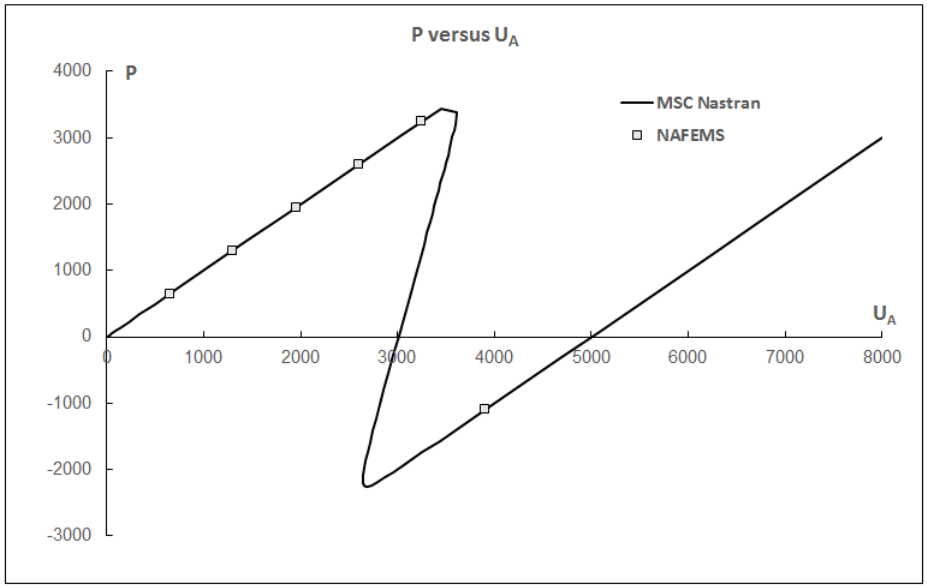


Figure 6.42-2 P versus U_A Solution Path

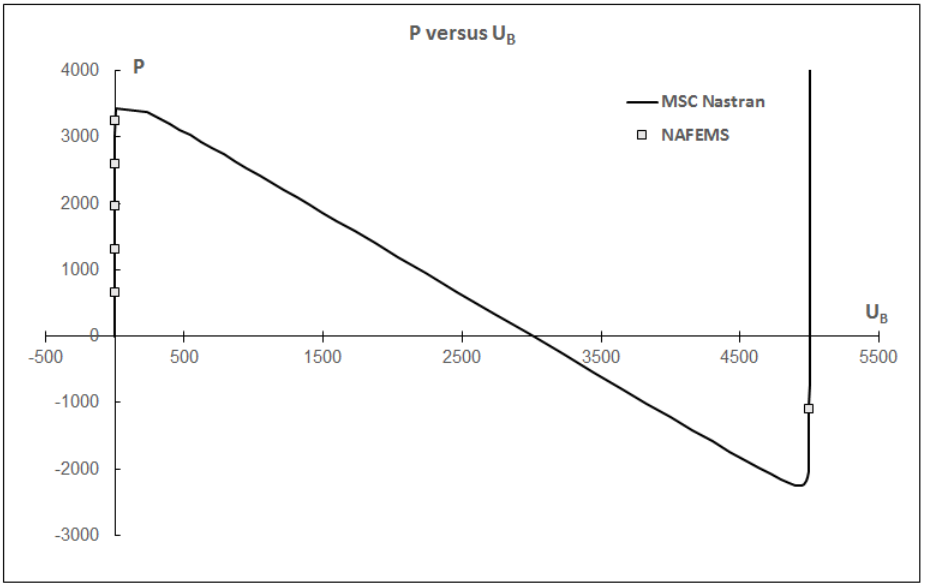


Figure 6.42-3 P versus U_A Solution Path

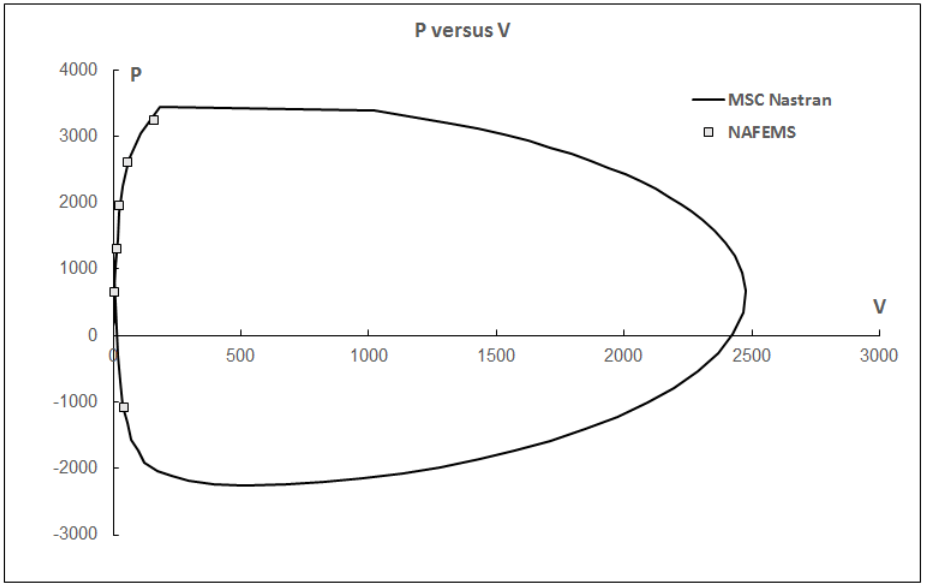


Figure 6.42-4 P versus V Solution Path

Main Index

Conclusion

The results from MSC Nastran are compared with NAFEMS results at loads P= 649.9, 1300.0, 1949.0, 2599.0, 3243.0 and -1099.0 in the above figures. The largest deviation from the NAFEMS solution is less than 1% at all these reference points.

6.43 NL6: Straight Cantilever with Axial End Point Load

Problem Description

A cantilever beam with an axial compressive load combines bending and membrane deformation with bifurcation of initially straight elements using the modified Riks-Ramm arc length procedure available in MSC Nastran.

Solution Number

SOL 400

Features Used

- Implicit nonlinear analysis.
- Modified Riks/Ramm arc length method.

Reference

1. Test NL6 from NAFEMS Publication NNB, Rev. 1, NAFEMS Non-Linear Benchmarks, October 1989 as well as a previous report Finite Element Benchmarks for 2D beams and Axisymmetric shells involving Geometric Non-Linearity by P Lyons and S Holsgrove NAFEMS Report FEBNLGBAS, March 1989.

Modeling Techniques Used

The cantilever beam is modeled using 32 plane stress quadratic CQUAD8 elements. The model details are given below:

Material Properties

The material is elastic with a Young's modulus of E = 210 x 10^9 N/m^2 and a Poisson's ratio of 0.0.

Loading

The loading at end A is applied in increments up to a maximum value of PL^2/EI = 22.493.

Boundary Condition

The displacements $u_x = u_y = 0.0$ at node 1 and $u_x = 0.0$ at nodes 2 and 3 of the end B.

Illustrations of the Model

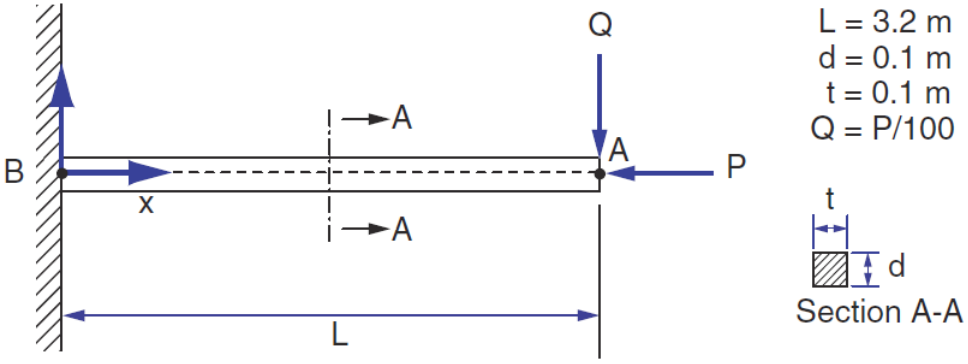


Figure 6.43-1 Straight Cantilever Beam with Axial End Point Load

Model File Name and Location

Model File Name	Description	Location
`nafemnl6.dat`	NL6: Straight Cantilever With Axial End Point Load	\verifman\nafems

Results

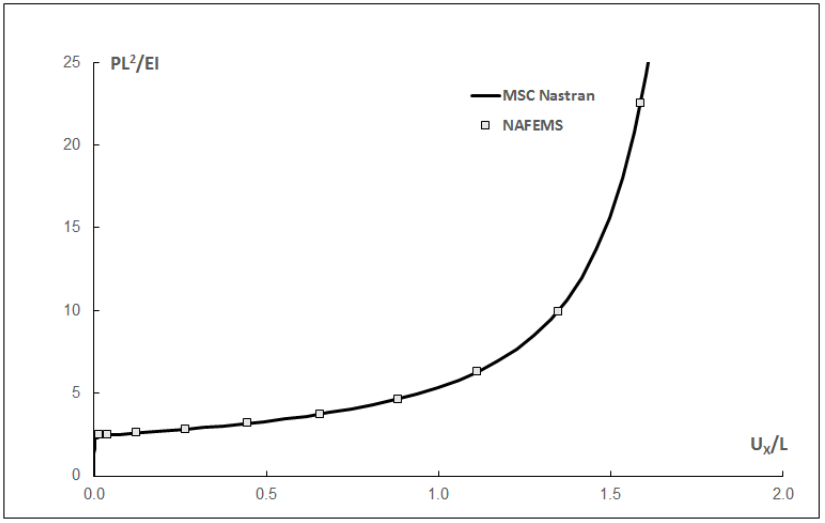


Figure 6.43-2 PL^2/EI versus U_X/L Solution Path

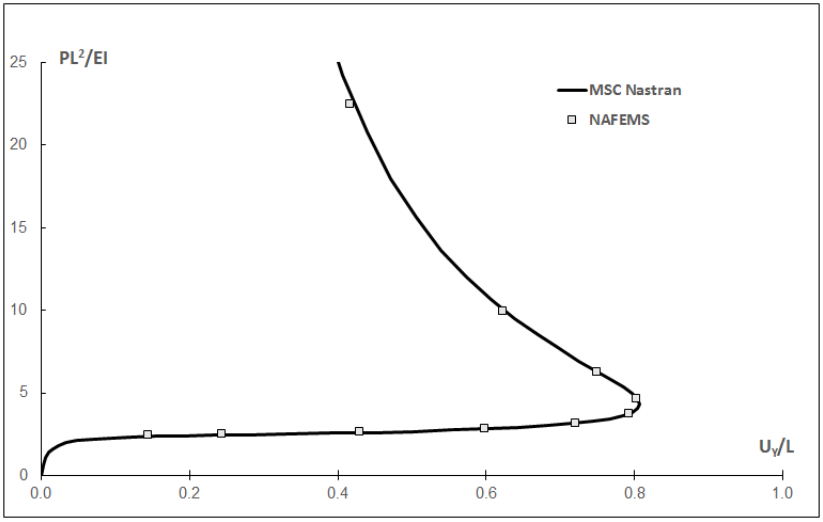


Figure 6.43-3 PL^2/EI versus U_Y/L Solution Path

Conclusion

The results from MSC Nastran are matching well with NAFEMS results as shown in the above figures.

6.44 NL7: Lee's Frame Buckling Problem

Problem Description

This example demonstrates the ability of MSC Nastran to model a beam column structure composed of initially straight beams that will suddenly snap through and back. Modified Riks/Ramm & Crisfield arc length methods are used to trace the solution path to this unstable problem.

Solution Number

SOL 400

Features Used

- Implicit nonlinear analysis.
- Modified Riks/Ramm and Crisfield arc length methods.

Reference

1. Finite Element Benchmarks for 2D beams and Axisymmetric shells involving Geometric Non-Linearity by P Lyons and S Holsgrove NAFEMS Report FEBNLGBAS, March 1989.

Modeling Techniques Used

The frame is modeled using 100 CBAR elements. The model details are given below:

Material Properties

The material is elastic with a Young's modulus of E = 71.74 x 10^9 N/m^2 and a Poisson's ratio of 0.0.

Loading

A concentrated load P at point C with an ultimate value of 50000 N is applied incrementally.

Boundary Condition

The beams are pinned at points B and D.

Illustrations of the Model

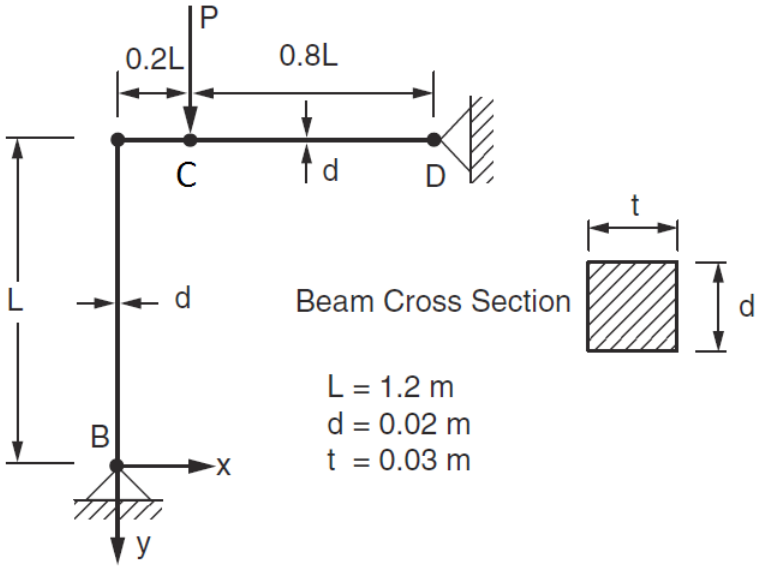


Figure 6.44-1 Lee's Frame Buckling Problem

Model File Name and Location

Model File Name	Description	Location
nafemnl7a.dat	NL7: Lee's Frame Buckling Problem with MRIKS method	\verifman\nafems
nafemnl7b.dat	NL7: Lee's Frame Buckling Problem with Crisfield method	\verifman\nafems

Results

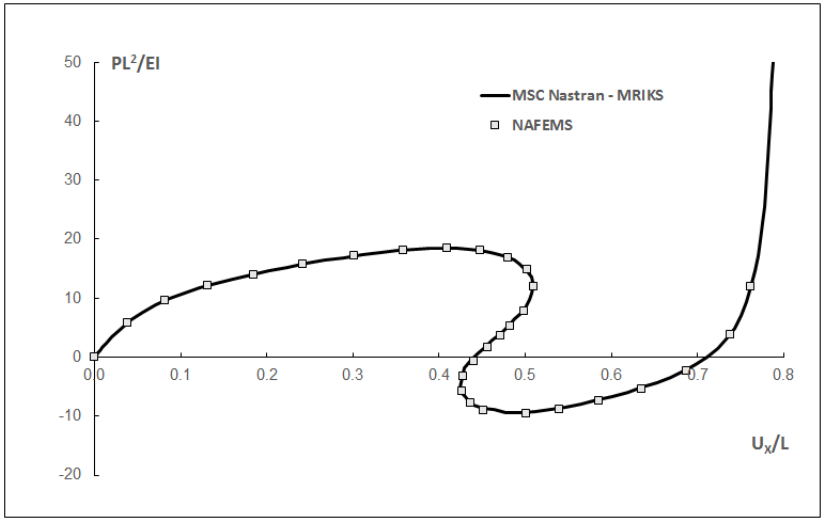


Figure 6.44-2 Lee's Frame Buckling with Modified Riks/Ramm Method

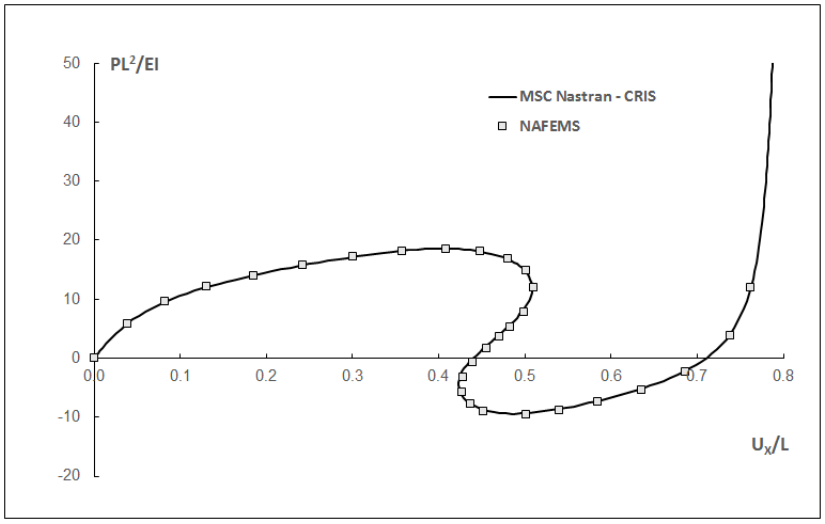


Figure 6.44-3 Lee's Frame Buckling with Crisfield Method

Conclusion

The results from MSC Nastran are matching well with NAFEMS results as shown in the above figures for both modified Riks/Ramm and Crisfield methods.

Fracture Mechanics

6.45 Test 1.1: Center Cracked Plate in Tension

Problem Description

The following test case describe finite element analysis of a centre cracked plate tension specimen. Normalized stress intensity factor for the test case is calculated using virtual crack extension method. The results are compared to the analytical solution [1].

Solution Number

Sol 400

Features Used

- Implicit Nonlinear

Reference

1. Test 1.1 from NAFEMS Publication, Ref: R0020, 2D Test Cases in Linear Elastic Fracture Mechanics, Jan,1992.

Modeling Techniques Used

Due to symmetry in loading and boundary conditions one-quarter of test geometry along the symmetry axis is used for the analysis. The finite element model is shown in Figure 6.45-1. The model is meshed with QUAD8 and quarter point TRIA6 elements. The quarter point elements are used at the crack tip. VCCT bulk data entry is used to obtain the energy release rate.

Material Properties

The material is linear elastic with a Young's modulus, E = 20700 and Poisson's ratio ν = 0.3

Geometrical properties

a/b = 0.5; h/b = 1.0; b = 20

Loading

Uniform stress of 100 unit is applied on top face as shown in Figure 6.45-1.

Boundary Condition

Symmetry boundary condition as shown in Figure 6.45-1.

$u_y = 0$ on x axis

$u_x = 0$ on y axis

Illustrations of the Model

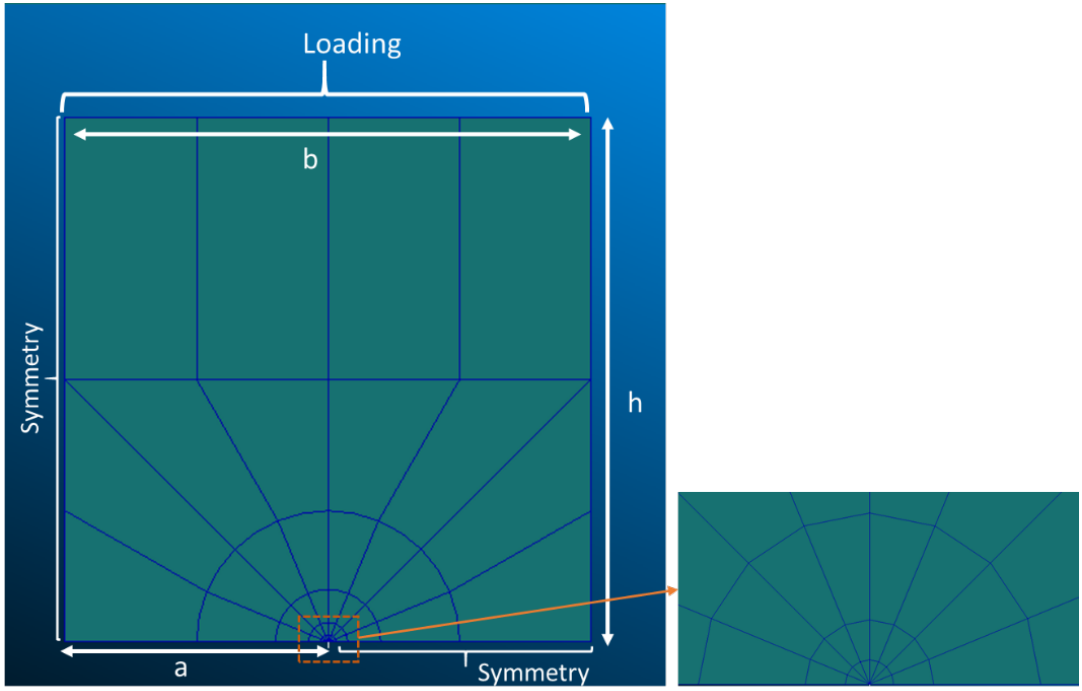


Figure 6.45-1 One Quarter of Centre Cracked Plate

Model File Name and Location

Model Name	Description	Location
nvcct001	Centre Cracked Plate in Tension, Plain Strain	tpl\verifman\nafems

Results

Reference solution [1] of the normalized stress intensity factor is $\frac{K_I}{K_o} = 1.325$ where $K_o = \sigma(\pi a)^{0.5}$

Virtual Crack Extension Method

The details of this method is explained in NAFEMS fracture mechanics benchmarks article [1].

Energy release rate (G)= 2.4665

Stress intensity factor $K_I = \left(\frac{GE}{1-\nu^2}\right)^{0.5} = 749.0$

$K_o = \sigma(\pi a)^{0.5} = 560.5$

$\frac{K_I}{K_o} = 1.3364$

Conclusion

The results from MSC Nastran are in a good agreement with the reference solution.

6.46 Test 1.2: Center Cracked Plate with Thermal Load

Problem Description

The following test case describe finite element analysis of a center cracked plate subjected to quadratic temperature distribution. Normalized stress intensity factor for the test case is calculated using virtual crack extension method. The results are compared to the analytical solution [1].

Solution Number

Sol 400

Features Used

- Implicit Nonlinear

Reference

1. Test 1.2 from NAFEMS Publication, Ref: R0020, 2D Test Cases in Linear Elastic Fracture Mechanics, Jan,1992.

Modeling Techniques Used

Due to symmetry in loading and boundary conditions one-quarter of test geometry along the symmetry axis is used for the analysis. The finite element model is shown in Figure 6.45-1. The model is meshed with QUAD8 and quarter point TRIA6 elements. The quarter point elements are used at the crack tip. VCCT bulk data entry is used to obtain the energy release rate.

Material Properties

The material is linear elastic with a Young's modulus E = 20700 and a Poisson's ratio $\nu = 0.3$ and coefficient of thermal expansion $\alpha = 1.35 \times 10^{-5}$

Geometrical properties

a/b = 0.1; h/b = 2.5; b = 100

Loading

Quadratic thermal distribution

$$T = 0.01 \times x^2$$

Boundary Condition

Symmetry boundary condition as shown in Figure 6.45-1.

$u_y = 0$ on x axis

$u_x = 0$ on y axis

Illustrations of the Model

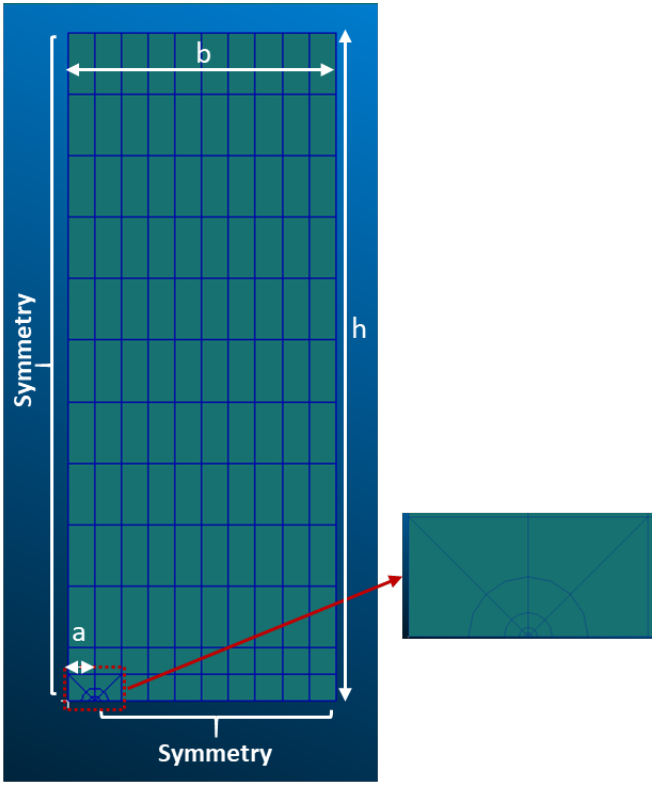


Figure 6.46-1 One Quarter of Centre Cracked Plate

Model File Name and Location

Model Name	Description	Location
nvcct002	Centre cracked plate with quadratic thermal distribution, Plane Stress	tpl\verifman\nafems

Results

Reference solution [1] of the normalized stress intensity factor is $\frac{K_I}{K_o} = 1.00$ where $K_o = \frac{E\alpha T}{c^2}\left(\frac{b^2}{3} - \frac{a^2}{2}\right)(\pi a)^{0.5}$

Virtual Crack Extension Method

The details of this method is explained in NAFEMS fracture mechanics benchmarks article [1].

Energy release rate $(G) = 1.3004$

Stress intensity factor $K_1 = (GE)^{0.5} = 518.9$

$K_0 = \sigma(\pi a)^{0.5} = 514.3$

$\frac{K_I}{K_o} = 1.008$

Conclusion

The results from MSC Nastran are in a good agreement with the reference solution.

6.47 Test 2.1: Single Edge Cracked Plate in Tension

Problem Description

The following test case describe finite element analysis of a Single Edge Cracked Plate subjected to uniform stress. Normalized stress intensity factor for the test case is calculated using virtual crack extension method. The results are compared to the analytical solution [1].

Solution Number

Sol 400

Features Used

- Implicit Nonlinear

Reference

1. Test case 2.1 from NAFEMS Publication, Ref: R0020, 2D Test Cases in Linear Elastic Fracture Mechanics, Jan,1992.

Modeling Techniques Used

Due to symmetry in loading and boundary conditions one-quarter of test geometry along the symmetry axis is used for the analysis. The finite element model is shown in Figure 6.47-1. The model is meshed with QUAD8 and quarter point TRIA6 elements. The quarter point elements are used at the crack tip. VCCT bulk data entry is used to obtain the energy release rate.

Material Properties

The material is linear elastic with a Young's modulus of E = 20700 and a Poisson's ratio ν = 0.3

Geometrical properties

a/b = 0.5; h/b = 0.5; b = 20

Loading

Uniform stress of 100 unit is applied on top face as shown in Figure 6.47-1.

Boundary Condition

Symmetry boundary condition as shown in Figure 6.47-1.

$u_y = 0$ on x axis

$u_x = 0$ at point A

Illustrations of the Model

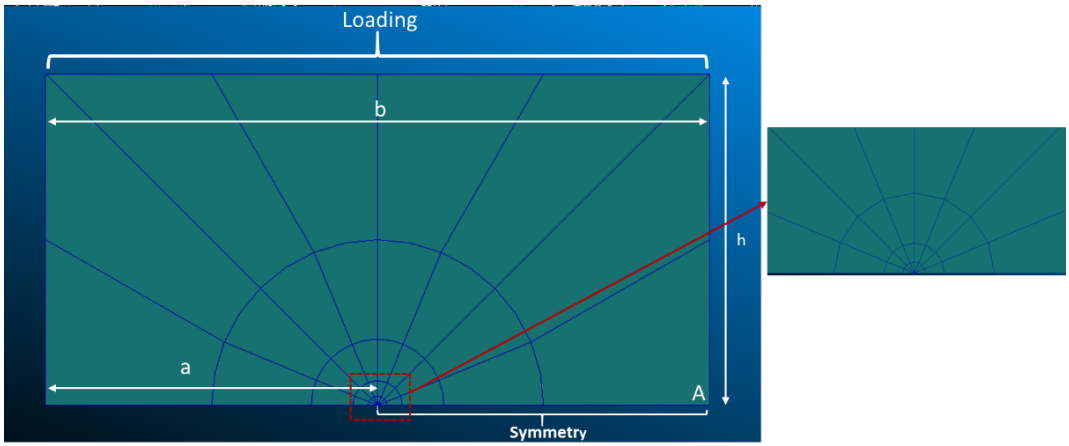


Figure 6.47-1 Half of the Single Edge Cracked Plate

Model File Name and Location

Model Name	Description	Location
nvcct003	Single edge cracked plate: uniform tensile stress, Plane Strain	tpl\verifman\nafems

Results

Reference solution [1] of the normalized stress intensity factor is $\frac{K_I}{K_o} = 3.00$ where $K_o = \sigma(\pi a)^{0.5}$

Virtual Crack Extension Method

The details of this method is explained in NAFEMS fracture mechanics benchmarks article [1].

Energy release rate $(G) = 12.468$

Stress intensity factor $K_I = \left(\frac{GE}{1-\nu^2}\right)^{0.5} = 1684.1$

$K_o = \sigma(\pi a)^{0.5} = 560.5$

$$\frac{K_I}{K_o} = 3.00$$

Conclusion

The results from MSC Nastran are in a good agreement with the reference solution.

6.48 Test 3: Angle Crack Embedded in a Plate

Problem Description

The following test case describe finite element analysis of an angle crack plate subjected to uniform stress. Normalized stress intensity factor for the test case is calculated using virtual crack extension method. The results are compared to the analytical solution [1].

Solution Number

Sol 400

Features Used

- Implicit Nonlinear

Reference

1. Test case 3.1 from NAFEMS Publication, Ref: R0020, 2D Test Cases in Linear Elastic Fracture Mechanics, Jan,1992.

Modeling Techniques Used

The finite element model is shown in Figure 6.48-1. The model is meshed with QUAD8 and quarter point TRIA6 elements. The quarter point elements are used at the crack tip. VCCT bulk data entry is used to obtain the energy release rate.

Material Properties

The material is linear elastic with a Young's modulus of E = 20700 and a Poisson's ratio ν = 0.3

Geometrical properties

a/b = 0.5; h/b = 1.25; b = 50

Loading

Uniform stress of 100 unit is applied on top face as shown in Figure 6.48-1.

Boundary Condition

Boundary condition is shown in Figure 6.48-1.

$u_y = 0$ on x axis at y = 0

$u_x = 0$ at point A

Illustrations of the Model

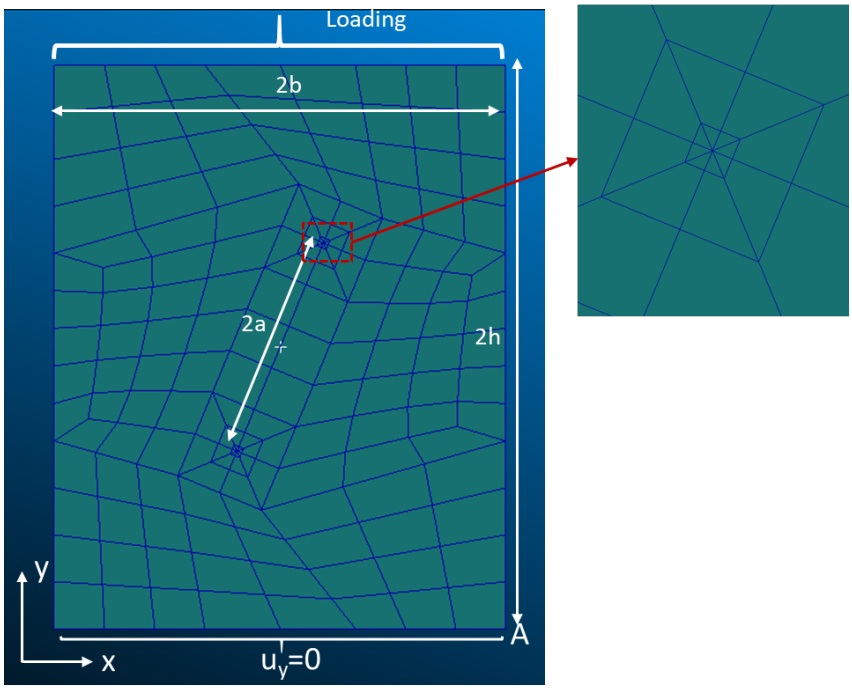


Figure 6.48-1 Angle Crack Plate

Model File Name and Location

Model Name	Description	Location
nvcct004	Angle crack plate embedded in a plate: Uniaxial tension, Plane Strain	tpl\verifman\nafems

Results

Reference solution [1] of the normalized stress intensity factor is $\frac{K_I}{K_o} = 0.190$; $\frac{K_{II}}{K_o} = 0.405$ where

$K_o = \sigma(\pi a)^{0.5}$

Virtual Crack Extension Method

The details of this method is explained in NAFEMS fracture mechanics benchmarks article [2].

Energy release rate (G)= 0.6717

Ratio of mode II to mode I separation of crack face nodes (ΔR) = $\frac{\Delta U}{\Delta V}$ =2.1426

Stress intensity factor $K_I = \left(\frac{GE}{(1-\nu^2)(1-\Delta R^2)}\right)^{0.5} = 165.32$

$K_{II} = \Delta R K_I = 354.21$

$K_o = \sigma(\pi a)^{0.5} = 886.2$

$\frac{K_I}{K_o} = 0.186$, $\frac{K_{II}}{K_o} = 0.400$

Conclusion

The results from MSC Nastran are in a good agreement with the reference solution.

6.49 Test 4: Cracks at a Hole in a Plate

Problem Description

The following test case describe finite element analysis of cracks at a hole in a plate and subjected to uniform stress. Normalized stress intensity factor for the test case is calculated using virtual crack extension method. The results are compared to the analytical solution [1].

Solution Number

Sol 400

Features Used

- Implicit Nonlinear

Reference

1. Test case 4.1 from NAFEMS Publication, Ref: R0020, 2D Test Cases in Linear Elastic Fracture Mechanics, Jan,1992.

Modeling Techniques Used

Due to symmetry in loading and boundary conditions, one-quarter of test geometry along the symmetry axis is used for the analysis. The finite element model is shown in Figure 6.45-1. The model is meshed with QUAD8 and quarter point TRIA6 elements. The quarter point elements are used at the crack tip. VCCT bulk data entry is used to obtain the energy release rate.

Material Properties

The material is linear elastic with a Young's modulus E = 207000 and a Poisson's ratio ν = 0.3

Geometrical properties

a/b = 0.3; R/b = 0.25; b = 10

Loading

Uniform stress of 100 unit is applied on top face as shown in Figure 6.45-1.

Boundary Condition

Boundary condition as shown in Figure 6.45-1.

$u_y = 0$ on x axis

$u_x = 0$ on y axis

Illustrations of the Model

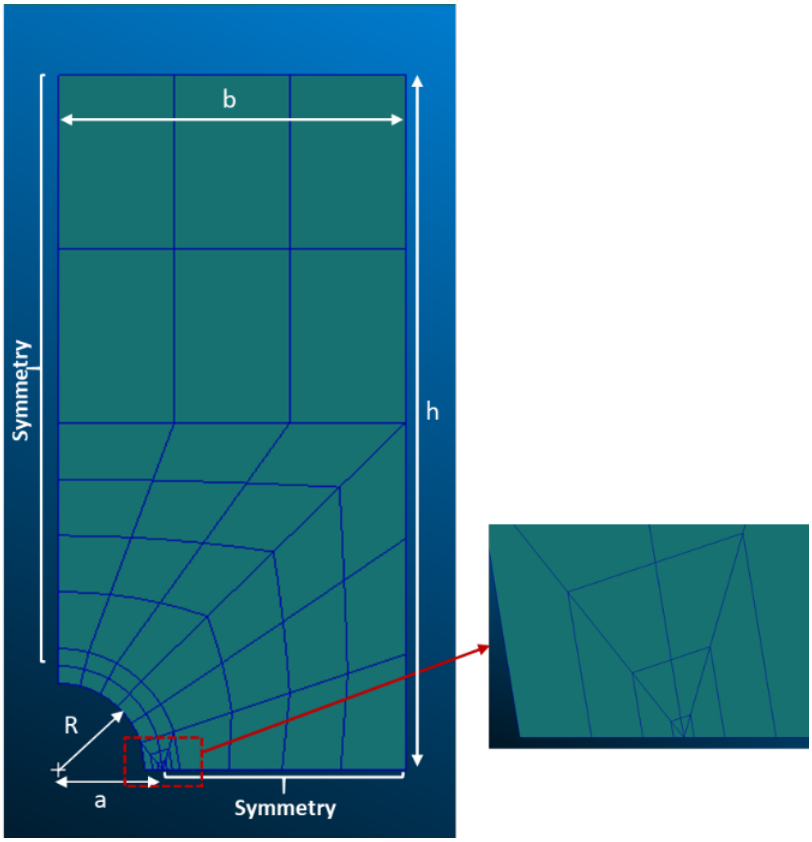


Figure 6.49-1 Cracks at a Hole in a Plate

Model File Name and Location

Model Name	Description	Location
nvcct005	Cracks at a Hole in a Plate, Plane Stress	tpl\verifman\nafems

Results

Reference solution [1] of the normalized stress intensity factor is $\frac{K_I}{K_o} = 1.050$ where $K_o = \sigma(\pi a)^{0.5}$

Virtual Crack Extension Method

The details of this method are explained in NAFEMS fracture mechanics benchmarks article [1].

Virtual Crack Extension Method:

Energy release rate $(G) = 0.5198$

Stress intensity factor $K_I = (GE)^{0.5} = 328.02$

$K_o = \sigma(\pi a)^{0.5} = 307.0$

$\frac{K_I}{K_o} = 1.068$

Conclusion

The results from MSC Nastran are in a good agreement with the reference solution.

6.50 Test 5: Axisymmetric Crack in a Bar

Problem Description

The following test case describe finite element analysis of axisymmetric crack in a bar and subjected to uniform stress. Normalized stress intensity factor for the test case is calculated using virtual crack extension method. The results are compared to the analytical solution [1].

Solution Number

Sol 400

Features Used

- Implicit Nonlinear

Reference

1. Test case 5 from NAFEMS Publication, Ref: R0020, 2D Test Cases in Linear Elastic Fracture Mechanics, Jan,1992.

Modeling Techniques Used

Due to symmetry in loading and boundary conditions one-quarter of test geometry along the symmetry axis is used for the analysis. The finite element model is shown in Figure 6.45-1. The model is meshed with CQUADX and quarter point CTRIAX elements. The quarter point elements are used at the crack tip. VCCT bulk data entry is used to obtain the energy release rate.

Material Properties

The material is linear elastic with a Young's modulus, E = 207000 and Poisson's ratio, ν= 0.3

Geometrical properties

b/R = 0.5; h/R = 1.5; R = 20

Loading

Uniform stress of 100 unit is applied on top face as shown in Figure 6.45-1.

Boundary Condition

Boundary condition as shown in Figure 6.45-1.

$u_y = 0$ on x axis

$u_x = 0$ on y axis

Illustrations of the Model

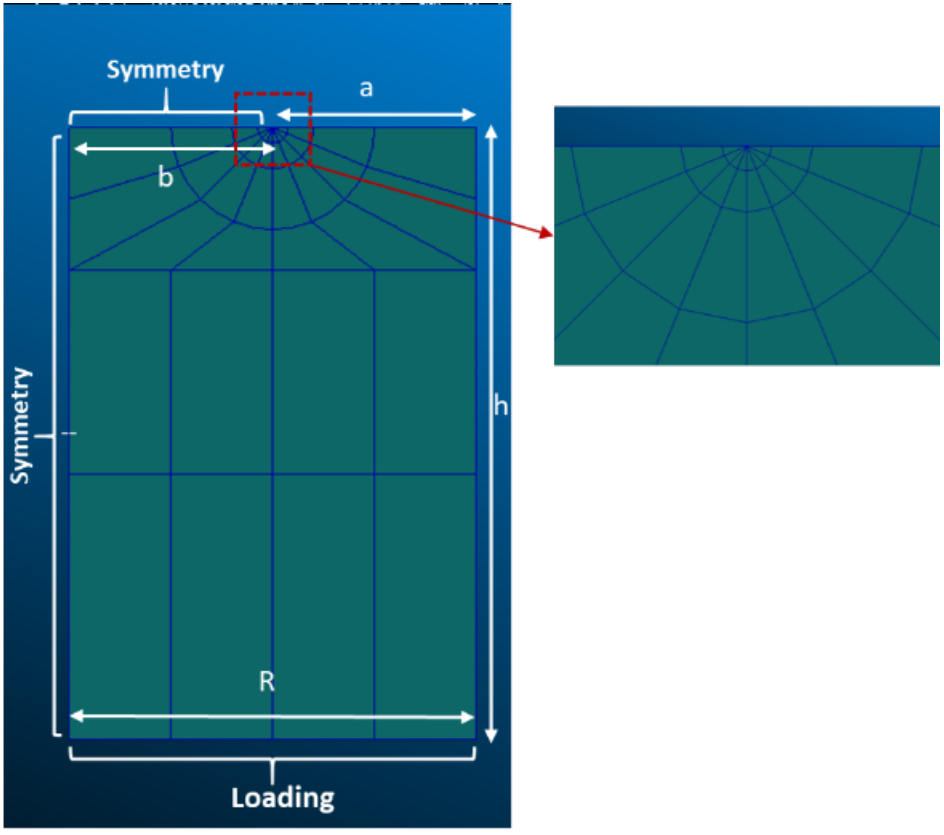


Figure 6.50-1 Cracks at a Hole in a Plate

Model File Name and Location

Model Name	Description	Location
nvcct006	Axisymmetric Crack in a Bar, Plane Strain	tpl\verifman\nafems

Results

Reference solution [1] of the normalized stress intensity factor is $\frac{K_I}{K_o} = 0.475$ where $K_o = \frac{P}{\pi b^2}(\pi b)^{0.5}$

Virtual Crack Extension Method

The details of this method are explained in NAFEMS fracture mechanics benchmarks article [1].

Virtual Crack Extension Method:

Energy release rate $(G) = 5.181$

Stress intensity factor $K_I = \left(\frac{GE}{1-\upsilon^2}\right)^{0.5} = 1085.6$

$K_o = \frac{P}{\pi b^2}(\pi b)^{0.5} = 2242.0$

$\frac{K_I}{K_o} = 0.4842$

Conclusion

The results from MSC Nastran are in a good agreement with the reference solution.

6.51 Test 6: Compact Tension Specimen

Problem Description

The following test case describe finite element analysis of compact tension specimen subjected to a point load. Normalized stress intensity factor for the test case is calculated using virtual crack extension method. The results are compared to the analytical solution [1].

Solution Number

SOL 400

Features Used

- Implicit Nonlinear

Reference

1. Test case 6 from NAFEMS Publication, Ref: R0020, 2D Test Cases in Linear Elastic Fracture Mechanics, Jan,1992.

Modeling Techniques Used

Due to symmetry in loading and boundary conditions one half of the test geometry along crack plane is used for the analysis. The finite element model is shown in Figure 6.45-1. The model is meshed with CQUAD8 and quarter point CTRIA6 elements. The quarter point elements are used at the crack tip. VCCT bulk data entry is used to obtain the energy release rate.

Material Properties

The material is linear elastic with a Young's modulus E = 207000 and a Poisson's ratio ν= 0.3

Geometrical properties

a/W=0.5; W= 50.0; t = 1.0.

Loading

Point load of 100 unit is applied on top face as shown in Figure 6.45-1.

Boundary Condition

Boundary condition as shown in Figure 6.45-1.

$u_y = 0$ on x axis

$u_x = 0$ at point A.

Illustrations of the Model

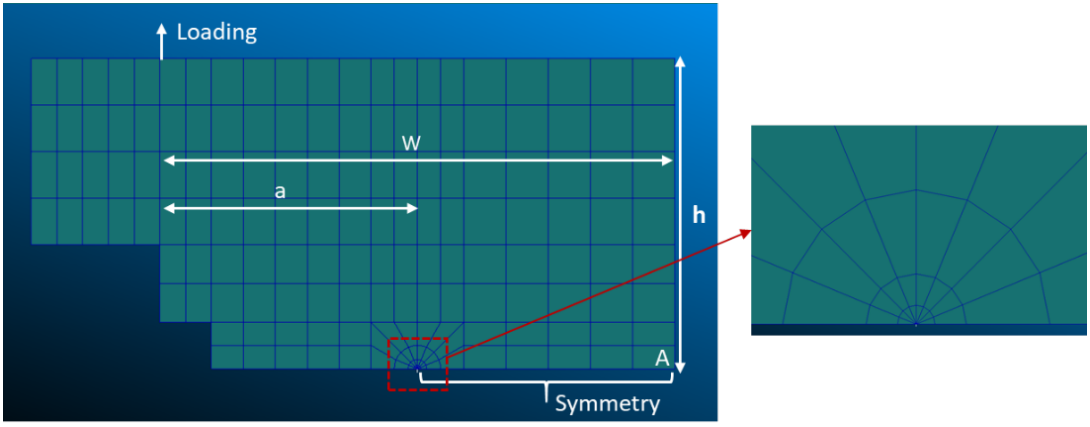


Figure 6.51-1 Compact Tension Specimen

Model File Name and Location

Model Name	Description	Location
nvcct007	Compact Tension Specimen, Plane Strain	tpl\verifman\nafems

Results

Reference solution [1] of the normalized stress intensity factor is $\frac{K_I}{K_o} = 9.659$ where $K_o = \frac{P}{tw^{0.5}}$

Virtual Crack Extension Method

The details of this method are explained in NAFEMS fracture mechanics benchmarks article [2].

Virtual Crack Extension Method:

Energy release rate $(G) = 8.224$

Stress intensity factor $K_I = \left(\frac{GE}{1-\upsilon^2}\right)^{0.5} = 1367.7$

$K_o = \frac{P}{tw^{0.5}} = 141.42$

$$\frac{K_I}{K_o} = 9.671$$

Conclusion

The results from MSC Nastran are in a good agreement with the reference solution.

6.52 Test 7: T-Joint Weld Attachment

Problem Description

The following test case describe finite element analysis of T joint subjected to uniform stress at the top edge. Normalized stress intensity factor for the test case is calculated using virtual crack extension method. The results are compared to the analytical solution [1].

Solution Number

SOL 400

Features Used

- Implicit Nonlinear

Reference

1. Test 7.1 from NAFEMS Publication, Ref: R0020, 2D Test Cases in Linear Elastic Fracture Mechanics, Jan,1992.

Modeling Techniques Used

The finite element model is shown in Figure 6.45-1. The model is meshed with CQUAD8 and quarter point CTRIA6 elements. The quarter point elements are used at the crack tip. VCCT bulk data entry is used to obtain the energy release rate.

Material Properties

The material is linear elastic with a Young's modulus, E = 207000 and Poisson's ratio, ν= 0.3

Geometrical properties

a/T=0.1; W/T= 0.5; t/T=1.0; L/T = 12.0; θ=45°; T=50.

Loading

Point load of 100 unit is applied on top face as shown in Figure 6.45-1.

Boundary Condition

Boundary condition as shown in Figure 6.45-1.

$u_y = 0$ on base

$u_x = 0$ on base and top.

Illustrations of the Model

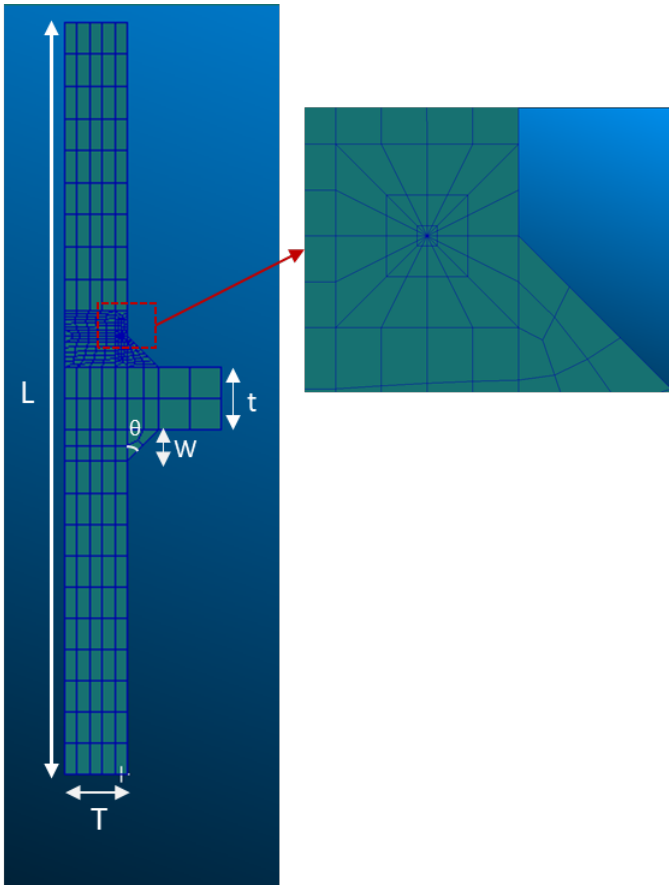


Figure 6.52-1 Compact Tension Specimen

Model File Name and Location

Model Name	Description	Location
nvcct008	Weld Attachment, T-Joint, Plane Strain	tpl\verifman\nafems

Results

Reference solution [1] of the normalized stress intensity factor is $\frac{K_I}{K_o} = 1.317$ where $K_o = \sigma(\pi b)^{0.5}$

Virtual Crack Extension Method

The details of this method are explained in NAFEMS fracture mechanics benchmarks article [1].

Virtual Crack Extension Method:

Energy release rate $(G) = 1.233$

Stress intensity factor $K_I = \left(\frac{GE}{1-\upsilon^2}\right)^{0.5} = 529.60$

$K_o = \sigma(\pi b)^{0.5} = 396.3$

$\frac{K_I}{K_o} = 1.335$

Conclusion

The results from MSC Nastran are in a good agreement with the reference solution.

6.53 Test 8: V-Notch Specimen in Tension

Problem Description

The following test case describe finite element analysis of V-Notch cracked plate subjected to uniform stress at the top edge. Normalized stress intensity factor for the test case is calculated using virtual crack extension method. The results are compared to the analytical solution [1].

Solution Number

SOL 400

Features Used

- Implicit Nonlinear

Reference

1. Test case 8.1 from NAFEMS Publication, Ref: R0020, 2D Test Cases in Linear Elastic Fracture Mechanics, Jan,1992.

Modeling Techniques Used

Due to symmetry in loading and boundary conditions, one-quarter of test geometry along the plate axis is used for the analysis. The finite element model is shown in Figure 6.45-1. The model is meshed with CQUAD8 and quarter point CTRIA6 elements. The quarter point elements are used at the crack tip. VCCT bulk data entry is used to obtain the energy release rate.

Material Properties

The material is linear elastic with a Young's modulus, E = 207000 and Poisson's ratio, ν= 0.3

Geometrical properties

a/d=0.2; d/W= 0.1; h/W=1.0; β=45°; W=250.

Loading

Uniform stress of 100 unit is applied on top face as shown in Figure 6.45-1.

Boundary Condition

Boundary condition as shown in Figure 6.45-1.

$u_y = 0$ on X axis.

$u_x = 0$ on Y axis.

Illustrations of the Model

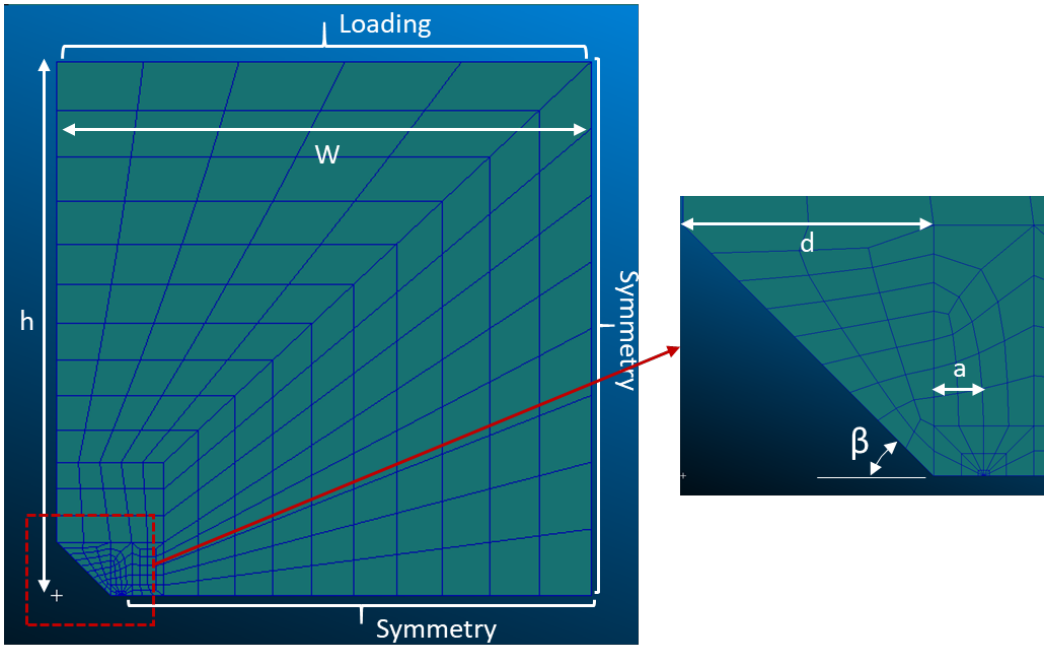


Figure 6.53-1 V-Notch Cracked Plate

Model File Name and Location

Model Name	Description	Location
nvcct009	V-Notch Cracked Plate (Plane Strain)	tpl\verifman\nafems

Results

Reference solution [1] of the normalized stress intensity factor is $\frac{K_I}{K_o} = 2.740$ where $K_o = \sigma(\pi a)^{0.5}$

Virtual Crack Extension Method

The details of this method are explained in NAFEMS fracture mechanics benchmarks article [1].

Virtual Crack Extension Method:

Energy release rate (G)= 5.461

Stress intensity factor $K_I = \left(\frac{GE}{1-\upsilon^2}\right)^{0.5} = 1114.6$

$K_o = \sigma(\pi a)^{0.5} = 396.3$

$\frac{K_I}{K_o} = 2.81$

Conclusion

The results from MSC Nastran are in a good agreement with the reference solution.

Creep

6.54 Test 1A: 2-D Plane Stress - Uniaxial Load, Secondary Creep

Problem Description

This NAFEMS creep test involves secondary creep of a two dimensional thin plate subjected to uniaxial load.

Solution Number

SOL 400

Features Used

- Secondary Creep Analysis with Plane Stress Elements

Reference

1. Test 1A from NAFEMS Publication Ref: R0027, NAFEMS Fundamental Tests of Creep Behavior, June 1993.

Modeling Techniques Used

A thin square plate subjected to secondary creep under uniaxial tensile load is modeled using CQUAD8 plane stress elements. The model details are given below:

Material Properties

Young's Modulus E = 200×10^3 N/mm^2 Poisson's ratio $\nu = 0.3$

Creep Law: $\varepsilon = A\,\sigma^n\,t$ $A = 3.125 \times 10^{-14}$ per hour (σ in N/mm^2) $n = 5$

Loading

Prescribed tensile stress σ_1 = 200 N/mm^2 on line BC.

Boundary Condition

$U_x = 0.0$ at line AD

$U_y = 0.0$ on midpoint of line AD.

Illustrations of the Model

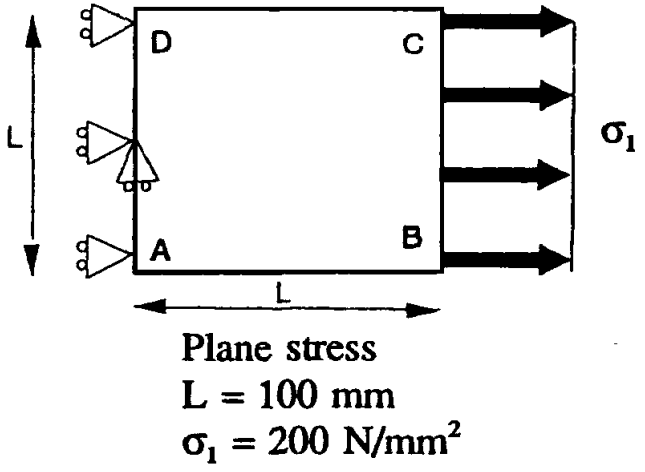


Figure 6.54-1 Plane Stress Model under Uniaxial Load and Secondary Creep

Model File Name and Location

Model Name	Description	Location
nctest1a.dat	Test1A: Plane stress - Uniaxial Load and Secondary Creep	tpl\verifman

Results

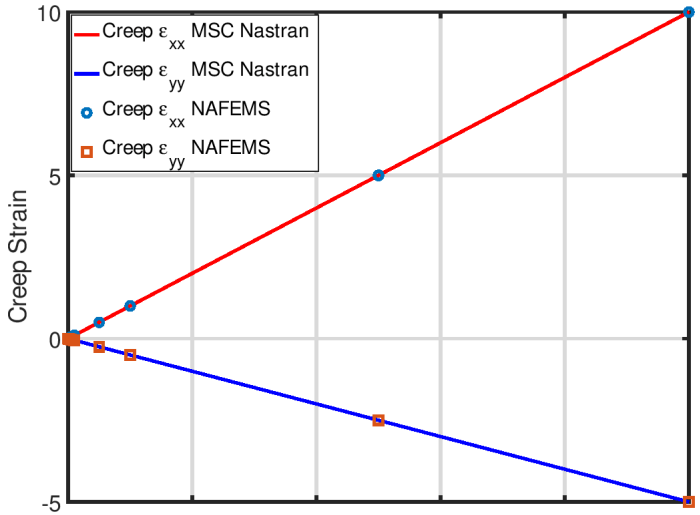


Figure 6.54-2 Creep Strain versus Time Plot

Conclusion

The reference solutions from NAFEMS are defined as and $\varepsilon_{YY}^{C} = \varepsilon_{ZZ}^{C} = (-0.005)t$.

$$\varepsilon_{XX}^{C} = \varepsilon_{eff}^{C} = 0.01t$$

The X and Y components of creep strain values from MSC Nastran are compared to these reference results from NAFEMS in Figure 6.54-2 and also in the following table:

Time	ε_{XX}^{C}			ε_{YY}^{C}		
	NAFEMS	MSC Nastran	% Error	NAFEMS	MSC Nastran	% Error
0.1	0.001	0.001	0.0%	-0.0005	-0.0005	0.0%
1.0	0.01	0.01	0.0%	-0.005	-0.005	0.0%
5.0	0.05	0.05	0.0%	-0.025	-0.025	0.0%
10.0	0.1	0.1	0.0%	-0.05	-0.05	0.0%
50.0	0.5	0.5	0.0%	-0.25	-0.25	0.0%
100.0	1.0	1.0	0.0%	-0.50	-0.50	0.0%
500.0	5.0	5.0	0.0%	-2.5	-2.5	0.0%
1000.0	10.0	10.0	0.0%	-5.0	-5.0	0.0%

6.55 Test 1B: 2-D Plane Stress - Uniaxial Displacement, Secondary Creep

Problem Description

This NAFEMS creep test involves secondary creep of a two dimensional thin plate subjected to uniaxial displacement.

Solution Number

SOL 400

Features Used

- Secondary Creep Analysis with Plane Stress Elements

Reference

1. Test 1B from NAFEMS Publication Ref: R0027, NAFEMS Fundamental Tests of Creep Behavior, June 1993.

Modeling Techniques Used

A thin square plate subjected to secondary creep under uniaxial displacement load is modeled using CQUAD8 plane stress elements. The model details are given below:

Material Properties

Young's Modulus $E = 200 \times 10^3$ N/mm^2 Poisson's ratio $\nu = 0.3$

Creep Law: $\varepsilon = A\,\sigma^n\,t$ $A = 3.125 \times 10^{-14}$ per hour (σ in N/mm^2) $n = 5$

Loading

None.

Boundary Condition

$U_x = 0.0$ at line AD

U_y = 0.0 on midpoint of line AD

Uniaxial displacement u_1 = 0.1 mm on line BC

Illustrations of the Model

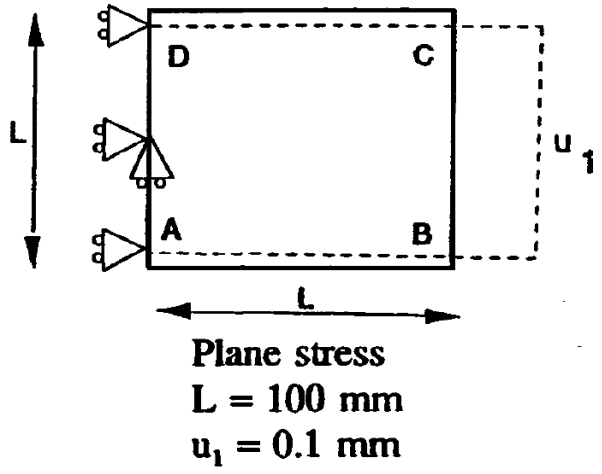


Figure 6.55-1 Plane Stress Model under Uniaxial Displacement and Secondary Creep

Model File Name and Location

Model Name	Description	Location
nctest1b.dat	Test1B: Plane stress - Uniaxial Displacement and Secondary Creep	tpl\verifman

Results

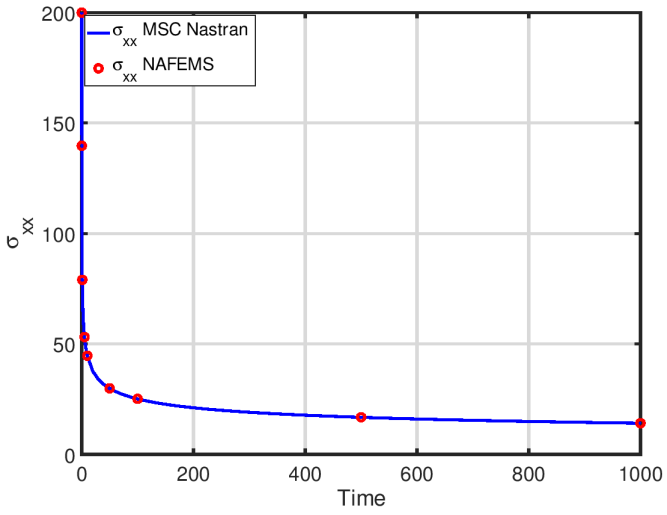


Figure 6.55-2 Stress versus Time Plot

Conclusion

The X component of stress values from MSC Nastran are compared to the reference results from NAFEMS in Figure 6.54-2 and also in the following table.

Time	SIGMA - XX (Time Hardening)		
	NAFEMS	MSC Nastran	% Error
0.0	200.00	200.00	0.00%
0.1	139.71	133.70	-4.30%
1.0	79.04	79.00	-0.05%
5.0	53.12	53.07	-0.09%
10.0	44.69	44.65	-0.09%
50.0	29.91	29.86	-0.16%

Time	SIGMA - XX (Time Hardening)		
	NAFEMS	MSC Nastran	% Error
100.0	25.15	25.11	-0.15%
500.0	16.82	16.78	-0.24%
1000.0	14.14	14.11	-0.21%

6.56 Test 2A: 2-D Plane Stress - Biaxial Load, Secondary Creep

Problem Description

This NAFEMS creep test involves secondary creep of a two dimensional thin plate subjected to biaxial load.

Solution Number

SOL 400

Features Used

- Secondary Creep Analysis with Plane Stress Elements

Reference

1. Test 2A from NAFEMS Publication Ref: R0027, NAFEMS Fundamental Tests of Creep Behavior, June 1993.

Modeling Techniques Used

A thin square plate subjected to secondary creep under biaxial tensile load is modeled using CQUAD8 plane stress elements. The model details are given below:

Material Properties

Young's Modulus $E = 200 \times 10^3$ N/mm^2 Poisson's ratio $\nu = 0.3$

Creep Law: $\varepsilon = A\,\sigma^n\,t$ $A = 3.125 \times 10^{-14}$ per hour (σ in N/mm^2) $n = 5$

Loading

Prescribed tensile stress $\sigma_1 = 200$ N/mm^2 on line BC and $\sigma_2 = 200$ N/mm^2 on line CD

Boundary Condition

$U_x = 0.0$ at line AD

$U_y = 0.0$ on midpoint of line AB

Illustrations of the Model

Plane stress
L = 100 mm
$\sigma_1 = \sigma_2 = 200$ N/mm^2

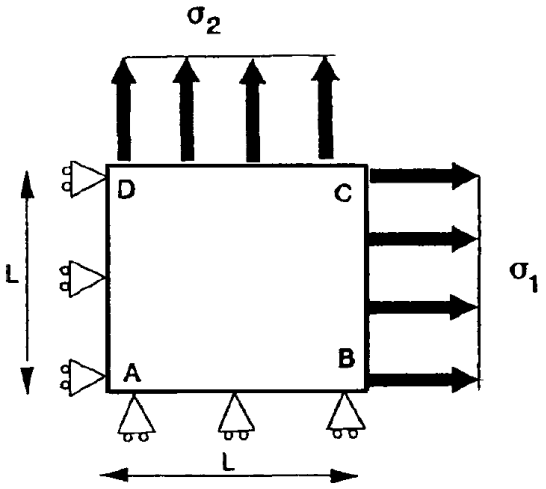


Figure 6.56-1 Plane Stress Model under Biaxial Load and Secondary Creep

Model File Name and Location

Model Name	Description	Location
nctest2a.dat	Test2A: Plane stress - Biaxial Load and Secondary Creep	tpl\verifman

Results

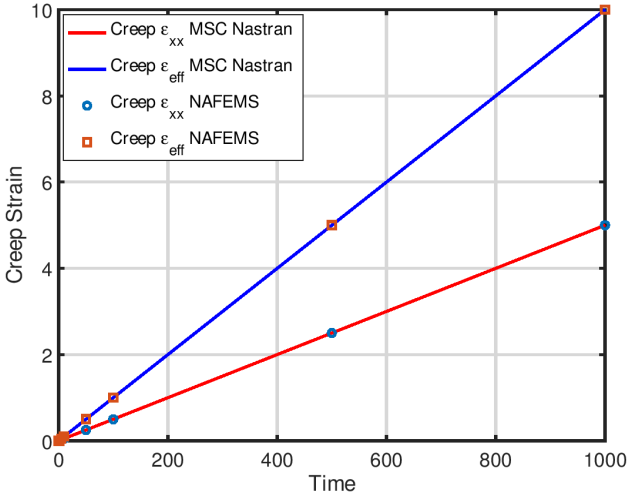


Figure 6.56-2 Creep Strain versus Time Plot

Conclusion

The reference solutions from NAFEMS are defined as $\varepsilon_{XX}^{C} = \varepsilon_{eff}^{C} = 0.005t$ and $\varepsilon_{eff}^{C} = 0.01t$.

The X and Y components of creep strain values from MSC Nastran are compared to these reference results from NAFEMS in Figure 6.54-2 and also in the following table:

Time	ε_{XX}^{C}			ε_{eff}^{C}		
	NAFEMS	MSC Nastran	% Error	NAFEMS	MSC Nastran	% Error
0.1	0.0005	0.0005	0.0%	0.001	0.001	0.0%
1.0	0.005	0.005	0.0%	0.01	0.01	0.0%
5.0	0.025	0.025	0.0%	0.05	0.05	0.0%
10.0	0.05	0.05	0.0%	0.10	0.10	0.0%
50.0	0.25	0.25	0.0%	0.50	0.50	0.0%
100.0	0.50	0.50	0.0%	1.00	1.00	0.0%
500.0	2.50	2.50	0.0%	5.00	5.00	0.0%
1000.0	5.00	5.00	0.0%	10.00	10.00	0.0%

6.57 Test 2B: 2-D Plane Stress - Biaxial Displacement, Secondary Creep

Problem Description

This NAFEMS creep test involves secondary creep of a two dimensional thin plate subjected to biaxial displacement.

Solution Number

SOL 400

Features Used

- Secondary Creep Analysis with Plane Stress Elements

Reference

1. Test 2B from NAFEMS Publication Ref: R0027, NAFEMS Fundamental Tests of Creep Behavior, June 1993.

Modeling Techniques Used

A thin square plate subjected to secondary creep under biaxial displacement load is modeled using CQUAD8 plane stress elements. The model details are given below:

Material Properties

Young's Modulus $E = 200 \times 10^3$ N/mm^2 Poisson's ratio $\nu = 0.3$

Creep Law: $\varepsilon = A\,\sigma^n\,t$ $A = 3.125 \times 10^{-14}$ per hour (σ in N/mm^2) $n = 5$

Loading

None.

Boundary Condition

$U_x = 0.0$ at line AD

$U_y = 0.0$ on midpoint of line AD

Main Index

Biaxial displacement u_1 = 0.1 mm on line BC and u_2 = 0.1 mm on line CD

Illustrations of the Model

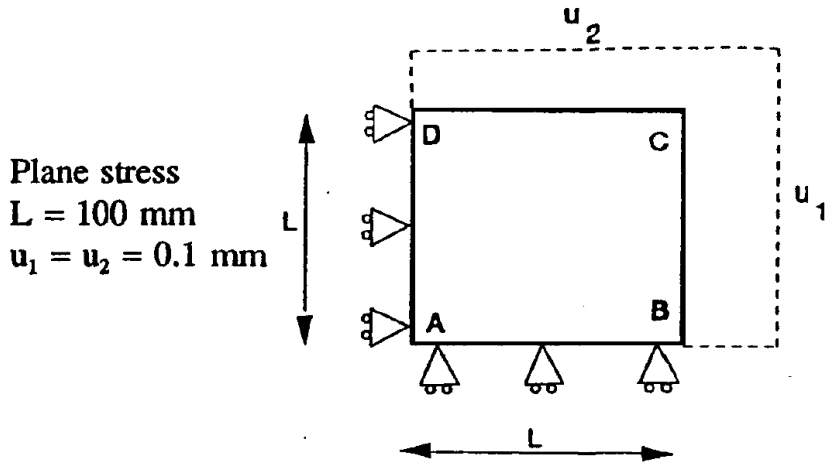


Figure 6.57-1 Plane Stress Model under Biaxial Displacement and Secondary Creep

Model File Name and Location

Model Name	Description	Location
nctest2b.dat	Test2B: Plane stress - Biaxial Displacement and Secondary Creep	tpl\verifman

Results

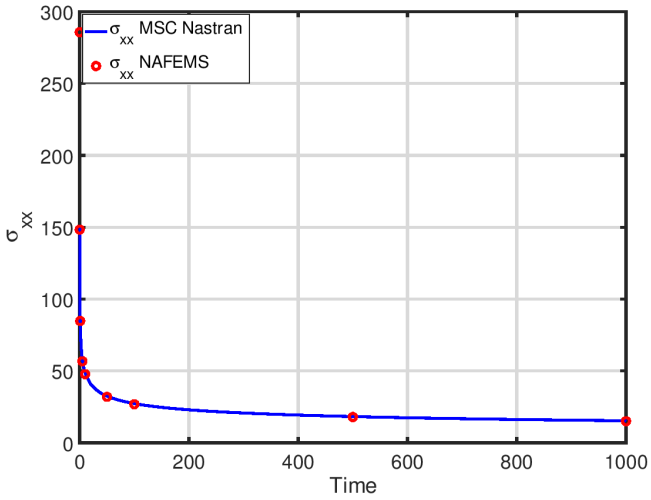


Figure 6.57-2 Stress versus Time Plot

Conclusion

The X component of stress values from MSC Nastran are compared to the reference results from NAFEMS in Figure 6.54-2 and also in the following table.

Time	SIGMA - XX (Time Hardening)		
	NAFEMS	MSC Nastran	% Error
0.0	285.71	285.71	0.00%
0.1	148.35	150.72	1.60%
1.0	84.77	86.28	1.78%
5.0	56.76	57.78	1.80%
10.0	47.79	48.59	1.69%
50.0	31.96	32.49	1.66%

Time	SIGMA - XX (Time Hardening)		
	NAFEMS	MSC Nastran	% Error
100.0	26.86	27.31	1.67%
500.0	17.99	18.25	1.44%
1000.0	15.07	15.35	1.86%

6.58 Test 3B: 2-D Plane Stress - Biaxial (-ve) Displacement, Secondary Creep

Problem Description

This NAFEMS creep test involves secondary creep of a two dimensional thin plate subjected to biaxial displacement.

Solution Number

SOL 400

Features Used

- Secondary Creep Analysis with Plane Stress Elements

Reference

1. Test 3B from NAFEMS Publication Ref: R0027, NAFEMS Fundamental Tests of Creep Behavior, June 1993.

Modeling Techniques Used

A thin square plate subjected to secondary creep under biaxial displacement load is modeled using CQUAD8 plane stress elements. The model details are given below:

Material Properties

Young's Modulus E = 200×10^3 N/mm^2 Poisson's ratio $\nu = 0.3$

Creep Law: $\varepsilon = A \sigma^n t$ $A = 3.125 \times 10^{-14}$ per hour (σ in N/mm^2) $n = 5$

Loading

None.

Boundary Condition

$U_x = 0.0$ at line AD

$U_y = 0.0$ on midpoint of line AB

Main Index

Biaxial displacement u_1 = 0.1 mm on line BC and u_2 = -0.1 mm on line CD

Illustrations of the Model

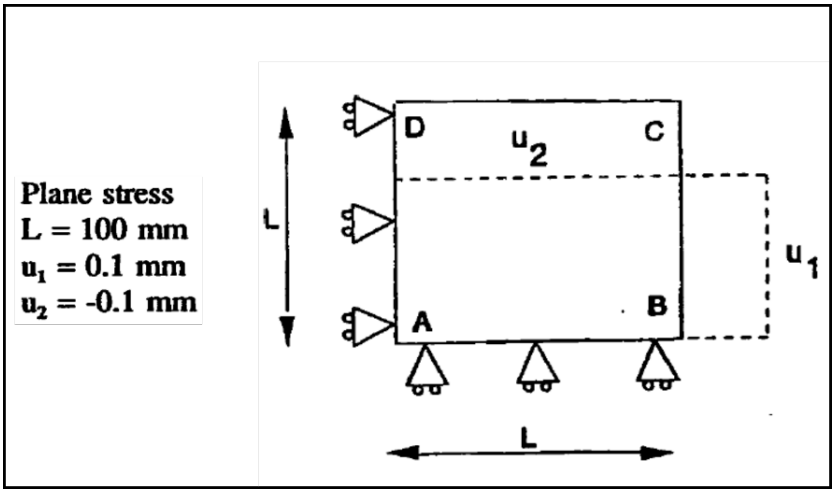


Figure 6.58-1 Plane Stress Model under Biaxial (Negative) Displacement and Secondary Creep

Model File Name and Location

Model Name	Description	Location
nctest3b.dat	Test3B: Plane stress - Biaxial (Negative) Displacement and Secondary Creep	tpl\verifman

Results

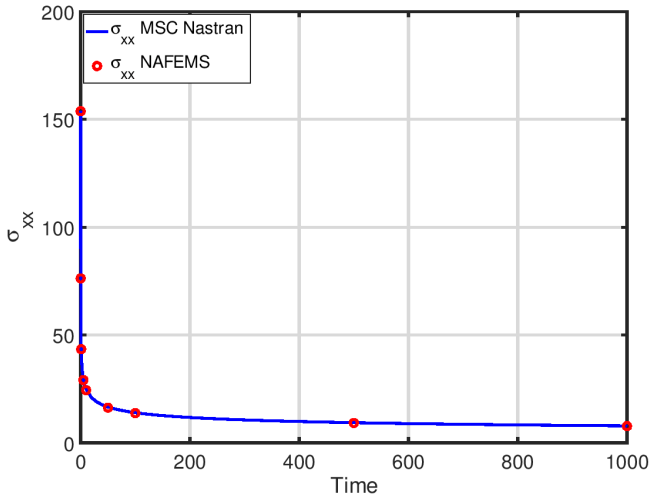


Figure 6.58-2 Stress versus Time Plot

Conclusion

The X component of stress values from MSC Nastran are compared to the reference results from NAFEMS in Figure 6.54-2 and also in the following table.

Time	σ_{XX}		
	NAFEMS	MSC Nastran	% Error
0.0	153.85	153.85	0.00%
0.1	76.29	77.46	1.53%
1.0	43.42	44.20	1.80%
5.0	29.11	29.59	1.65%
10.0	24.45	24.88	1.76%
50.0	16.33	16.64	1.90%

Time	σ_{XX}		
	NAFEMS	MSC Nastran	% Error
100.0	13.78	13.99	1.52%
500.0	9.20	9.34	1.52%
1000.0	7.73	7.86	1.68%

6.59 Test 8A: 2D Plane Stress - Uniaxial Load, Primary Creep

Problem Description

This NAFEMS creep test involves primary creep of a two dimensional thin plate subjected to uniaxial load.

Solution Number

SOL 400

Features Used

- Primary Creep Analysis with Plane Stress Elements

Reference

1. Test 8A from NAFEMS Publication Ref: R0027, NAFEMS Fundamental Tests of Creep Behavior, June 1993.

Modeling Techniques Used

A thin square plate subjected to primary creep under uniaxial tensile load is modeled using CQUAD8 plane stress elements. The model details are given below:

Material Properties

Young's Modulus $E = 200 \times 10^3$ N/mm^2 Poisson's ratio $\nu = 0.3$

Creep Law: $\varepsilon = A\,\sigma^n\,t^m$ $A = 3.125 \times 10^{-14}$ per hour (σ in N/mm^2) $n = 5$ $m = 0.5$

Loading

Prescribed tensile stress $\sigma_1 = 200$ N/mm^2 on line BC.

Boundary Condition

$U_x = 0.0$ at line AD

$U_y = 0.0$ on midpoint of line AD.

Illustrations of the Model

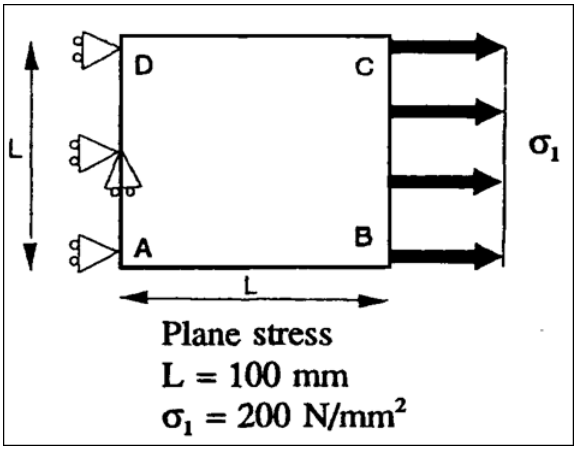


Figure 6.59-1 Plane Stress Model under Uniaxial Load and Primary Creep

Model File Name and Location

Model Name	Description	Location
`lefm008a.dat`	Weld Attachment, T-Joint, Plane Strain	tpl\verifman

Results

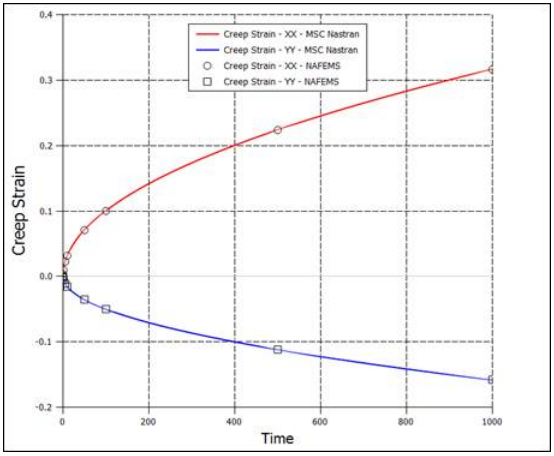


Figure 6.59-2 Creep Strain versus Time Plot

Conclusion

The reference solutions from NAFEMS are defined as $\varepsilon_{XX}^{C} = \varepsilon_{eff}^{C} = 0.01t^{0.5}$ and $\varepsilon_{YY}^{C} = \varepsilon_{ZZ}^{C} = (-0.005)t^{0.5}$.

The X & Y components of creep strain values from MSC Nastran are compared to these reference results from NAFEMS in Figure 6.54-2 and also in the following table:

Time	ε_{XX}^{C}			ε_{YY}^{C}		
	NAFEMS	MSC Nastran	% Error	NAFEMS	MSC Nastran	% Error
0.1	0.00316	0.003278	3.74%	-0.00158	-0.001639	3.74%
1.0	0.01	0.010141	1.41%	-0.005	-0.005071	1.42%
5.0	0.0224	0.022530	0.58%	-0.0112	-0.011265	0.58%
10.0	0.0316	0.031806	0.65%	-0.0158	-0.015903	0.65%
50.0	0.0707	0.070914	0.30%	-0.0354	-0.035458	0.16%
100.0	0.1	0.100224	0.22%	-0.05	-0.050112	0.22%
500.0	0.2236	0.223871	0.12%	-0.112	-0.111935	-0.06%
1000.0	0.3162	0.316511	0.10%	-0.158	-0.158251	0.16%

6.60 Test 8B: 2-D Plane Stress - Uniaxial Displacement, Primary Creep

Problem Description

This NAFEMS creep test involve primary creep of a two dimensional thin plate subjected to uniaxial displacement.

Solution Number

SOL 400

Features Used

- Primary Creep Analysis with Plane Stress Elements

Reference

1. Test 8B from NAFEMS Publication Ref: R0027, NAFEMS Fundamental Tests of Creep Behavior, June 1993.

Modeling Techniques Used

A thin square plate subjected to primary creep under uniaxial displacement load is modeled using CQUAD8 plane stress elements. The model details are given below:

Material Properties

Young's Modulus E = 200×10^3 N/mm^2 Poisson's ratio $\nu = 0.3$

Creep Law: $\varepsilon = A\,\sigma^n\,t^m$ $A = 3.125 \times 10^{-14}$ per hour (σ in N/mm^2) $n = 5$ $m = 0.5$

Loading

None.

Boundary Condition

$U_x = 0.0$ at line AD and $U_y = 0.0$ on midpoint of line AD

Uniaxial displacement $u_1 = 0.1$ mm on line BC.

Illustrations of the Model

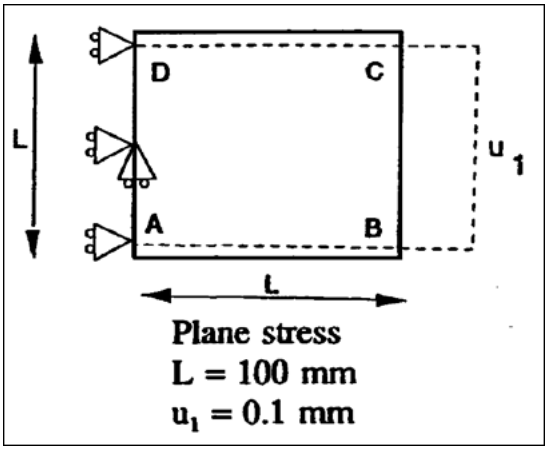


Figure 6.60-1 Plane Stress Model under Uniaxial Displacement and Primary Creep

Model File Name and Location

Model Name	Description	Location
`nctest8b.dat`	Test8B: Plane Stress - Uniaxial Displacement and Primary Creep	tpl\verifman

Results

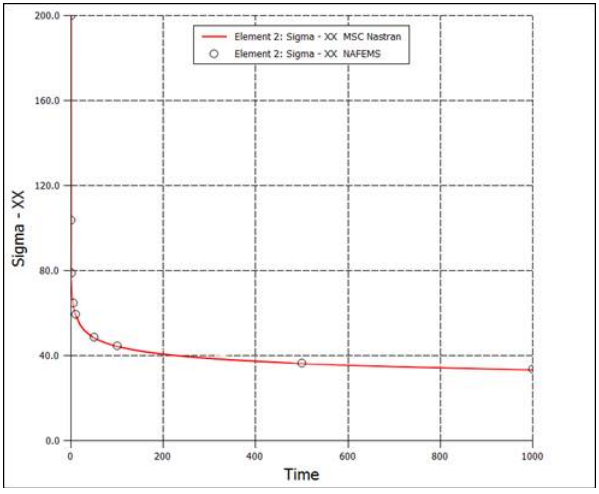


Figure 6.60-2 Creep Strain versus Time Plot

Conclusion

The X component of stress values from MSC Nastran are compared to the reference results from NAFEMS in Figure 6.54-2 and also in the following table:

Time	SIGMA - XX (Time Hardening)		
	NAFEMS	MSC Nastran	% Error
0.0	200	200.000000	0.00%
0.1	103.74	103.592590	-0.14%
1.0	78.9	78.619415	-0.36%
5.0	64.76	64.521416	-0.37%
10.0	59.45	59.268791	-0.30%
50.0	48.65	48.407439	-0.50%
100.0	44.6	44.357185	-0.54%
500.0	36.48	36.316412	-0.45%
1000.0	33.69	33.351093	-1.01%

6.61 Test 8C: 2D Plane Stress - Stepped Load Primary Creep

Problem Description

This NAFEMS creep test involve primary creep of a two dimensional thin plate subjected to stepped load.

Solution Number

SOL 400

Features Used

- Primary Creep Analysis with Plane Stress Elements

Reference

1. Test 8C from NAFEMS Publication Ref: R0027, NAFEMS Fundamental Tests of Creep Behavior, June 1993.

Modeling Techniques Used

A thin square plate subjected to primary creep under stepped load is modeled using CQUAD8 plane stress elements. The model details are given below:

Material Properties

Young's Modulus E = 200×10^3 N/mm^2 Poisson's ratio $\nu = 0.3$

Creep Law: $\varepsilon = A \sigma^n t^m$ $A = 3.125 \times 10^{-14}$ per hour (σ in N/mm^2) $n = 5$ $m = 0.5$

Loading

Prescribed tensile stress σ_1 on line BC

$\sigma_1 = 200$ N/mm^2 for t = 0 to 100 hoursand $\sigma_1 = 250$ N/mm^2 for t > 100 hours

Boundary Condition

$U_x = 0.0$ at line AD

Main Index

U_y = 0.0 on midpoint of line AD

Illustrations of the Model

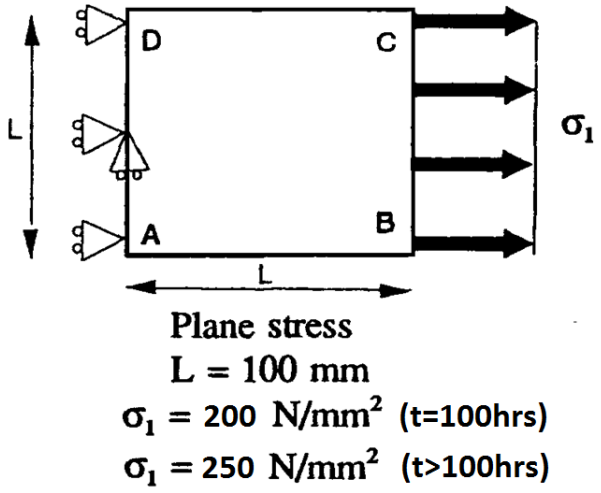


Figure 6.61-1 Plane Stress Model under Stepped Load and Primary Creep

Model File Name and Location

Model Name	Description	Location
nctest8c.dat	Test8C: Plane Stress - Stepped Load and Primary Creep	tpl\verifman

Results

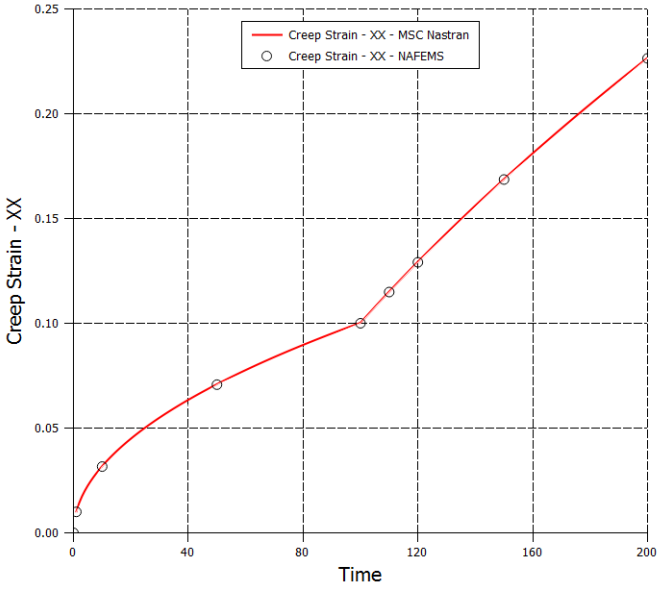


Figure 6.61-2 Creep Strain - XX versus Time Plot

Conclusion

The X component of creep strain values from MSC Nastran are compared to the reference results from NAFEMS in Figure 6.54-2 and also in the following table:

Time	Creep Strain - XX (Time Hardening)		
	NAFEMS	MSC Nastran	% Error
1.0	0.01	0.010141	1.41%
10.0	0.03162	0.031809	0.60%
50.0	0.07071	0.070935	0.32%
100.0	0.1	0.100238	0.24%
110.0	0.1149	0.115134	0.20%
120.0	0.1291	0.129365	0.21%
150.0	0.1686	0.168827	0.13%
200.0	0.2264	0.226660	0.11%

6.62 Test 9A: 2-D Plane Stress - Biaxial Load, Primary Creep

Problem Description

This NAFEMS creep test involve primary creep of a two dimensional thin plate subjected to biaxial load.

Solution Number

SOL 400

Features Used

- Primary Creep Analysis with Plane Stress Elements

Reference

1. Test 9A from NAFEMS Publication Ref: R0027, NAFEMS Fundamental Tests of Creep Behavior, June 1993.

Modeling Techniques Used

A thin square plate subjected to primary creep under biaxial tensile load is modeled using CQUAD8 plane stress elements. The model details are given below:

Material Properties

Young's Modulus $E = 200 \times 10^3$ N/mm^2 Poisson's ratio $\nu = 0.3$

Creep Law: $\varepsilon = A\,\sigma^n\,t^m$ $A = 3.125 \times 10^{-14}$ per hour (σ in N/mm^2) $n = 5$ $m = 0.5$

Loading

Prescribed tensile stress $\sigma_1 = 200$ N/mm^2 on line BC and $\sigma_2 = 200$ N/mm^2 on line CD.

Boundary Condition

$U_x = 0.0$ at line AD

$U_y = 0.0$ at line AB

Illustrations of the Model

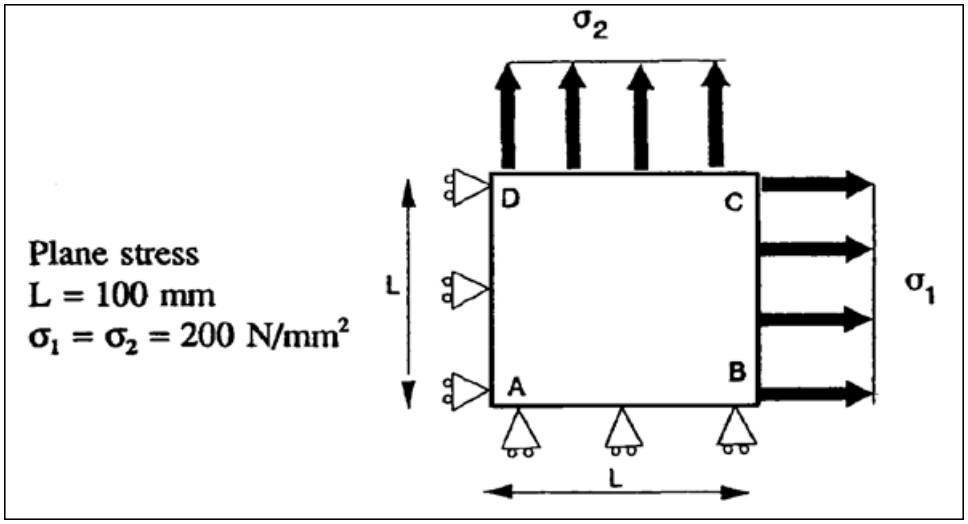


Figure 6.62-1 Plane Stress Model under Stepped Load and Primary Creep

Model File Name and Location

Model Name	Description	Location
nctest9a.dat	Test9A: Plane Stress - Biaxial Load and Primary Creep	tpl\verifman

Results

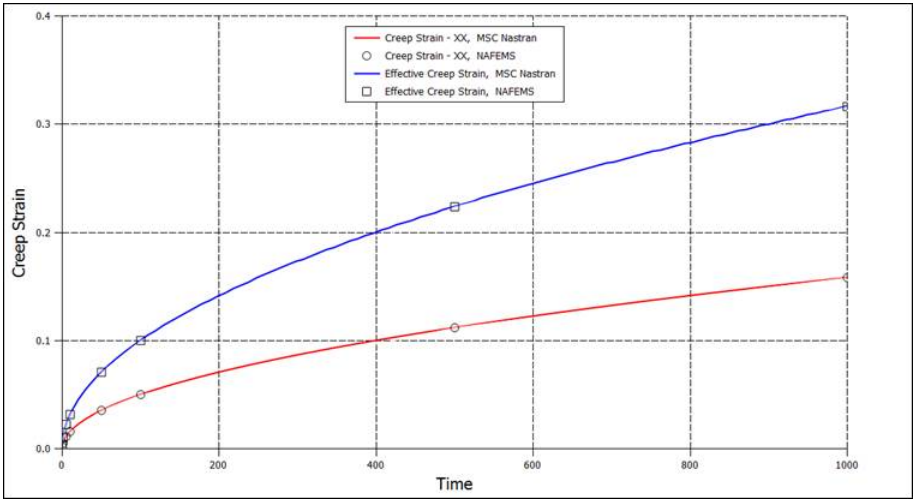


Figure 6.62-2 Creep Strain - XX versus Time Plot

Conclusion

The reference solutions from NAFEMS are defined as $\varepsilon_{XX}^{C} = \varepsilon_{YY}^{C} = 0.005t^{0.5}$ and $\varepsilon_{eff}^{C} = 0.01t^{0.5}$.

The X component and effective creep strain values from MSC Nastran are compared to these reference results from NAFEMS in Figure 6.54-2 and also in the following table:

Time	ε_{XX}^{C}			ε_{eff}^{C}		
	NAFEMS	MSC Nastran	% Error	NAFEMS	MSC Nastran	% Error
0.1	0.00158	0.001642	3.93%	0.00316	0.003280	3.80%
1.0	0.005	0.005076	1.53%	0.01	0.010200	2.00%
5.0	0.01112	0.011273	1.38%	0.0224	0.022500	0.45%
10.0	0.0158	0.015912	0.71%	0.0316	0.031800	0.63%
50.0	0.0354	0.035467	0.19%	0.0707	0.071325	0.88%
100.0	0.05	0.050121	0.24%	0.1	0.100273	0.27%
500.0	0.1118	0.111942	0.13%	0.2236	0.223990	0.17%
1000.0	0.1581	0.158268	0.11%	0.3162	0.317000	0.25%

6.63 Test 9B: 2-D Plane Stress - Biaxial Displacement, Primary Creep

Problem Description

This NAFEMS creep test involve primary creep of a two dimensional thin plate subjected to biaxial displacement.

Solution Number

SOL 400

Features Used

- Primary Creep Analysis with Plane Stress Elements

Reference

1. Test 9B from NAFEMS Publication Ref: R0027, NAFEMS Fundamental Tests of Creep Behavior, June 1993.

Modeling Techniques Used

A thin square plate subjected to primary creep under biaxial displacement load is modeled using CQUAD8 plane stress elements. The model details are given below:

Material Properties

Young's Modulus $E = 200 \times 10^3$ N/mm^2 Poisson's ratio $\nu = 0.3$

Creep Law: $\varepsilon = A\,\sigma^n\,t^m$ $A = 3.125 \times 10^{-14}$ per hour (σ in N/mm^2) $n = 5$ $m = 0.5$

Loading

None.

Boundary Condition

$U_x = 0.0$ at line AD and $U_y = 0.0$ at line AB

Biaxial displacement: $u_1 = 0.1$ mm on line BC and $u_2 = 0.1$ mm on line CD.

Main Index

Illustrations of the Model

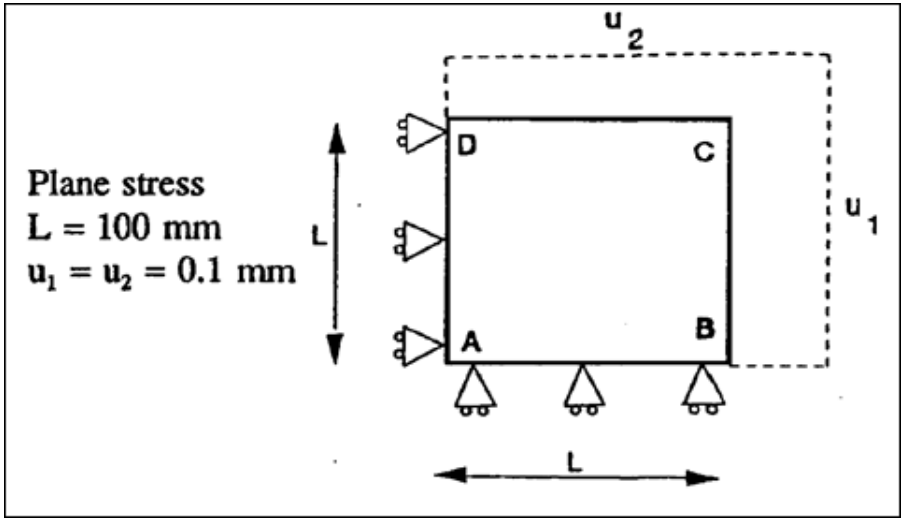


Figure 6.63-1 Plane Stress Model under Biaxial Displacement and Primary Creep

Model File Name and Location

Model Name	Description	Location
nctest9b.dat	Test9B: Plane Stress - Biaxial Displacement and Primary Creep	tpl\verifman

Results

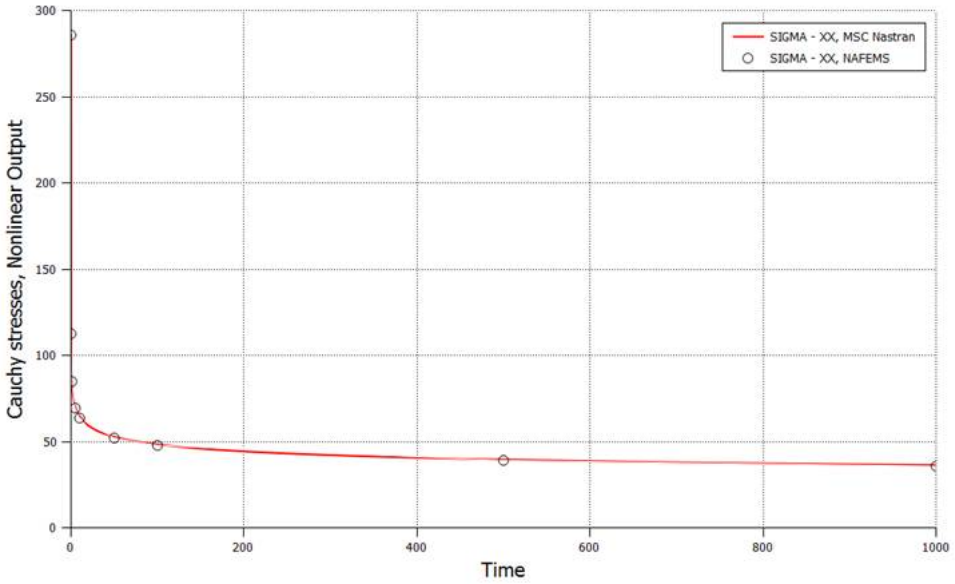


Figure 6.63-2 Stress vs Time Plot

Conclusion

The X component of stress values from MSC Nastran are compared to the reference results from NAFEMS in Figure 6.63-2 and also in the following table:

Time	SIGMA - XX (Time Hardening)		
	NAFEMS	MSC Nastran	% Error
0.0	285.71	285.714290	0.00%
0.1	112.52	114.182870	1.48%
1.0	84.78	85.921959	1.35%
5.0	69.37	70.340996	1.40%
10.0	63.59	64.572617	1.55%
50.0	52.02	52.715802	1.34%
100.0	47.74	48.303667	1.18%
500.0	39.04	39.523781	1.24%
1000.0	35.71	36.291130	1.63%

6.64 Test 9C: 2-D Plane Stress - Biaxial Stepped Load, Primary Creep

Problem Description

This NAFEMS creep test involve primary creep of a two dimensional thin plate subjected to biaxial stepped load.

Solution Number

SOL 400

Features Used

- Primary Creep Analysis with Plane Stress Elements

Reference

1. Test 9C from NAFEMS Publication Ref: R0027, NAFEMS Fundamental Tests of Creep Behavior, June 1993.

Modeling Techniques Used

A thin square plate subjected to primary creep under biaxial stepped load is modeled using CQUAD8 plane stress elements. The model details are given below:

Material Properties

Young's Modulus $E = 200 \times 10^3$ N/mm^2 Poisson's ratio $\nu = 0.3$

Creep Law: $\varepsilon = A\,\sigma^n\,t^m$ $A = 3.125 \times 10^{-14}$ per hour (σ in N/mm^2) $n = 5$ $m = 0.5$

Loading

Prescribed tensile stress σ_1 on line BC and σ_2 on line CD.

$\sigma_1 = \sigma_1 = 200$ N/mm^2 for t = 0 to 100 hours, and

$\sigma_1 = \sigma_1 = 250$ N/mm^2 for t > 100 hours.

Boundary Condition

$U_x = 0.0$ at line AD and $U_y = 0.0$ at line AB

Illustrations of the Model

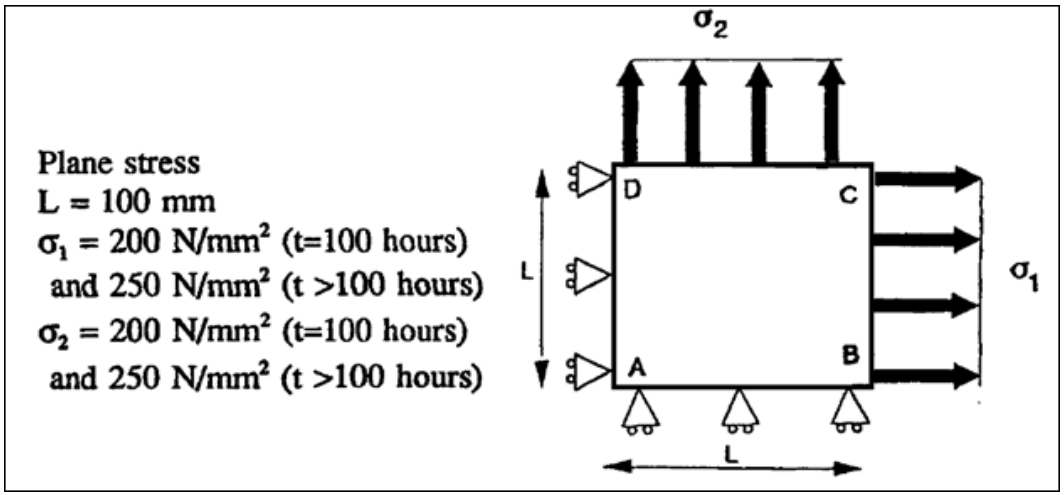


Figure 6.64-1 Plane Stress Model under Biaxial Stepped Load and Primary Creep

Model File Name and Location

Model Name	Description	Location
nctest9c.dat	Test9C: Plane Stress - Biaxial Stepped Load and Primary Creep	tpl\verifman

Results

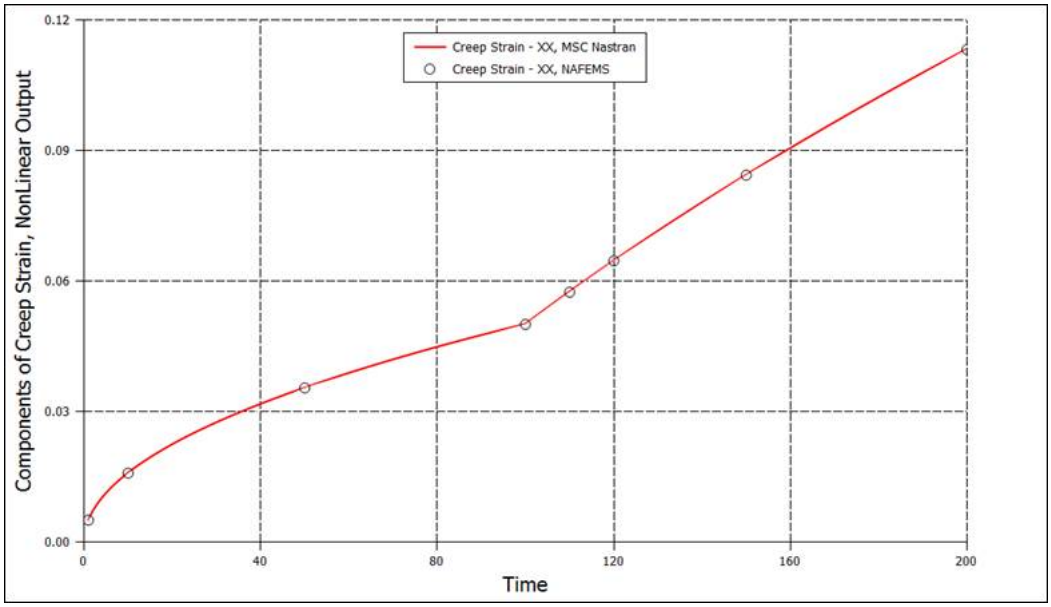


Figure 6.64-2 Creep Strain - XX versus Time Plot

Conclusion

The X component of creep strain values from MSC Nastran are compared to the reference results from NAFEMS in Figure 6.63-2 and also in the following table:

Time	Creep Strain - XX (Time Hardening)		
	NAFEMS	MSC Nastran	% Error
1.0	0.005	0.005077	1.53%
10.0	0.0158	0.015912	0.71%
50.0	0.0354	0.035473	0.21%
100.0	0.05	0.050126	0.25%
110.0	0.0574	0.057577	0.31%
120.0	0.0646	0.064692	0.14%
150.0	0.0843	0.084424	0.15%
200.0	0.1132	0.113340	0.12%

Composite

6.65 R0031(1): Laminated Strip under Three-point Bending

Problem Description

This NAFEMS composite benchmark is a simply supported 7 layer symmetric strip with a central line load. A quarter model, with appropriate boundary conditions, is used and predictions for stress and displacement are obtained using composite shell elements of Nastran.

Solution Number

Sol 101

Features Used

- Linear Static Analysis
- Layered Composite Elements

Reference

1. Test R0031/1 from NAFEMS publication R0031, Composites Benchmarks, February 1995.

Modeling Techniques Used

One quarter of 7 layer laminated strip is modeled using CQUAD4 composite shell elements of Nastran. The model details are given below:

Material Properties

E_1 = 100GPa	ν_{12} = 0.4	G_{12} = 3GPa
E_2 = 5GPa	ν_{13} = 0.3	G_{13} = 2GPa
E_3 = 5GPa	ν_{23} = 0.3	G_{23} = 2GPa

Loading

Line load of 10 N/mm at point C.

Boundary Condition

U_z = 0.0 at point A

$U_x = \theta_y = \theta_z = 0.0$ at x = 25 and $U_y = \theta_x = \theta_z = 0.0$ at y = 5

Illustrations of the Model

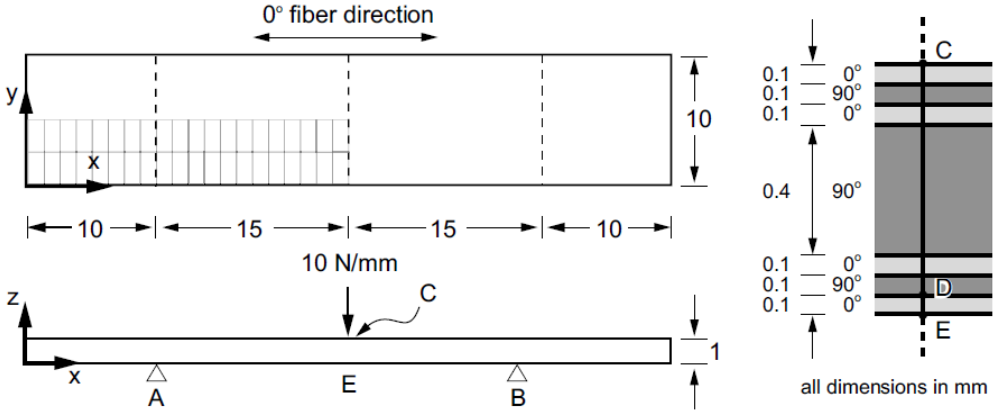


Figure 6.65-1 Laminated Strip in a Three-point Bending

Model File Name and Location

Model Name	Description	Location
`nafem_r00311.dat`	R0031(1): Laminated Strip under Three-point Bending	\verifman\nafems

Results

	NAFEMS	MSC Nastran	
		Values	% Error
σ_{11} at E (MPa)	683.9	682.7	-0.18%
σ_{13} at D (MPa)	-4.1	-4.0908	-0.22%
u_z at E (mm)	-1.06	-1.0625	0.24%

Conclusion

The results from MSC Nastran are compared with the NAFEMS results in the above table. The value of σ_{11} at point E are obtained by extrapolating stress values at element centroid of two elements adjacent to point E. More variants of this model with other composite element types are demonstrated with SOL400 in Chapter 7: Laminated Strip under Three-point Bending of *MSC Nastran Implicit Nonlinear Demonstration Guide*.

6.66 R0031(2): Wrapped Thick Cylinder under Pressure and Thermal Loading

Problem Description

This NAFEMS composite benchmark is a long thick cylinder made from isotropic material onto which external hoop windings of orthotropic material have been added. This problem demonstrates the ability to model pressure and thermal loading for composite laminated material.

Solution Number

Sol 101

Features Used

- Linear Static Analysis
- Laminated Composites

Reference

1. Test R0031/2 from NAFEMS publication R0031, Composites Benchmarks, February 1995.

Modeling Techniques Used

Due to symmetry, one eighth of the cylinder is modeled with composite CQUAD4 shell elements. The cylinder consists of two layers with layer thickness and orientation as shown in Figure 6.66-1. The model details are given below:

Material Properties

$E = 210GPa$	$\nu = 0.3$	$\alpha = 2.0 \times 10^{-5}/°C$	for inner cylinder and
$E_1 = 130GPa$	$\nu_{12} = 0.25$	$G_{12} = 10GPa$	$\alpha_{11} = 3.0 \times 10^{-6}/°C$

E_2 = 5GPa	ν_{13} = 0.25	G_{13} = 10GPa	$\alpha_{22} = 2.0 \times 10^{-5}$/°C
E_3 = 5GPa	ν_{23} = 0.0	G_{23} = 5GPa	$\alpha_{33} = 2.0 \times 10^{-5}$/°C for outer cylinder

Loading

Case-1: Internal Pressure of 200 MPa

Case-2: Internal Pressure of 200 MPa and temperature rise of 130°C. The temperature loading simulates the pre-stress due to tension in the hoop windings.

Boundary Condition

Axial displacement U_z = 0 at z = 0.

Illustrations of the Model

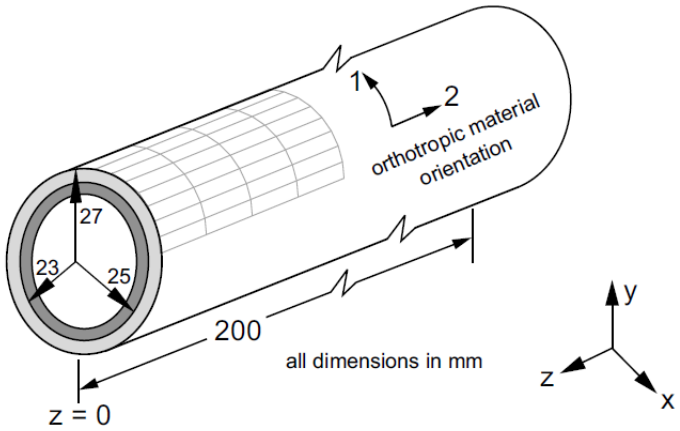


Figure 6.66-1 Wrapped Thick Cylinder under Pressure and Thermal Loading

Model File Name and Location

Model Name	Description	Location
`nafem_r00312.dat`	R0031(2): Wrapped Thick Cylinder under Pressure and Thermal Loading	\verifman\nafems

Results

	Hoop Stress (MPa)	NAFEMS	MSC Nastran	
			Values	% Error
Case 1	at radius = 24	1483	1414.2	-4.64%
	at radius = 26	822	874.8	6.42%
Case 2	at radius = 24	1309	1235.9	-5.58%
	at radius = 26	994	1052.9	5.93%

Conclusion

The hoop stress values in the inner and outer cylinders for the two load cases at r = 24 and r = 26 from MSC Nastran are compared with the NAFEMS results in the above table. SOL400 variant of this model is demonstrated in Chapter 8: Wrapped Thick Cylinder under Pressure and Thermal Loading of *MSC Nastran Implicit Nonlinear Demonstration Guide.*

6.67 R0031 (3): Three-layer Sandwich Shell under Normal Pressure Loading

Problem Description

This NAFEMS composite benchmark is a simply supported square sandwich plate subjected to a normal pressure loading. The orthotropic materials of the sandwich plate are defined in such a way that the facing sheets carry the bending and the shear is carried by the core.

Solution Number

Sol 101

Features Used

- Linear Static Analysis
- Laminated Composites

Reference

1. Test R0031/3 from NAFEMS publication R0031, Composites Benchmarks, February 1995.

Modeling Techniques Used

One quarter of the square sandwich plate is modeled with composite CQUAD4 shell elements. It consists of two outer facing sheets and a thick central core as shown in Figure 6.67-1. The model details are given below:

Material Properties

$E_1 = 10\times10^5$ psi	$E_2 = 4\times10^6$ psi	$\nu_{12} = 0.3$	and
$G_{12} = 1.875\times10^6$ psi	$G_{13} = 1.875\times10^6$ psi	$G_{23} = 1.875\times10^6$ psi	for face sheets
$E_1 = 10$ psi	$E_2 = 10$ psi	$\nu_{12} = 0.0$	and
$G_{12} = 10$ psi	$G_{13} = 3\times10^4$ psi	$G_{23} = 1.2\times10^4$ psi	for core

Loading

Plate is subjected to uniform pressure of 100 psi.

Main Index

Boundary Condition

The plate is simply supported at all four edges and fixed at four corners. Symmetric boundary conditions are applied at the symmetric planes of one quarter model.

Illustrations of the Model

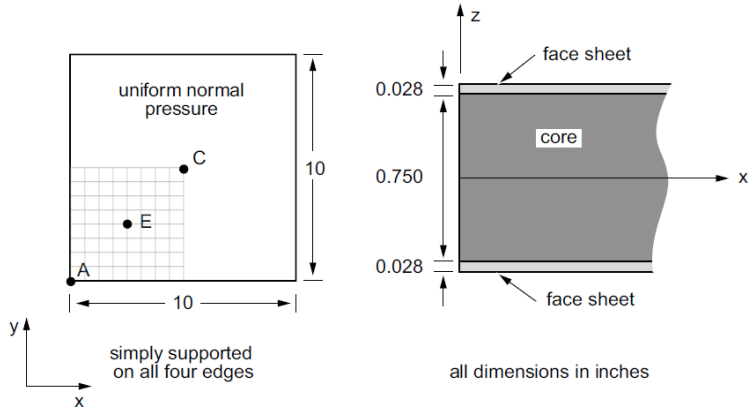


Figure 6.67-1 Wrapped Thick Cylinder under Pressure and Thermal Loading

Model File Name and Location

Model Name	Description	Location
`nafem_r00313.dat`	R0031(3): Three-layer Sandwich Shell under Normal Pressure Loading	\verifman\nafems

Results

	NAFEMS	MSC Nastran	
		Values	% Error
U_z at C (in)	-0.123	-0.1226	-0.33%
σ_{11} at C (kpsi)	34.45	34.03	-1.22%
σ_{22} at C (kpsi)	13.93	13.29	-4.59%
σ_{12} at E (kpsi)	-5.07	-5.04	-0.59%

Conclusion

The Z-displacement at point C and stress values at points C and E from MSC Nastran are compared with the NAFEMS results in the above table. The variants of this model with other composite element types are demonstrated with SOL400 in Chapter 9: Three-layer Sandwich Shell under Normal Pressure Loading of *MSC Nastran Implicit Nonlinear Demonstration Guide.*

Contact

6.68 R0094 (1): Benchmark 1: 2D Cylinder Roller Contact

Problem Description

In this NAFEMS contact benchmark, a steel cylinder is pressed into an aluminum block. The solution with a 2-D plane strain approximation of this problem from Nastran is compared to an analytical solution for a frictionless case.

Solution Number

SOL 400

Features Used

- Nonlinear static analysis.
- Advancing contact area with curved contact surfaces
- Deformable to deformable contact

Reference

1. Benchmark 1 from NAFEMS publication R0094, Advanced Finite Element Contact Benchmarks, 2006.

Modeling Techniques Used

Steel cylinder and aluminum block are modeled as two contact bodies with either linear or quadratic plane strain elements. The model details are given below:

Material Properties

$E_{cylinder}$	=	210 kN/mm^2
$\nu_{cylinder}$	=	0.3
E_{block}	=	70 kN/mm^2
ν_{block}	=	0.3

Loading

Vertical point load F = 35kN

Boundary Condition

Symmetric displacement constraints along vertical symmetric line

Bottom surface of the block is fixed ($U_x = U_y = 0.0$)

Touching contact between cylinder and block with segment to segment contact

Illustrations of the Model

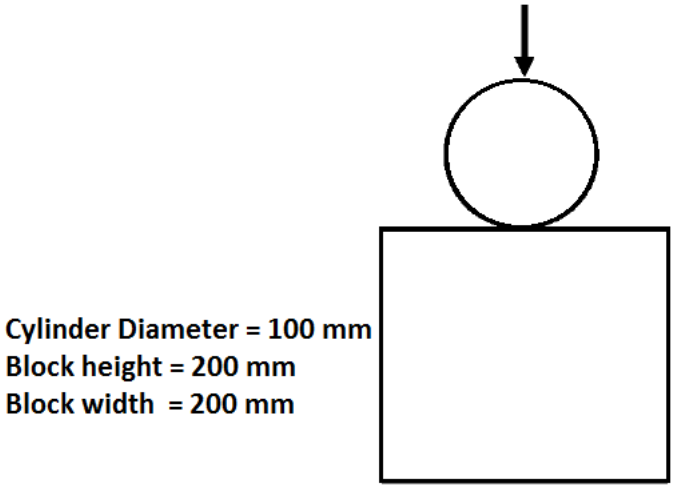
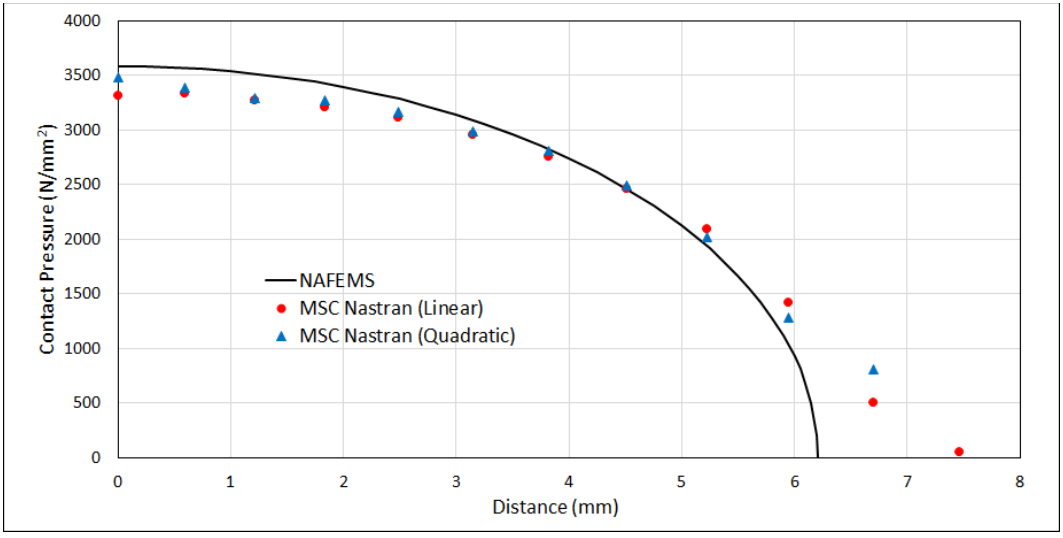


Figure 6.68-1 2D Cylinder Roller Contact

Model File Name and Location

Model Name	Description	Location
nafem_r00941a.dat	Linear element	\verifman\nafems
nafem_r00941b.dat	Quadratic element	\verifman\nafems

Results



Element	a_{min} (mm)	a_{avg} (mm)	a_{max} (mm)	% Error	P_{max}	% Error
Linear	6.698577	7.0811275	7.463678	14.03%	3335.70	-6.96%
Quadratic	5.952867	6.325722	6.698577	1.86%	3486.60	-2.75
NAFEMS		6.21			3585.37	

Conclusion

The contact pressure results along the touching surfaces of steel cylinder from MSC Nastran are compared with the NAFEMS results in the above plot. The length of contact zone and maximum contact pressure values from MSC Nastran are compared with NAFEMS results in the above table. More details of this contact benchmark model with and without friction are demonstrated using node to segment contact in 2-D Cylindrical Roller Contact of *MSC Nastran Implicit Nonlinear Demonstration Problems Guide*.

6.69 R0094 (2): Benchmark 2: 3D Punch (Rounded Edges)

Problem Description

In this NAFEMS contact benchmark, an axisymmetric steel punch is compressed on an aluminum foundation. The solution with a 2-D axisymmetric model of this problem from Nastran is compared to NAFEMS solution for a frictionless case.

Solution Number

SOL 400

Features Used

- Nonlinear static analysis.
- Axisymmetric contact
- Deformable to deformable contact

Reference

1. Benchmark 2 from NAFEMS publication R0094, Advanced Finite Element Contact Benchmarks, 2006.

Modeling Techniques Used

Steel punch and aluminum foundation are modeled as two contact bodies with either linear axisymmetric CQUADX elements. The model details are given below:

Material Properties

E_{punch}	=	210 kN/mm^2
ν_{punch}	=	0.3
$E_{foundation}$	=	70 kN/mm^2
$\nu_{foundation}$	=	0.3

Loading

Uniform pressure in punch P = 100 N/mm^2

Boundary Condition

Bottom surface of the foundation is fixed ($U_x = U_y = 0.0$)

Touching contact between punch and foundation with node to segment contact

Illustrations of the Model

- Punch diameter = 100 mm
- Punch height = 100 mm
- Fillet radius = 10 mm

- Foundation diameter = 200 mm
- Foundation height = 200 mm

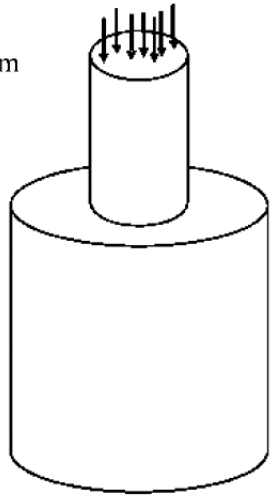
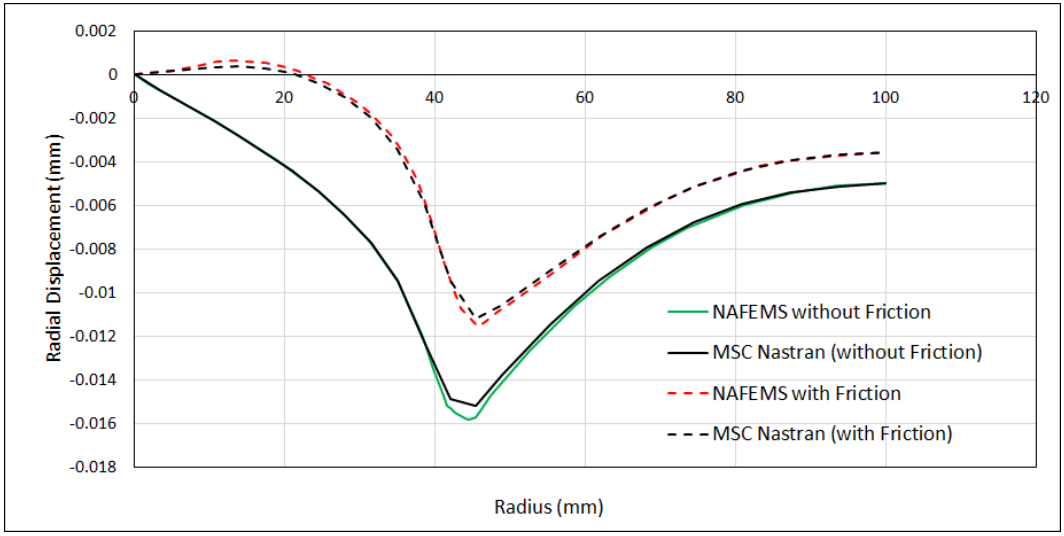


Figure 6.69-1 3D Punch

Model File Name and Location

Model Name	Description	Location
nafem_r00942a.dat	Without friction	\verifman\nafems
nafem_r00942b.dat	With friction	\verifman\nafems

Results



Conclusion

The radial displacement results along the touched surfaces of aluminum foundation from MSC Nastran are compared with the NAFEMS results in the above plot for the cases with and without friction. More details of this contact benchmark model with 3D solid elements are demonstrated in Chapter 2: 3-D Punch (Rounded Edges) Contact of *MSC Nastran Implicit Nonlinear Demonstration Guide*.

6.70 R0094 (3): Benchmark 3: 3D Sheet Metal Forming

Problem Description

This NAFEMS contact benchmark is an approximation of Numisheet 2002 - Benchmark B problem. Simulations are carried out with MSC Nastran with 2D plane strain and 3D shell models to find the angles before and after spring back.

Solution Number

SOL 400

Features Used

- Nonlinear static analysis.
- Large sliding contact around circular surface
- Deformable to deformable contact

Reference

1. Benchmark 3 from NAFEMS publication R0094, Advanced Finite Element Contact Benchmarks, 2006.

Modeling Techniques Used

Sheet is modeled with either 2D plane strain or shell elements. The model details are given below:

Material Properties

E	=	70.5 kN/mm^2
ν	=	0.342
Initial yield stress σ_0	=	194 N/mm^2
Hollomon hardening: σ	=	$k\varepsilon^n$
		with K = 550.4 N/mm^2 & n =0.223

Loading

Prescribed vertical displacement of punch = -28.5 mm

Boundary Condition

Symmetric displacement restraints (half symmetry)

Rigid surface for Die is fixed

Touching contact of sheet with punch and die using node to segment contact

Illustrations of the Model

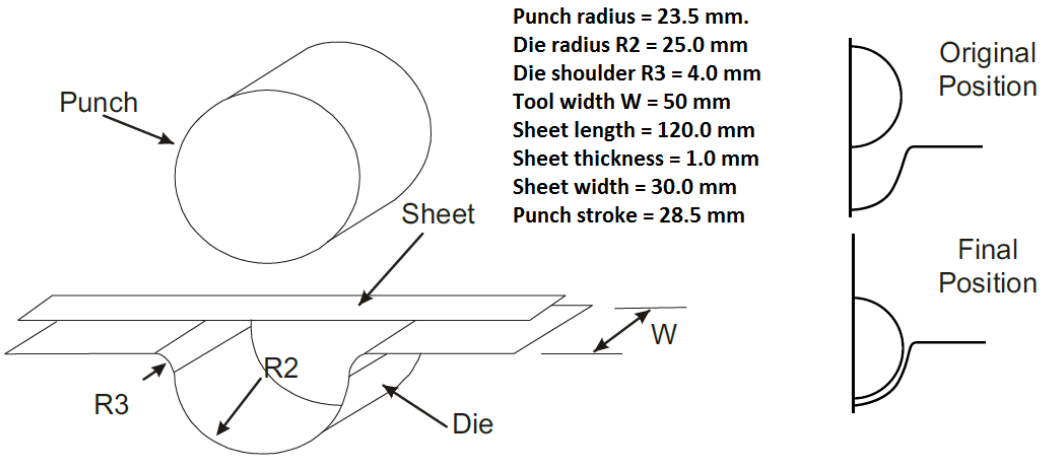
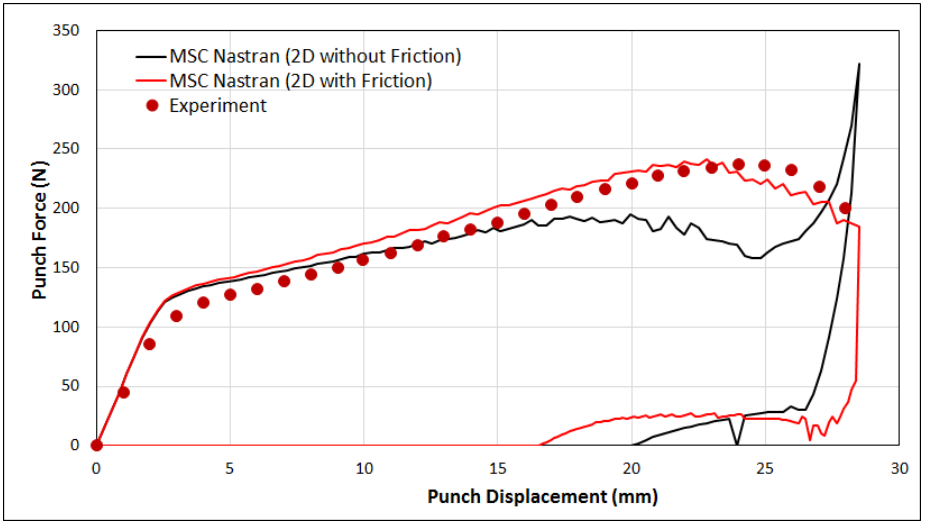


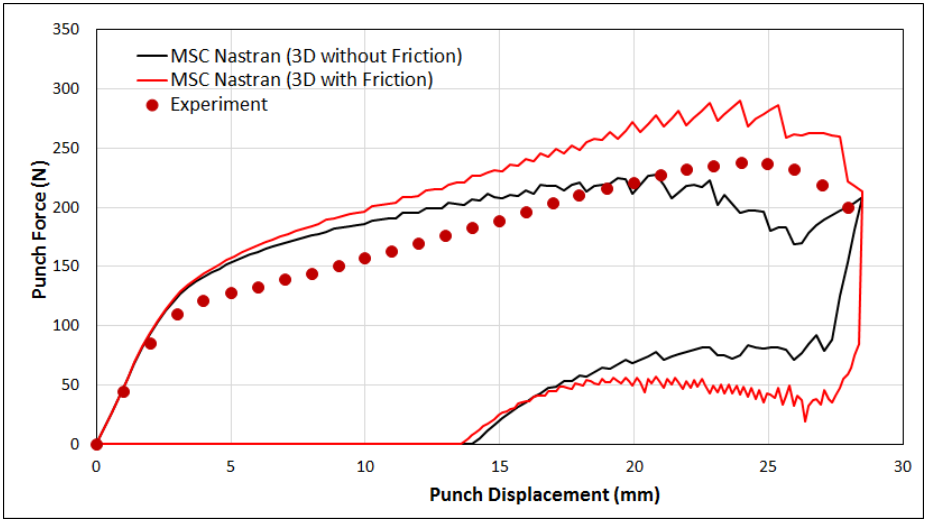
Figure 6.70-1 3D Sheet Forming

Model File Name and Location

Model Name	Description	Location
nafem_r00943a.dat	2D Without friction	\verifman\nafems
nafem_r00943b.dat	2D With friction	\verifman\nafems
nafem_r00943c.dat	3D Without friction	\verifman\nafems
nafem_r00943d.dat	3D With friction	\verifman\nafems

Results





Conclusion

The force-displacement curves of punch from MSC Nastran for 2D plane strain and 3D shell models are compared with the NAFEMS results in the above plots for the cases with and without friction. More details of this contact benchmark model are presented in Chapter 3: 3-D Sheet Metal Forming of *MSC Nastran Implicit Nonlinear Demonstration Guide*.

6.71 R0094 (4): Benchmark 4: 3D Loaded Pin

Problem Description

This NAFEMS contact benchmark evaluates the performance of contact algorithms at the curved boundaries. A cylindrical pin is located in the cylindrical hole of a strip, with identical diameters of the hole and the pin. Two equal point loads are applied to the center of the pin, resulting to a loss of contact at one side of the pin and a localized contact area on the other side. A 2D plane strain representation of the model is used to evaluate the contact normal stress in the contact surfaces of strip and pin.

Solution Number

SOL 400

Features Used

- Nonlinear Static Analysis.
- Receding Contact Area with Curved Contact Surfaces
- Deformable to Deformable Contact with Friction

Reference

1. Benchmark 4 from NAFEMS publication R0094, Advanced Finite Element Contact Benchmarks, 2006.

Modeling Techniques Used

Both pin and strip are modeled with either 2D plane strain quadratic elements. The model details are given below:

Material Properties

E_{pin}	=	210 kN/mm^2
ν_{pin}	=	0.3
E_{sheet}	=	70 kN/mm^2
ν_{sheet}	=	0.3

Loading

Point load of 100kN is applied at the center of pin.

Boundary Condition

Symmetric displacement restraints (half symmetry).

Left side of sheet is fixed.

Touching contact of strip with pin (Node to Segment contact).

Illustrations of the Model

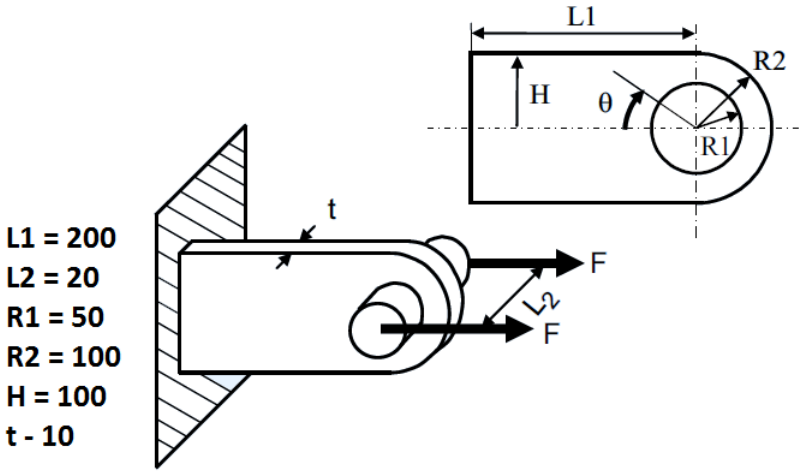
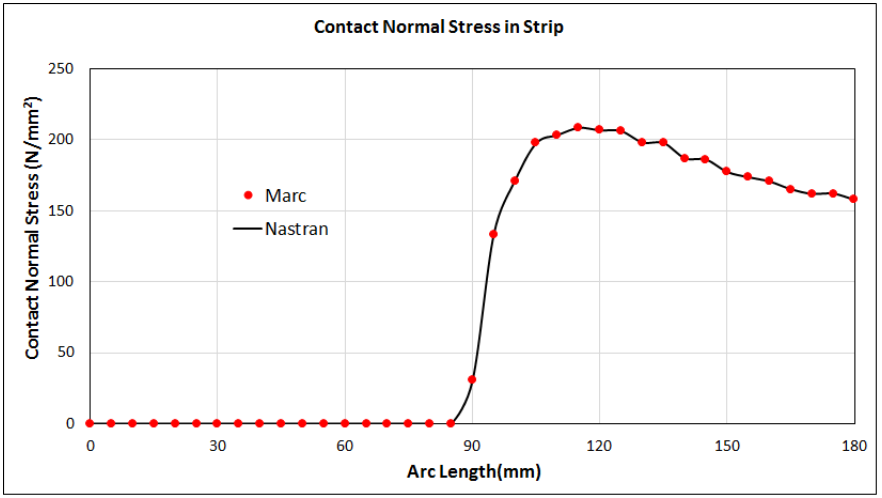


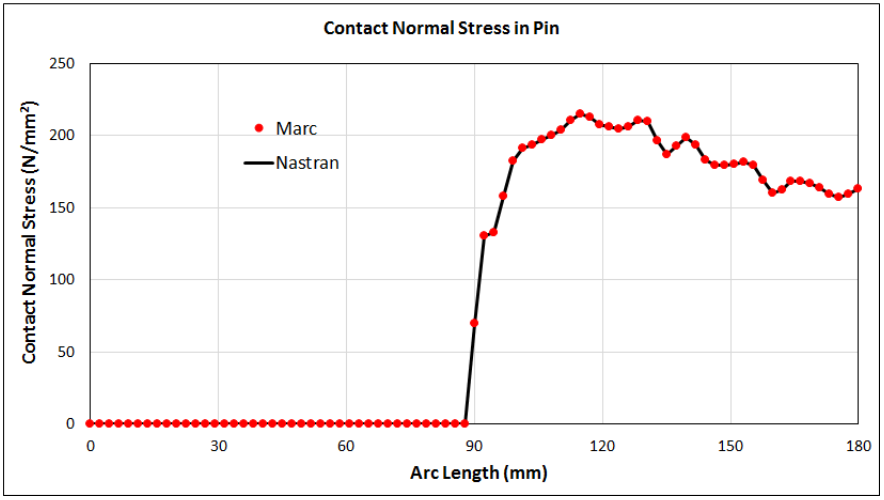
Figure 6.71-1 3D Loaded Pin

Model File Name and Location

Model Name	Description	Location
nafem_r00944a.dat	2D Without friction	\verifman\nafems

Results





Conclusion

The results from MSC Nastran for the variation of contact normal stress along the contact surfaces of pin and strip are compared with the results from Marc, which is also used with the node to segment contact in the NAFEMS publication for this benchmark. Segment to segment contact will give more smooth variation of the contact normal stress in both the pin and strip surfaces. Further details of this contact benchmark model are presented in Chapter 4: 3-D Loaded Pin with Friction of *MSC Nastran Implicit Nonlinear Demonstration Problems Guide*.

6.72 R0094 (5): Benchmark 5: 3D Steel Roller on Rubber

Problem Description

In this NAFEMS contact benchmark, a steel cylindrical roller rotates on a rubber base. It tests the ability of contact algorithms to model the movement of two surfaces when they are in contact, as the rubber will stretch compared to the steel cylinder.

Solution Number

SOL 400

Features Used

- Nonlinear Static Analysis.
- Rolling Contact with Friction
- Deformable and Rigid Contact Bodies

Reference

1. Benchmark 5 from NAFEMS publication R0094, Advanced Finite Element Contact Benchmarks, 2006.

Modeling Techniques Used

Both 2D plane strain and 3D solid representations are used for this contact simulation of this benchmark. The model details are given below:

Material Properties

E_{steel} = 210 kN/mm^2

ν_{steel} = 0.3

$C_{10,\ rubber}$ = 10 kN/mm^2 with Neo-Hookean material description

Loading

Move bottom surface of rubber 3 mm up during time period 0-1 second and held at this position from 1-2 second. Prescribed rotation of steel roller by 360 degrees during time period 1-2 second.

Boundary Condition

No rotation of roller and sheet x-displacement fixed during time period 0-1 second.

Sheet is free to move in horizontal direction during time period 1-2 second.

Touching contact of roller and sheet and rotating rigid body is glued to roller (Segment to Segment contact).

Illustrations of the Model

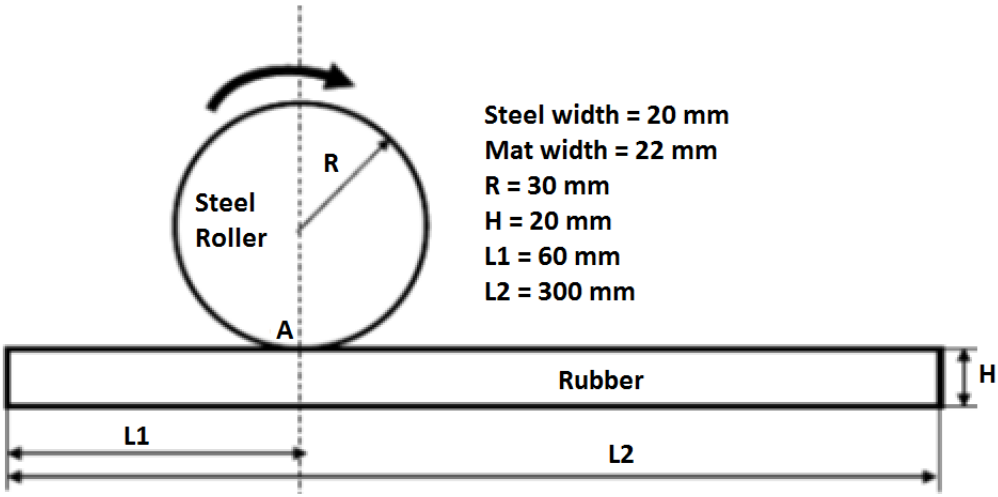
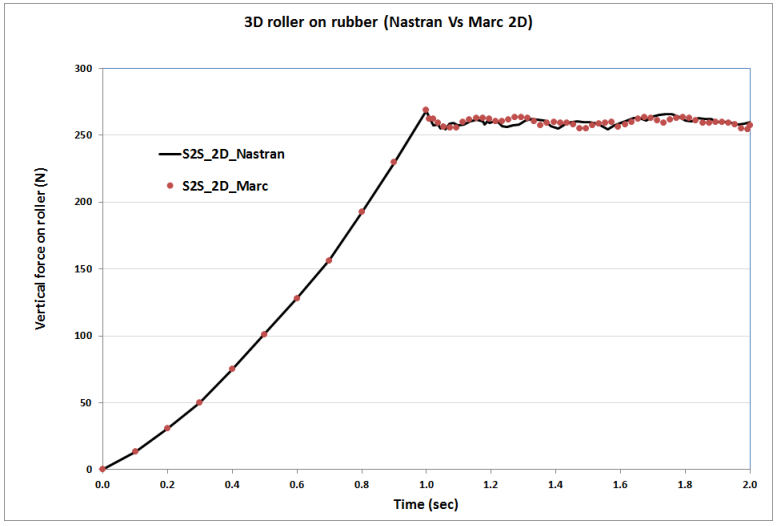


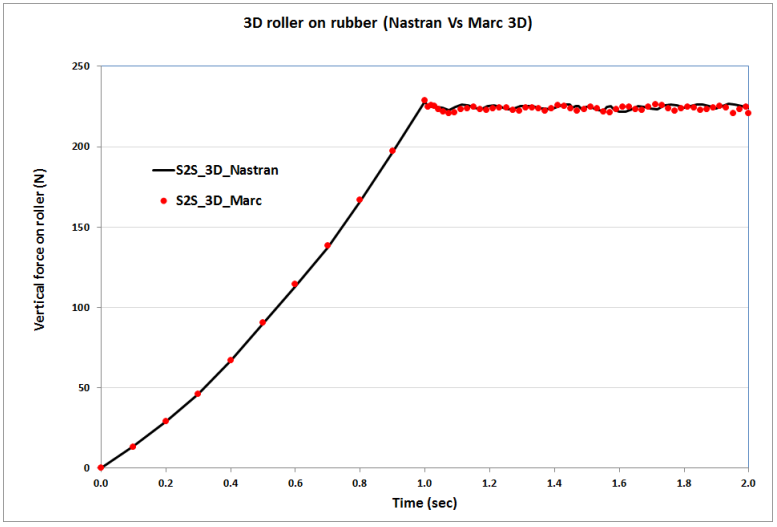
Figure 6-1 3D Steel Roller on Rubber

Model File Name and Location

Model Name	Description	Location
nafem_r00945a.dat	2D with friction	\verifman\nafems
nafem_r00945b.dat	2D with friction	\verifman\nafems

Results





Conclusion

The results from MSC Nastran for the variation of vertical force on roller are compared with the results from Marc, which is also used in the NAFEMS publication for this benchmark. Further details of this contact benchmark model are presented in Chapter 5: Steel Roller on Rubber of *MSC Nastran Implicit Nonlinear Demonstration Problems Guide*.

Geometric Nonlinear

6.73 3DNLG1: Elastic Large Deflection Response of A Z-Shaped Cantilever under End Load

Problem Description

This NAFEMS geometric nonlinear benchmark on Z-shaped cantilever under an end load is used to test beam element's formulation for large displacement and rotations. The model is subjected to significant stress stiffening accompanied by stress reversal.

Solution Number

SOL 400

Features Used

- Implicit nonlinear analysis.
- 3D Large rotation - large displacement with membrane and bending actions.
- Tension stiffening with change in sign of bending moment at section A.

Reference

1. Test 3DNLG-1 from NAFEMS Publication 'A Review of Benchmark Problems for Geometric Non-Linear Behaviour of 3D Beams & Shells (Summary)' by Prinja. N & Clegg. R, R0024, August 1993.

Modeling Techniques Used

The Z- shaped cantilever is modeled using CBEAM elements with different mesh densities (number of elements varying from 30 to 150). The model details are given below:

Material Properties

$E = 2.0 \times 10^7$ $\nu = 0.3$

Loading

Total load P = 4000.0 is applied at the free end in the Z-direction in 3 different steps.

Boundary Condition

All degrees of freedom are constrained at the built-in end.

Illustrations of the Model

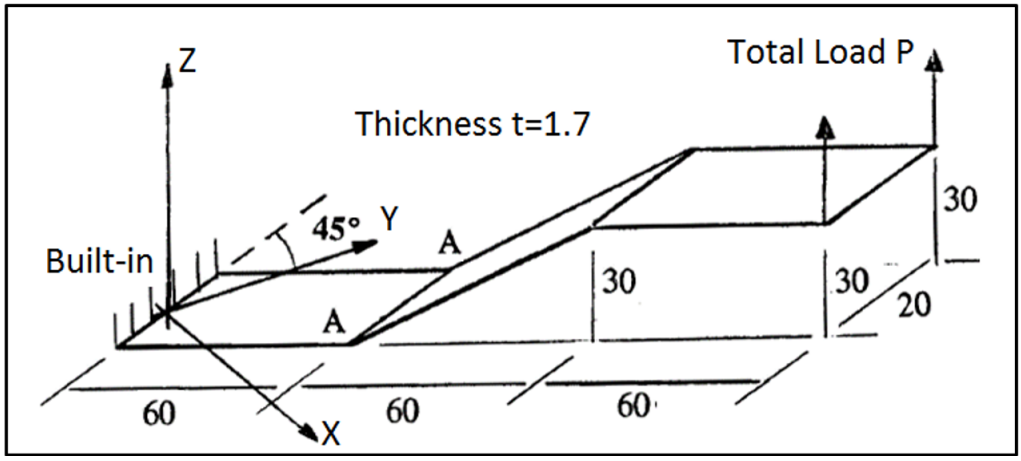


Figure 6.73-1 Elastic Large Deflection Response of A Z-Shaped Cantilever

Model File Name and Location

Model File Name	Description	Location
`nafem_3dnlg1n.dat`	3DNLG1: Elastic Large Deflection Response of A Z-Shaped Cantilever	\verifman\nafems

Results

Tip Z-Displacement						
Load	Target Value	Number of Elements				
		30	60	90	120	150
104.5	80.4	80.690	80.686	80.685	80.685	80.685
1263	133.1	133.146	133.126	133.122	133.121	133.120
4000	143.4	143.534	143.482	143.472	143.469	143.467

Moment at Section A-A						
Load	Target Value	Number of Elements				
		30	60	90	120	150
104.5	-8309	-8359.355	-8359.653	-8359.739	-8359.780	-8359.803
1263	-4966	-5125.586	-5145.153	-5148.734	-5149.957	-5150.507
4000	10743	11064.210	143.482	11060.750	11059.610	11059.120

Conclusion

The results for tip Z-displacement and moment at section A-A from MSC Nastran are compared with NAFEMS results at loads P = 104.5, 1263.0 and 4000.0 in the above tables. The largest deviation of tip Z-displacement from the NAFEMS solution is less than 0.4% and largest deviation of moment at section A-A from the NAFEMS solution is less than 3.8%.The results of MSC Nastran is also found to be accurate even for the course mesh of 30 elements for this model.

6.74 3DNLG2: Elastic Large Deflection Response of a Pear-Shaped Cylinder under End Shortening

Problem Description

This NAFEMS benchmark involves initial softening in the load defection path due to the collapse of the flat plate sections. This is accompanied by stress redistribution and subsequent stiffening to a maximum load, which causes buckling of curved plate sections.

Solution Number

SOL 400

Features Used

- Implicit nonlinear analysis with elastic large defection and large rotation
- Initial softening behavior followed by stiffening to final buckling
- Interaction of bending, membrane and shear effects.

Reference

1. Test 3DNLG-2 from NAFEMS Publication 'A Review of Benchmark Problems for Geometric Non-Linear Behavior of 3D Beams & Shells (Summary)' by Prinja. N & Clegg. R, R0024, August 1993.

Modeling Techniques Used

The pear shaped cylinder is modeled using 960 linear CQUAD4 shell elements. The model details are given below:

Material Properties

$E = 1.0\times10^{7}$ and $\nu = 0.3$

Loading

Uniform negative z-displacement at nodes initially lying on plane z =0.4. Due to limitation of arc length method's implementation in MSC Nastran for enforced displacement, the end shortening is applied with a maximum load of 2600 in -z direction.

Boundary Condition

$U_x = \theta_y = \theta_z = 0.0$ on plane x = 0.0, $U_z = \theta_x = \theta_y = 0.0$ on plane z = 0.0 and $U_x = U_y = 0.0$ on plane z = 0.4.

Illustrations of the Model

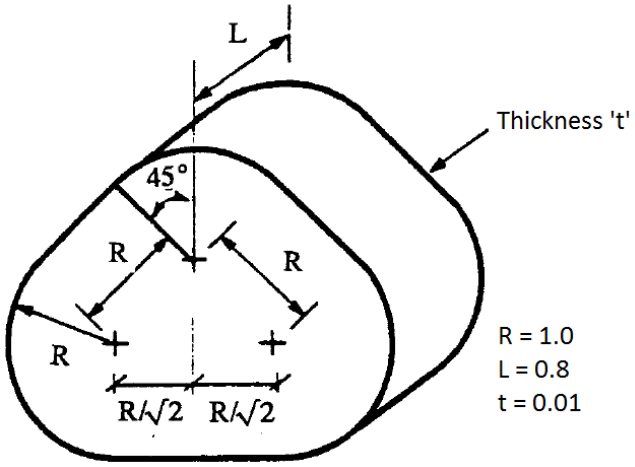
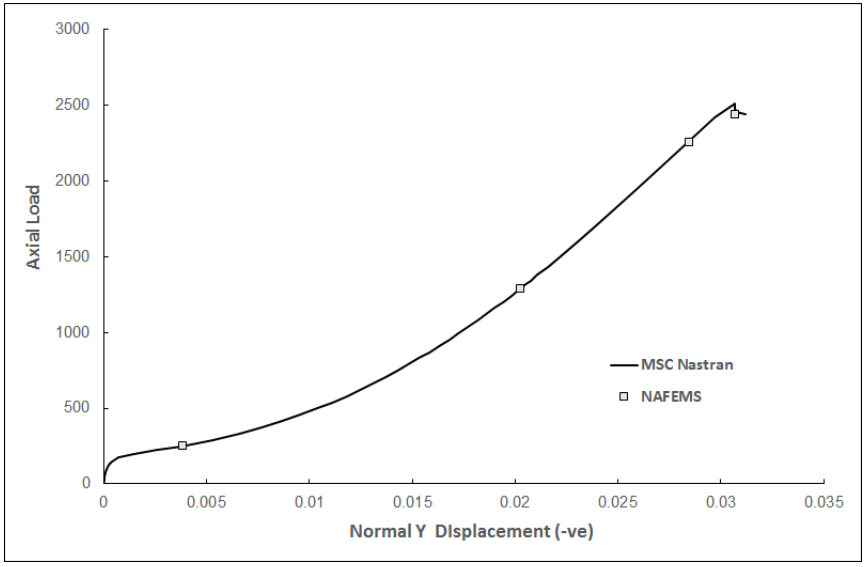


Figure 6.74-1 Pear Shaped Cylinder under End Shortening

Model File Name and Location

Model File Name	Description	Location
`nafem_3dnlg2.dat`	3DNLG2: Elastic Large Deflection Response of a Pear-Shaped Cylinder under End Shortening	\verifman\nafems

Results



Conclusion

The results from MSC Nastran are compared with NAFEMS results at loads 249.26, 1285.44, 2255.87, 2437 in the above figure. The largest deviation from the NAFEMS solution around 3.3% at all these reference points. The collapse load predicted by MSC Nastran is 2511.97 which is 3.1% higher compared to the NAFEMS estimated collapse load of 2437.

6.75 3DNLG6: Buckling of a Flat Plate When Subjected To In-Plane Shear

Problem Description

This NAFEMS benchmark on buckling of imperfect flat plate demonstrates shell element's formulation for modeling shear behavior.

Solution Number

SOL 400

Features Used

- Implicit nonlinear analysis.
- Large rotation - large displacement response with in-plane shear
- Initial imperfection.

Reference

1. Test 3DNLG-6 from NAFEMS Publication 'A Review of Benchmark Problems for Geometric Non-Linear Behavior of 3D Beams & Shells (Summary)' by Prinja. N & Clegg. R, R0024, August 1993.

Modeling Techniques Used

The square plate is modeled using 64 quadratic CQUAD8 shell elements. The model details are given below:

Material Properties

$E = 6.4 \times 10^6$ $\nu = 0.3$

Loading

Maximum shear load/edge = 2.12×10^8 is applied in all four edges using modified Riks arc length method.

Boundary Condition

$U_z = 0.0$ in all four edges. $U_x = U_y = z = 0.0$ at central node.

Initial Imperfection

The initial imperfection is defined by the following equations:

$$z = 0.5 \times t \times f(x, y) / f(L/2, L/2)$$

where;

$$f(x, y) = \sum_m \sum_n a_{mn} \sin\left(\frac{m\pi x}{L}\right) \sin\left(\frac{n\pi y}{L}\right)$$

a_{11}=1.0a_{22} = 0.2897

a_{13} = -0.0706a_{31} = -0.0691

a_{13} = 0.0384a_{42} = 0.0032

These imperfection amplitudes calculated at all grid points are used to define imperfection shape of the model. The bulk data entry 'IMPGEOM' is used to define this imperfection geometry in MSC Nastran input file.

Illustrations of the Model

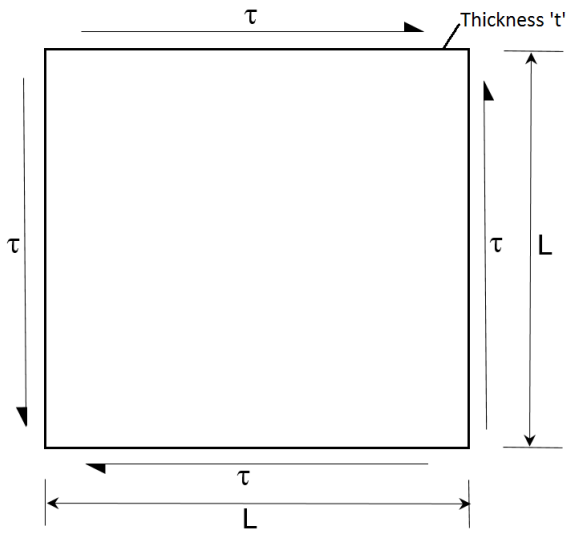
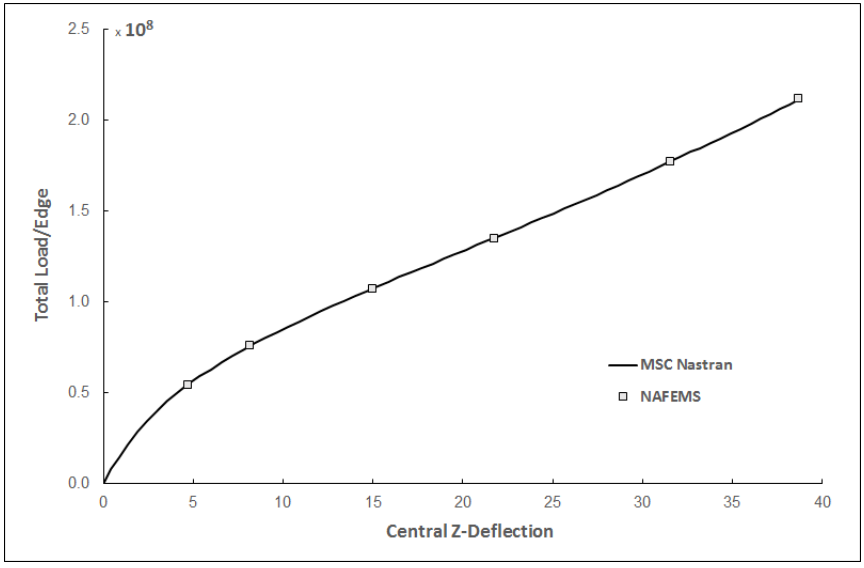


Figure 6.75-1 Buckling of flat plate

Model File Name and Location

Model File Name	Description	Location
`nafem_3dnlg6.dat` `nafem_3dnlg6.impf`	3DNLG6: Buckling of a Flat Plate when Subjected To In-Plane Shear	\verifman\nafems

Results



Conclusion

The results from MSC Nastran are compared with NAFEMS results at edge loads 0.5442E8, 0.7585E8, 1.074E8, 1.3521E8, 1.7718E8 and 2.1117E8 in the above figure. The largest deviation from the NAFEMS solution is less than 1.0% at all these reference points.

6.76 3DNLG7: Elastic Large Deflection Response of a Hinged Spherical Shell Under Pressure Loading

Problem Description

This NAFEMS benchmark involves snap through buckling response of hinged spherical shell under pressure load. It demonstrates the ability of Riks arc length method to trace the positive and negative tangents in the load-displacement equilibrium path.

Solution Number

SOL 400

Features Used

- Implicit nonlinear analysis.
- Snap through buckling with arc length method.
- Large displacement - large rotation response.

Reference

1. Test 3DNLG-7 from NAFEMS Publication 'A Review of Benchmark Problems for Geometric Non-Linear Behaviour of 3D Beams & Shells (Summary)' by Prinja. N & Clegg. R, R0024, August 1993.

Modeling Techniques Used

The hinged spherical shell is modelled using 1024 linear CQUAD4 shell elements. The shell mid surface is defined as Z = 2.0285x10-4[X(1570-X)+Y(1570-Y)]. Other details of the model are given below:

Material Properties

E = 69 ν = 0.3

Loading

Maximum distributed pressure of 0.1 applied normal to shell surface using Riks arc length method.

Boundary Condition

$U_x = U_y = U_z = 0.0$ on all four edges.

Illustrations of the Model

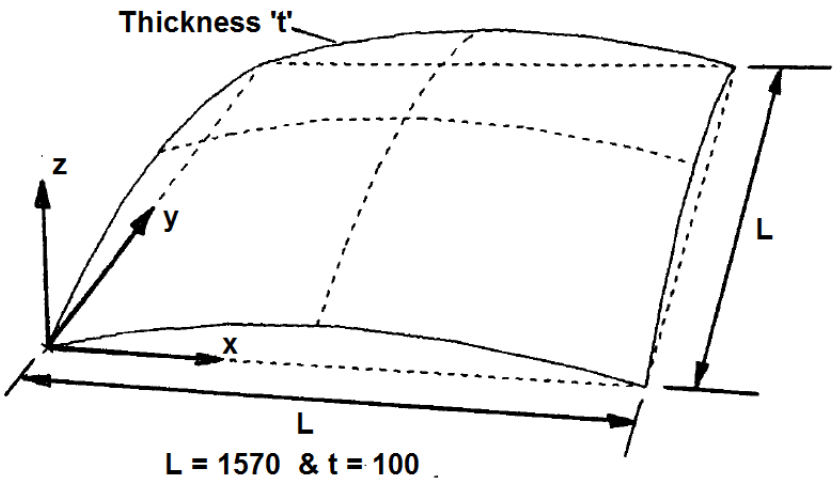
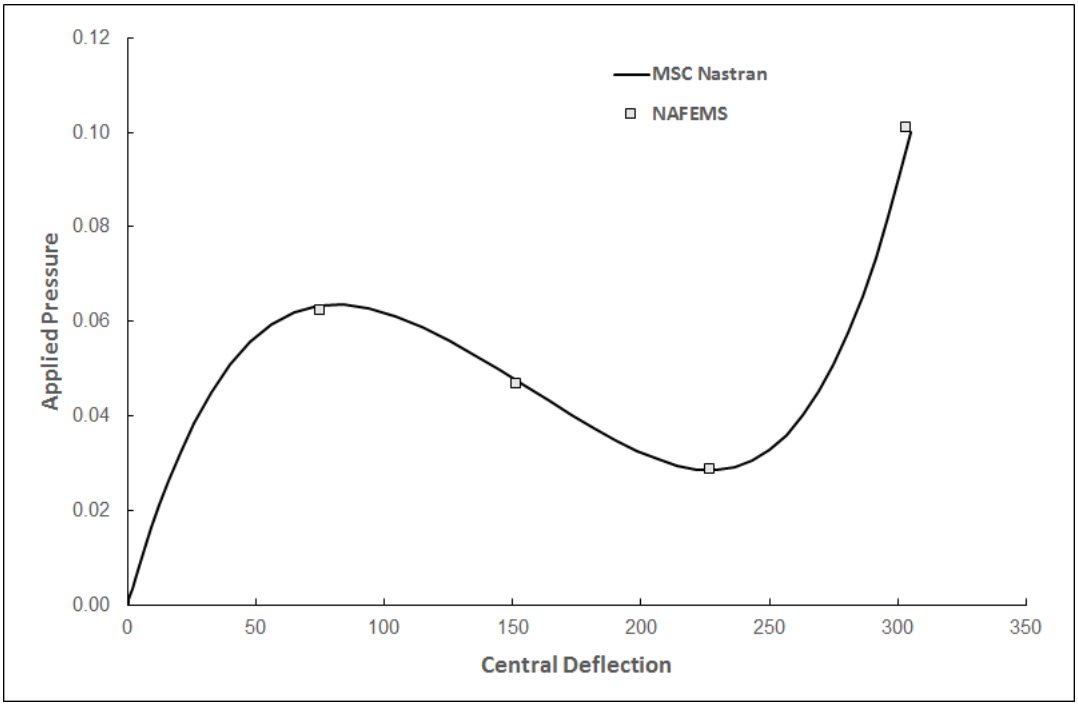


Figure 6.76-1 Hinged Spherical Shell under Pressure Loading

Model File Name and Location

Model File Name	Description	Location
`nafem_3dnlg7.dat`	3DNLG7: Elastic Large Deflection Response of a Hinged Spherical Shell Under Pressure Loading	\verifman\nafems

Results



Conclusion

The results from MSC Nastran are compared with NAFEMS results at loads 0.0624, 0.0468, 0.02885 and 0.1 in the above figure. The largest deviations from the NAFEMS solution are around 1.83% and -0.93% at limit points 1 and 2 respectively.

6.77 3DNLG9: Large Elastic Deflection of a Pinched Hemispherical Shell

Problem Description

This NAFEMS benchmark is a good test case for membrane locking of doubly curved shell and it is an extension of linear NAFEMS benchmark on pinched hemispherical shell to incorporate the effect of large displacement and large rotation.

Solution Number

SOL 400

Features Used

- Implicit nonlinear analysis.
- Large rigid body rotation with stress redistribution.
- Membrane, bending and twisting structural actions.

Reference

1. Test 3DNLG-9 from NAFEMS Publication 'A Review of Benchmark Problems for Geometric Non-Linear Behavior of 3D Beams & Shells (Summary)' by Prinja. N & Clegg. R, R0024, August 1993.

Modeling Techniques Used

The hemispherical shell is modeled using 256 linear CQUAD4 shell elements. Other details of the model are given below:

Material Properties

$E = 6.825 \times 10^7 \quad \nu = 0.3$

Loading

Maximum load of P_x =.-100 at point A and P_y = 100 at point B

Boundary Condition

$U_x = \theta_y = \theta_z = 0.0$ on plane x = 0, $U_y = \theta_x = \theta_z = 0.0$ on plane y = 0, $U_z = 0$ at point A to prevent rigid body motion.

Illustrations of the Model

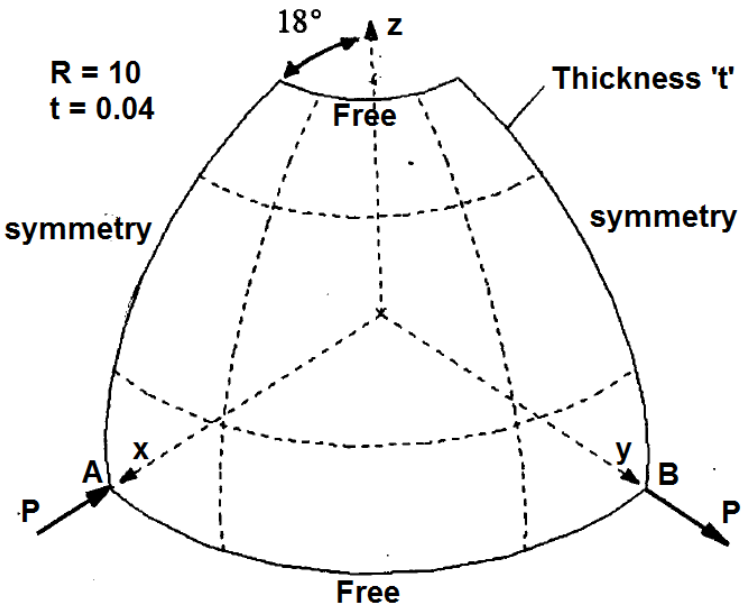
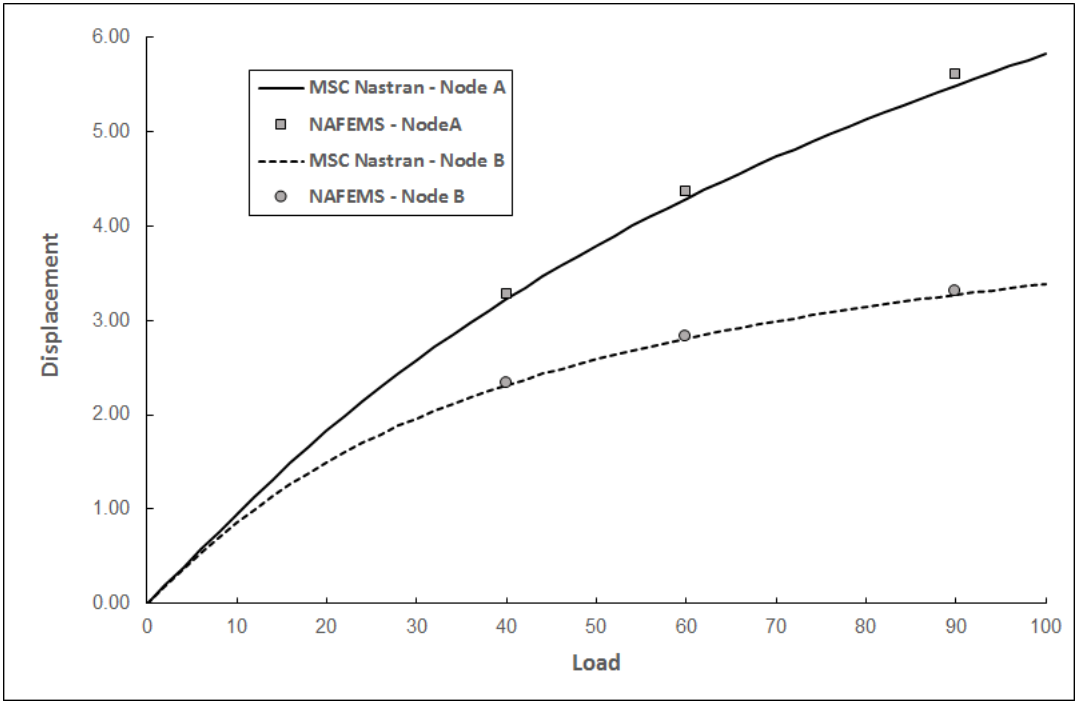


Figure 6.77-1 Hinged Spherical Shell under Pressure Loading

Model File Name and Location

Model File Name	Description	Location
`nafem_3dnlg9.dat`	3DNLG-9: Large Elastic Deflection of a Pinched Hemispherical Shell	\verifman\nafems

Results



Conclusion

The results from MSC Nastran are compared with NAFEMS results in the above figure. The maximum deviation of 2.1% and 1.2% are seen at points A and B respectively.

6.78 3DNLG10: Elastic-Plastic Behavior of a Stiffened Cylindrical Panel under Compressive End Load

Problem Description

This NAFEMS benchmark illustrate the snap through behavior of stiffened shell panel shell with interaction between geometric and material nonlinearities.

Solution Number

SOL 400

Features Used

- Implicit nonlinear analysis.
- Snap through with elasto-plastic large deflection behavior.
- Initial imperfection.

Reference

1. Test 3DNLG-10 from NAFEMS Publication 'A Review of Benchmark Problems for Geometric Non-Linear Behaviour of 3D Beams & Shells (Summary)' by Prinja. N & Clegg. R, R0024, August 1993.

Modeling Techniques Used

The cylindrical panel is modelled using 40 quadratic CQUAD8 shell elements. The model details are given below:

Material Properties

$E = 2.1x10^5$, $\nu = 0.3$, $\sigma_y=350$

Loading

Compressive load of 12875 distributed evenly at end A.

Boundary Condition

$U_{circ} = \theta_{radial} = \theta_z = 0.0$ at edges 1 and 2

$U_{circ} = U_{radial} = \theta_{radial} = 0.0$ at end A.

$U_z = \theta_{circ} = \theta_{radial} = 0.0$ for panel at Z = L/2

$U_z = \theta_X = \theta_Y = 0.0$ for stiffener at Z = L/2

Initial Imperfection

The initial imperfection for cylindrical panel and stiffener are defined by the following equations:

$$\Delta R = 0.569 \sin\left(\frac{\theta \pi^2 Z}{7200}\right)$$ for cylindrical panel (cylindrical system -RZ)

where;

with $-9 \le \theta \le 9$ and $0 \le Z \le 200$

and $$\Delta z = (-0.3) \sin\left(\frac{\pi x}{400}\right) \sin\left(\frac{\pi y}{20}\right)$$

with $0 \le x \le 200$ and $0 \le y \le 10$.

These imperfection amplitudes calculated at all grid points are used to define imperfection shape of the model. The bulk data entry IMPGEOM is used to define this imperfection geometry in MSC Nastran input file.

Illustrations of the Model

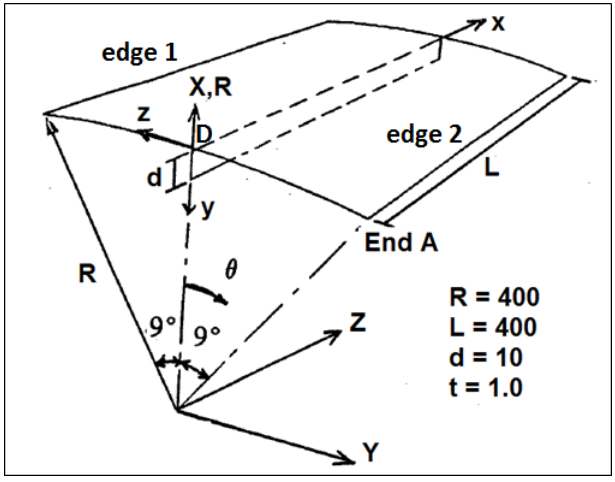
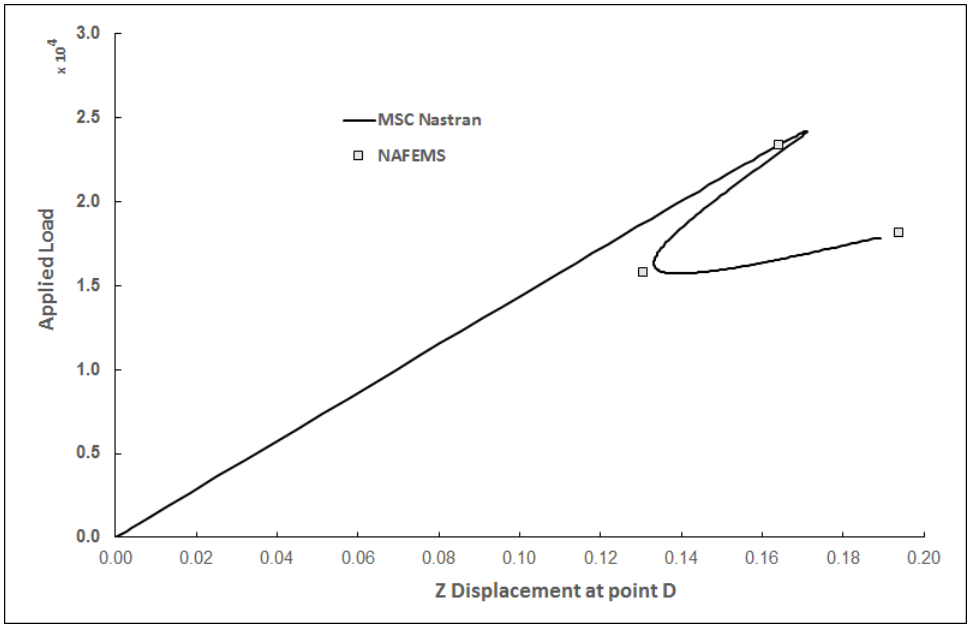


Figure 6.78-1 Buckling of flat plate

Model File Name and Location

Model File Name	Description	Location
`nafem_3dnlg10.dat` `nafem_3dnlg10.impf`	3DNLG-10: Elastic-Plastic Behavior of a Stiffened Cylindrical Panel under Compressive End Load	\verifman\nafems

Results



Conclusion

The results from MSC Nastran are compared with NAFEMS results at loads 23355, 15785 and 18122 in the above figure. As shown in the following table, the load at limit points 1 and 2 from MSC Nastran show the deviation of less than 2.0% compared to the NAFEMS solution.

	NAFEMS Results		MSC Nastran Results		
	P	UZ at point D	P	UZ at point D	% Error
Limit Point 1	23742.0	0.1679	24208.8	0.1710	1.97%
Limit Point 2	15456.0	0.1377	15715.5	0.1390	1.68%