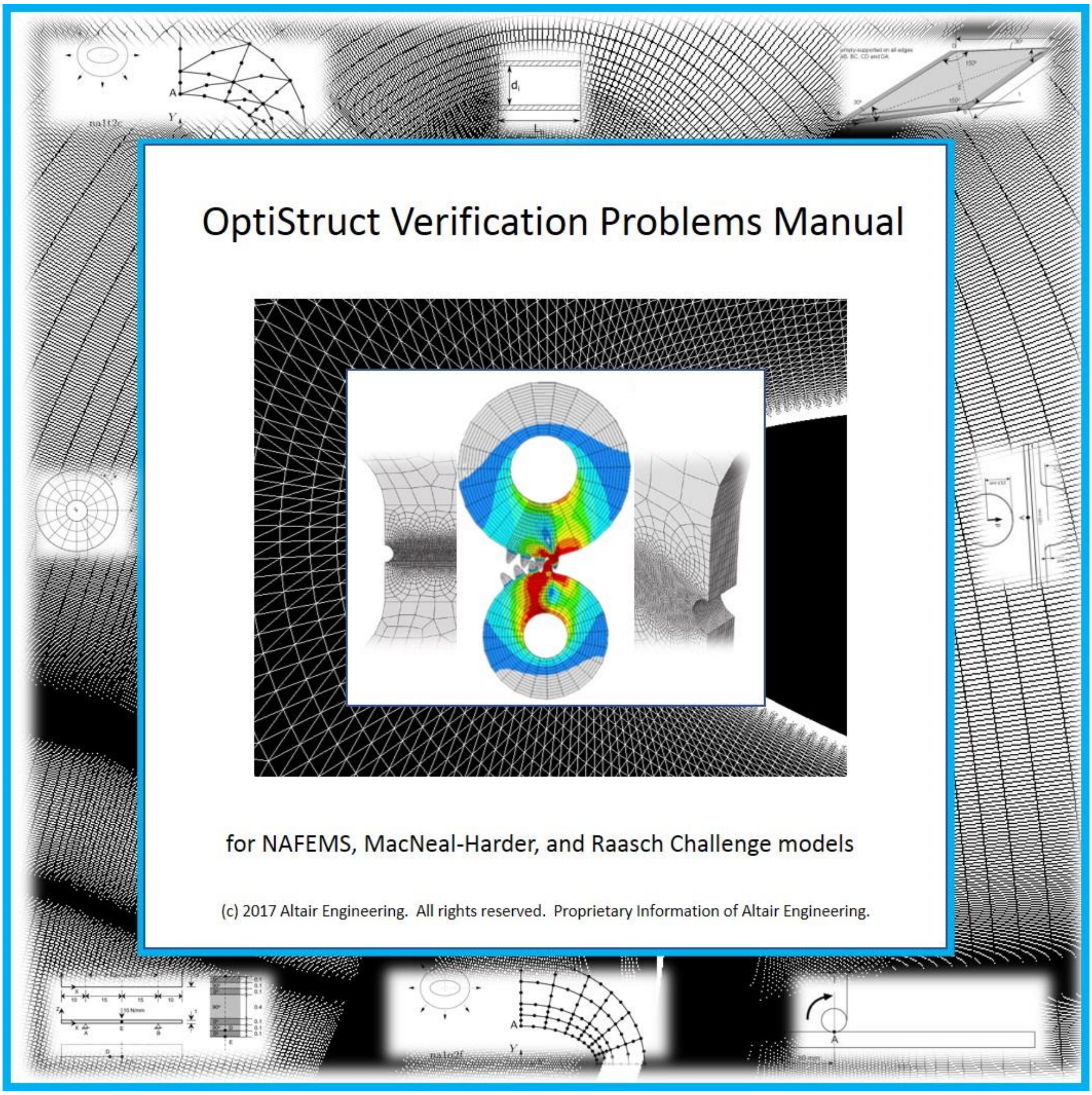
Altair



OptiStruct 2017.2.2

Proprietary Information of Altair Engineering. All Rights Reserved.

Intellectual Property Rights Notice:
Copyrights, Trademarks, Trade Secrets, Patents & Third Party Software Licenses

Altair® HyperWorks® v. 2017.2 A Platform for Innovation™ Copyright© 1986-2017 Altair Engineering, Inc. All Rights Reserved.

Special Notice: Pre-release versions of Altair software are provided 'as is', without warranty of any kind. Usage of pre-release versions is strictly limited to non-production purposes.

HyperWorks® products:
HyperMesh® ©1990-2017; HyperCrash® ©2001-2017; OptiStruct® ©1996-2017; RADIOSS® ©1986- 2017; HyperView® ©1999-2017; HyperView Player® ©2001-2017; HyperMath® ©2007-2017; HyperStudy® ©1999-2017; HyperGraph® ©1995-2017; MotionView® ©1993-2017; MotionSolve® ©2002-2017; HyperForm® ©1998-2017; HyperXtrude® ©1999- 2017; Process Manager™ ©2003-2016; Templex™ ©1990-2017; TextView™ ©1996-2017; MediaView™ ©1999-2017; TableView™ ©2013-2017; BatchMesher™ ©2003-2017; HyperWeld® ©2009-2017; HyperMold® ©2009-2017; Manufacturing Solutions™ ©2005-2017; solidThinking Inspire® 2017 ©2009-2017; solidThinking Evolve®2017 ©1993-2017; Durability Director™ ©2009-2017; Suspension Director™ ©2009-2017; AcuSolve® ©1997-2017; AcuConsole® ©2006-2017; SimLab® ©2004-2017; Virtual Wind Tunnel™ ©2012-2017; FEKO® (©1999-2014 Altair Development S.A. (Pty) Ltd.; ©2014-2017 Altair Engineering, Inc.); ConnectMe™ ©2014- 2017; Click2Extrude™ Polymer 2017 ©1996-2017; Click2Extrude™ Metal 2017 ©1996-2017; Click2Form™ 2017 ©1998-2017.

Additional Altair Products:
Multiscale Designer™ ©2011-2017;
Flux v.12.2 ©1983-2017;
InCa3D v.3.1 ©1996-2016;
CDE v.2 ©2012-2016;
Got-It v.3 ©2002-2016;
WinProp v.14.5 ©2000-2017

Altair Packaged Solution Offerings (PSOs):
Automated Reporting Director™ ©2008-2017; GeoMechanics Director ©2011-2017; Impact Simulation Director™ ©2010-2017; Model Mesher Director™ ©2010-2017; Model Verification Director™ ©2013-2017; NVH Director™ ©2010-2017; Squeak and Rattle Director™ ©2012-2017; Virtual Gauge Director™ ©2012-2017; Weight Analytics™ ©2013-2017; Weld Certification Director™ ©2014-2017; Multi-Disciplinary Optimization™ ©2012-2017.

Altair Simulation Cloud Suite:
Simulation Manager™ ©2003-2017; Compute Manager™ ©2003-2017; Display Manager™ ©2003–2017; and Process Manager™ ©2003-2017.

Altair PBS Works™:
Compute Manager™ ©2012-2017; Display Manager™ ©2013-2017; PBS Pro™ ©1994-2017; PBS Professional® ©1994-2017; PBS Application Services™ ©2008-2017; PBS Analytics™ ©2008-2017; PBS Desktop™ ©2008-2012; and e-Compute™ ©2000-2010.

Software products of solidThinking, Inc., a wholly owned subsidiary of Altair Engineering:

solidThinking Inspire® 2017 ©2009-2017; solidThinking Evolve®2017 ©1993-2017; solidThinking Compose® 2017 ©2007-2017, solidThinking Activate® 2017 ©1989-2017, solidThinking Embed® 2017 ©1989-2017, solidThinking Embed® SE 2017 ©1989-2017; Click2Extrude™ Metal 2017 ©1996-2017; Click2Extrude™ Polymer 2017 ©1996-2017; Click2Cast® 4.0 ©2011-2017; Click2Form™ 2017 ©1998- 2017; Envision® 4.0 ©2013-2017.

Altair intellectual property rights are protected under U.S. and international laws and treaties. Additionally, Altair software is protected under patent #6,859,792 and other patents pending. All other marks are the property of their respective owners.

ALTAIR ENGINEERING INC. Proprietary and Confidential. Contains Trade Secret Information.

Not for use or disclosure outside of Altair and its licensed clients. Information contained in Altair software shall not be decompiled, disassembled, “unlocked”, reverse translated, reverse engineered, or publicly displayed or publicly performed in any manner. Usage of the software is only as explicitly permitted in the end user software license agreement. Copyright notice does not imply publication.

Third party software licenses

AcuConsole contains material licensed from Intelligent Light (www.ilight.com) and used by permission.

Software Security Measures:
Altair Engineering Inc. and its subsidiaries and affiliates reserve the right to embed software security mechanisms in the Software for the purpose of detecting the installation and/or use of illegal copies of the Software. The Software may collect and transmit non-proprietary data about those illegal copies. Data collected will not include any customer data created by or used in connection with the Software and will not be provided to any third party, except as may be required by law or legal process or to enforce our rights with respect to the use of any illegal copies of the Software. By using the Software, each user consents to such detection and collection of data, as well as its transmission and use if an illegal copy of the Software is detected. No steps may be taken to avoid or detect the purpose of any such security mechanisms.

Technical Support

Altair provides comprehensive software support via web FAQs, tutorials, training classes, telephone, and e-mail.

Altair Support on the World Wide Web

The Altair web site is a valuable online companion to Altair software. Visit www.altairhyperworks.com for tips and tricks, training course schedules, training/tutorial videos, and other useful information.

Altair Training Classes

Altair training courses provide a hands-on introduction to our products, focusing on overall functionality. Courses are conducted at our main and regional offices or at your facility. If you are interested in training at your facility, please contact your account manager for more details. If you do not know who your account manager is, please send an e-mail to training@altair.com and your account manager will contact you.

Telephone and e-mail

When contacting Altair support, please specify the product and version number you are using along with a detailed description of the problem. Many times, it is very beneficial for the support engineer to know what type of workstation, operating system, RAM, and graphics board you have, so please have that information ready. If you send an e-mail, please specify the workstation type, operating system, RAM, and graphics board information in the e-mail. To contact an Altair support representative, reference the following table or the information available on the HyperWorks website:

http://www.altairhyperworks.com/ClientCenterHWSupportProduct.aspx

Location	Telephone	e-mail
Australia	64.9.413.7981	anzsupport@altair.com
Brazil	55.11.3884.0414	br_support@altair.com
Canada	416.447.6463	support@altairengineering.ca
China	86.400.619.6186	support@altair.com.cn
France	33.1.4133.0992	francesupport@altair.com
Germany	49.7031.6208.22	hwsupport@altair.de
India	91.80.6629.4500 1.800.425.0234 (toll free)	support@india.altair.com
Italy	39.800.905.595	support@altairengineering.it
Japan	81.3.5396.2881	support@altairjp.co.jp
Korea	82.70.4050.9200	support@altair.co.kr
Mexico	55.56.58.68.08	mx-support@altair.com

For questions or comments about this help system, send an email to hwsupport@altair.com. In addition, the following countries have resellers for Altair Engineering: Colombia, Czech Republic, Ecuador, Israel, Russia, Netherlands, Turkey, Poland, Singapore, Vietnam, Indonesia Official offices with resellers: Canada, China, France, Germany, India, Malaysia, Italy, Japan, Korea, Spain, Taiwan, United Kingdom, USA

See www.altair.com for complete contact information.

Proprietary Information of Altair Engineering. All Rights Reserved.

Introduction

OptiStruct is an industry proven, modern structural analysis solver for linear and nonlinear structural problems under static and dynamic loadings. It is the market-leading solution for structural design and optimization. Based on finite-element and multi-body dynamics technology, and through advanced analysis and optimization algorithms, OptiStruct helps designers and engineers rapidly develop innovative, lightweight and structurally efficient designs.

This is the OptiStruct verification problem manual and the verification problems in this manual are primarily intended to verify OptiStruct solutions & methods. Here an attempt has been made to include most significant analysis capabilities of OptiStruct in this manual. Although they are valuable as demonstration problems, the test cases are not presented with step-by-step procedures. Hence users should refer to the online help, user guides and to this manual for complete details.

OptiStruct verification problem manual contains benchmarked problems that provides affirmation results which OptiStruct produces from a benchmark defined by an external body or institution such as NAFEMS.

Each verification problem contains data and information, such as element type and material properties, results, and references. (along with a keyword search)

This Manual contains verification cases for:

- Linear Static Analysis
- Thermo-elastic Analysis
- Free Vibration
- Forced Vibration
- Response Spectrum Analysis
- Composites
- Nonlinear Contacts
- Nonlinear Material
- Nonlinear Geometry

The problems solved in this manual and the method of solution were selected with verification as the primary objective. As some problems, could have been solved more directly or in a manner other than the way presented. Also in some cases the same problem is solved in several different ways to demonstrate and verify other elements or capabilities of OptiStruct solver.

In this version 2017.2.2 of the OptiStruct verification problem manual we have included 33 new problems, bringing the total to 47 verification problems.

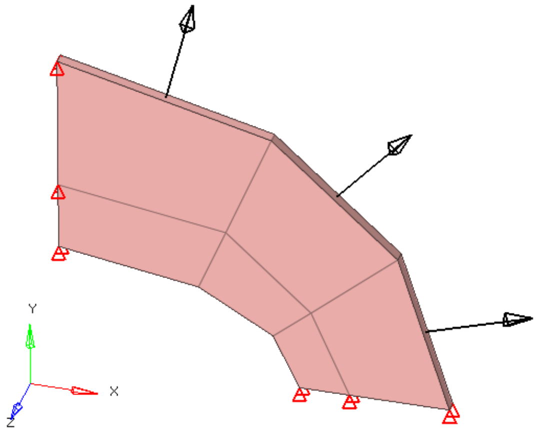
Proprietary Information of Altair Engineering. All Rights Reserved.

Table of Contents

Proprietary Information of Altair Engineering. All Rights Reserved.

Proprietary Information of Altair Engineering. All Rights Reserved.

1. NAFEMS Test No LE1 – Elliptic Membrane



Summary

This is the NAFEMS test problem LE1 for linear static analysis. The model is a thin plate of thickness 0.1m subjected to a uniform pressure. Optistruct examines the direct stress σ_{yy} at the point on inside the ellipse on the x-axis.

Benchmark Model

Second order Hexahedral, Penta, Tetra, Quad and Tria elements are used to create the coarse and fine mesh. A uniform outward pressure of 10 MPa is applied on the outer face. The pressure is converted to force and is applied to the nodes for Quad8 and Tria6 elements.

Material properties

Young's Modulus = 210×10^3 MPa
Poisson's Ratio = 0.3

Linear Static Analysis Results

All results are normalized with the target value (92.7 MPa).

Solid Hexahedral	**Direct stress σ_{yy} at point D (MPa)**	**Normalized with the target value**
Hex8 coarse	63.15	1.467933492
Hex20 coarse	87.46	1.059913103
Hex8 fine	79.8	1.161654135
Hex20 fine	91.01	1.018569388
Solid Wedges		
Penta6 coarse	48.68	1.904272802
Penta15 coarse	95.21	0.973637223
Penta6 fine	67.3	1.377414562
Penta15 fine	94.28	0.983241409

Proprietary Information of Altair Engineering. All Rights Reserved.

Solid Tetrahedral		
Tetra4 coarse	53.34	1.737907762
Tetra10 coarse	95.96	0.966027511
Tetra4 fine	66.56	1.392728365
Tetra10 fine	95.28	0.972921914
Quad Shell		
Quad4 coarse	61.83	1.499272198
Quad8 coarse	86.67	1.069574247
Quad4 fine	79.7	1.163111669
Quad8 fine	91.48	1.013336248
Triangular Shell		
Tria3 coarse	54.06	1.714761376
Tria6 coarse	97.07	0.954980942
Tria3 fine	74.21	1.249157795
Tria6 fine	96.64	0.959230132

File Location

<install_directory>/demos/hwsolvers/optistruct/LE1Hex8C.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Hex20C.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Hex8F.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Hex20F.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Pen6C.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Pen15C.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Pen6F.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Pen15F.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Tet4C.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Tet10C.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Tet4F.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Tet10F.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Quad4C.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Quad8C.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Quad4F.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Quad8F.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Tria3C.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Tria6C.fem

<install_directory>/demos/hwsolvers/optistruct/LE1Tria3F.fem

Proprietary Information of Altair Engineering. All Rights Reserved.

<install_directory>/demos/hwsolvers/optistruct/LE1Tria6F.fem

Reference

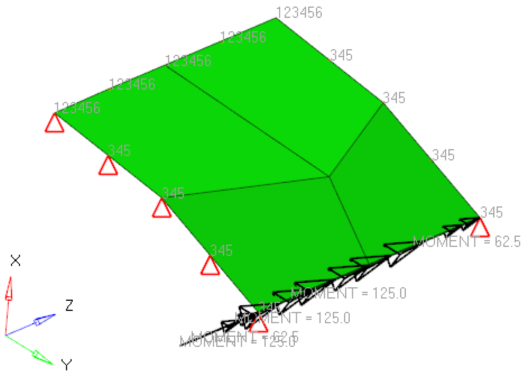
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

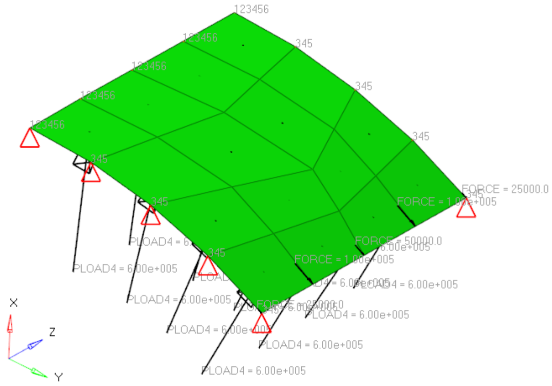
Keywords

GPSTRESS, PLOAD4

Proprietary Information of Altair Engineering. All Rights Reserved.

2. NAFEMS Test No LE2 – Cylindrical Shell Patch Test





Load Case 1 Load Case 2

Summary

This is the NAFEMS test problem LE2 for linear static analysis. Optistruct examines the outer surface tangential stress θ-θ at point E.

Benchmark Model

Quad8 elements are used to create a mesh on the cylindrical patch with 4 elements for the load case 1 and Quad4 elements are used to create a mesh with 16 elements for the load case 2. All the translations and rotations are constrained at edge AB, z-translation and normal rotations are constrained at the edges AD and BC. For Load case 1 a uniform normal edge moment of 1.0kNm/m is applied on the edge DC and for the Load case 2, uniform outward normal pressure of 0.6MPa is applied on the mid surface ABCD and a tangential outward normal pressure of 60.0 MPa is applied on edge DC.

Material properties

Young's Modulus = 210 × 10^3 MPa
Poisson's Ratio = 0.3

Linear Static Analysis Results

All results are normalized with the target value (60.0 MPa).

	Surface Tangential stress θ-θ at point E (MPa)	**Normalized with the target value**
Load case 1		
Quad8	58.03	1.033947958
Load case 2		
Quad4	58.92	1.018329939

File Location

<install_directory>/demos/hwsolvers/optistruct/le2quad8lc1.fem

Proprietary Information of Altair Engineering. All Rights Reserved.

<install_directory>/demos/hwsolvers/optistruct/le2quad4lc2.fem

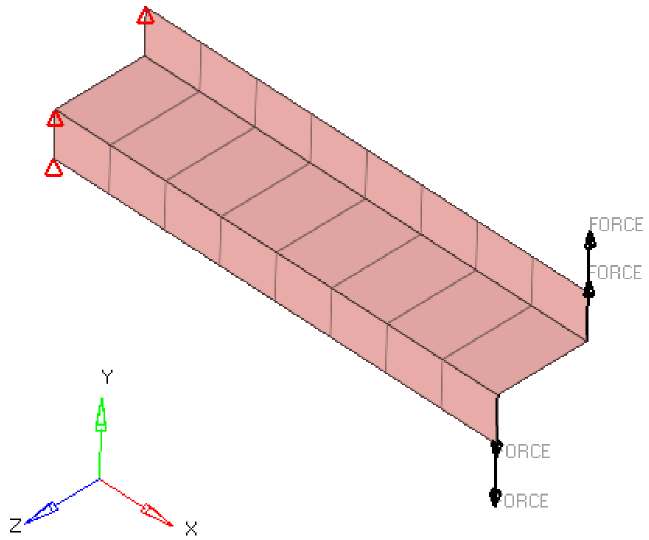
Reference

NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords
CORD2C, MOMENT

Proprietary Information of Altair Engineering. All Rights Reserved.

3. NAFEMS Test No LE5 – Z-Section Cantilever



Summary

This is the NAFEMS test problem LE5 for linear static analysis. Optistruct examines the axial (x-x) stress (compression) at mid-surface, point A.

Benchmark Model

Quad4 and Quad8 elements are used to create a uniform mesh of 8 elements along the length with one element across width of flange. All the displacements at one end are maintained zero, at the other end two uniformly distributed force of 0.6MN each are applied.

Material properties

Young's Modulus = 210×10^3 MPa
Poisson's Ratio = 0.3

Linear Static Analysis Results

All results are normalized with the target value (-108 MPa Compression).

Quadrilateral Shells	Axial stress (x-x) at mid-surface point A (MPa)	Normalized with the target value
Quad4	-112.2	0.962566845
Quad8	-110.9	0.973850316

File Location

<install_directory>/demos/hwsolvers/optistruct/LE5Quad4.fem

<install_directory>/demos/hwsolvers/optistruct/LE5Quad8.fem

Proprietary Information of Altair Engineering. All Rights Reserved.

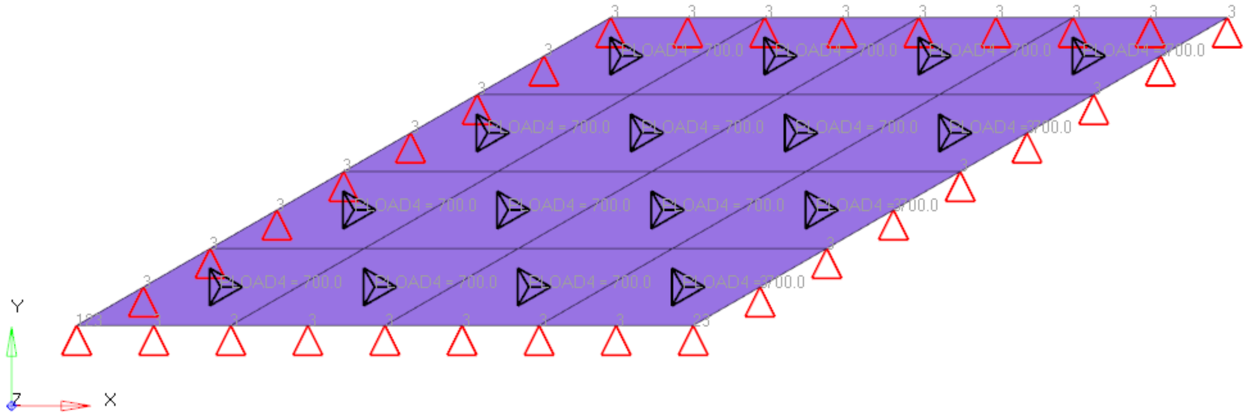
Reference

NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

FORCE

4. NAFEMS Test No LE6 - Skew Plate Normal Pressure



Summary

This is the NAFEMS test problem LE10 for linear static analysis. Optistruct examines the maximum principal stress on the lower surface at the plate center point E.

Benchmark Model

Quad4 and Quad8 elements are used to create a uniform mesh on the skew plate with 4 elements as coarse mesh and 16 elements as fine mesh. The plate is simply supported at all the four edges. A Normal pressure of -0.7 KPa is applied on the face of the plate in the vertical z-direction.

Material properties

Young's Modulus = 210 × 10^3 MPa
Poisson's Ratio = 0.3

Linear Static Analysis Results

All results are normalized with the target value (0.802 MPa).

Quadrilateral Shell	**Maximum principal stress on the lower surface at the plate center point E (MPa)**	**Normalized with the target value**
Quad8 Coarse	0.8294	0.96696407
Quad8 Fine	0.7869	1.019189224

File Location

<install_directory>/demos/hwsolvers/optistruct/le6quad8c.fem

<install_directory>/demos/hwsolvers/optistruct/le6quad8f.fem

Proprietary Information of Altair Engineering. All Rights Reserved.

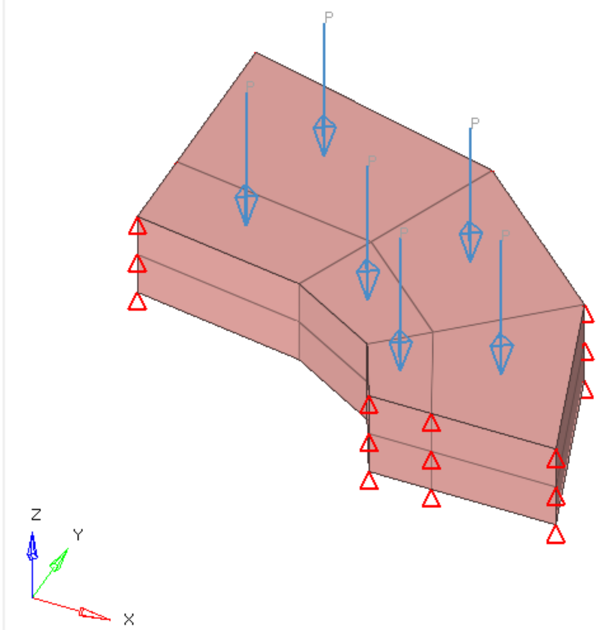
Reference

NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

PLOAD4

5. NAFEMS Test No LE10 - Thick Plate Pressure



Summary

This is the NAFEMS test problem LE10 for linear static analysis. The model is a thick plate subjected to uniform normal pressure of 1MPa on the upper surface of the plate. Optistruct examines the direct stress σ_{yy} at the point D.

Benchmark Model

Second order Hexahedral, Penta and Tetra elements are used to create the coarse and fine mesh. A uniform pressure of 1 MPa is applied on the upper surface of the plate.

Material properties

Young's Modulus = 210×10^3 MPa
Poisson's Ratio = 0.3

Linear Static Analysis Results

All results are normalized with the target value (5.38 MPa).

	Direct stress σ_{yy} at point D (MPa)	Normalized with the target value
Solid Hexahedral		
Hex20 coarse	5.32	1.011278195
Hex20 fine	5.58	0.964157706
Solid Wedges		
Penta15 coarse	4.91	1.095723014

Proprietary Information of Altair Engineering. All Rights Reserved.

Penta15 fine	5.94	0.905723906
Solid Tetrahedral		
Tetra10 coarse	5.741	0.937118969
Tetra10 fine	5.029	1.069795188

File Location

<install_directory>/demos/hwsolvers/optistruct/LE10Hex20C.fem

<install_directory>/demos/hwsolvers/optistruct/LE10Hex20F.fem

<install_directory>/demos/hwsolvers/optistruct/LE10Pyr15C.fem

<install_directory>/demos/hwsolvers/optistruct/LE10Pyr15F.fem

<install_directory>/demos/hwsolvers/optistruct/LE10Tet10C.fem

<install_directory>/demos/hwsolvers/optistruct/LE10Tet10F.fem

Reference

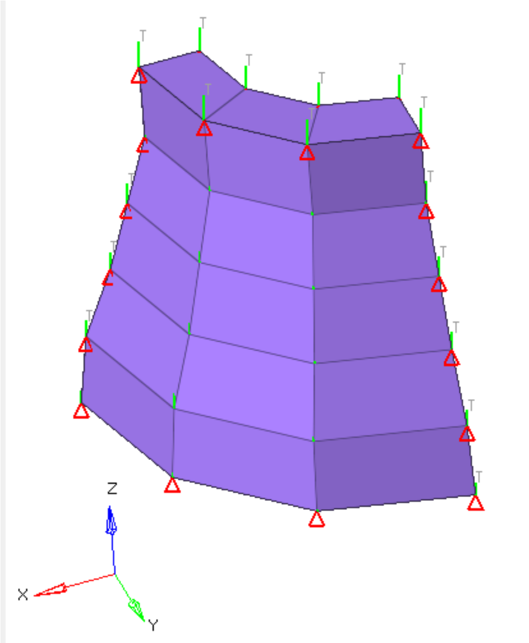
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

GPSTRESS, PLOAD4.

Proprietary Information of Altair Engineering. All Rights Reserved.

6. NAFEMS Test No LE11 – Solid Cylinder / Taper / Sphere - Temperature



Summary

This is the NAFEMS test problem LE11 for linear static analysis. The model is a thick solid cylinder subjected to linear temperature gradient in the radial and axial direction. Optistruct examines the direct stress σ_{yy} at the point A inside the cylinder on the y axis.

Benchmark Model

Second order Hexahedral, Penta and Tetra elements are used to create the coarse and fine mesh. A Linear temperature gradient of T°C = (x2 + y2)1/2 + z is applied in the radial and axial direction from the center of the cylinder. Only one quarter of the cylinder is considered.

Material Properties

MAT1 Isotropic
Young's Modulus = 210 × 10^3 MPa
Poisson's Ratio = 0.3
Coefficient of thermal expansion = 2.3 × 10^{-4}/°C.

Linear Static Analysis Results

All results are normalized with the target value (-105 MPa).

Solid Hexahedral	**Direct stress σ_{zz} at point A (MPa)**	**Normalized with the target value**
Hex20 coarse	-93.21	1.126488574
Hex20 fine	-99.12	1.059322034
Solid Wedges		

Proprietary Information of Altair Engineering. All Rights Reserved.

Penta15 coarse	-100.3	1.046859422
Penta15 fine	-103.7	1.012536162
Solid Tetrahedral		
Tetra10 coarse	-91.97	1.141676634
Tetra10 fine	-98.68	1.064045399

File Location

<install_directory>/demos/hwsolvers/optistruct/LE11Hex20C.fem

<install_directory>/demos/hwsolvers/optistruct/LE11Hex20F.fem

<install_directory>/demos/hwsolvers/optistruct/LE11Pyr15C.fem

<install_directory>/demos/hwsolvers/optistruct/LE11Pyr15F.fem

<install_directory>/demos/hwsolvers/optistruct/LE11Tet10C.fem

<install_directory>/demos/hwsolvers/optistruct/LE11Tet10F.fem

Reference

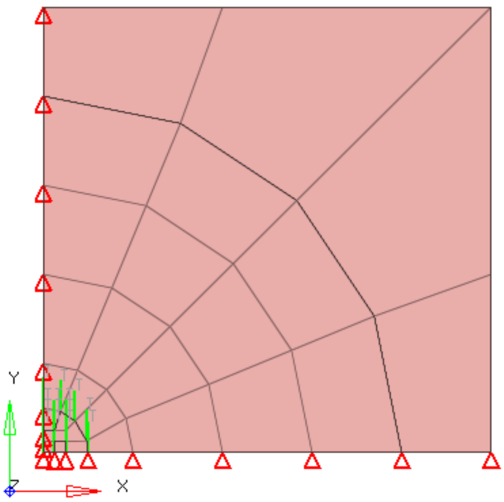
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

GPSTRESS, TEMP,

Proprietary Information of Altair Engineering. All Rights Reserved.

7. NAFEMS Test No T1 – Membrane with Hot-Spot



Summary

This is the NAFEMS test problem T1 for linear thermos elastic analysis. Optistruct examines the direct stress in y direction at a point D outside the hot-spot.

Benchmark Model

Quarter model is considered and Quad4 elements with the specific mesh specifications are used for model building. The hotspot area is maintained at a temperature 100°C.

Material properties

Hotspot area
Young's Modulus = 100×10^3 MPa
Poisson's Ratio = 0.3
Coefficient of thermal expansion = 1×10^{-5}/°C

Rest of the area
Young's Modulus = 100×10^3 MPa
Poisson's Ratio = 0.3
Coefficient of thermal expansion = 0.0

Linear Static Analysis Results

All results are normalized with the target value (50 MPa)

Quadrilateral Shells	Direct stress in y direction at a point D, outside hot-spot (MPa)	Normalized with the target value
Quad4	45.51	1.098659635

Proprietary Information of Altair Engineering. All Rights Reserved.

File Location

<install_directory>/demos/hwsolvers/optistruct/T1Quad4.fem

Reference

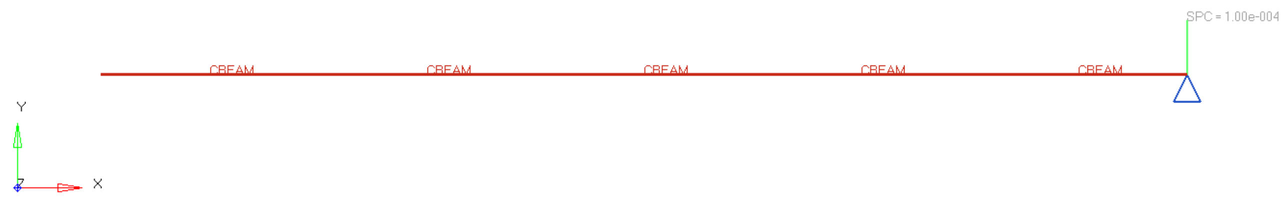
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

TEMP.

Proprietary Information of Altair Engineering. All Rights Reserved.

8. NAFEMS Test No T3 – One Dimensional Transient Heat Transfer



Summary

This is the NAFEMS test problem T3 for transient heat transfer analysis. Optistruct examines the material temperature at point C, 0.08m from point A and the total simulation time is 32 seconds.

Benchmark Model

The 2-noded beam elements, Quad4 elements and Quad8 elements are used to build the model with 5 elements each for the coarse mesh and 10 elements each for the fine mesh. At time t=0, all temperature = zero and at time t>0, at one end temperature is zero and at the other end temperature is 100 sin(πt/40) °C. There is no heat flux perpendicular to the length of the beam.

Material properties

Conductivity = 35.0 W/m°C,
Specific Heat = 440.5 J/kg°C, Density = 7200 kg/m^3.

Linear Static Analysis Results

All results are normalized with the target value (36.60°C)

Beam Element	**Material temperature at point C, x=0.08m, time t=32sec (°C)**	**Normalized with the target value**
CBEAM Coarse	33.6	1.089285714
CBEAM Fine	34.58	1.058415269
Quadrilateral Element		
Quad4 Coarse	33.6	1.089285714
Quad8 Coarse	35.1	1.042735043
Quad4 Fine	34.58	1.058415269
Quad8 Fine	35.19	1.040068201

File Location

<install_directory>/demos/hwsolvers/optistruct/T3CbeamC.fem

<install_directory>/demos/hwsolvers/optistruct/T3CbeamF.fem

Proprietary Information of Altair Engineering. All Rights Reserved.

<install_directory>/demos/hwsolvers/optistruct/T3Quad4C.fem

<install_directory>/demos/hwsolvers/optistruct/T3Quad8C.fem

<install_directory>/demos/hwsolvers/optistruct/T3Quad4F.fem

<install_directory>/demos/hwsolvers/optistruct/T3Quad8F.fem

Reference

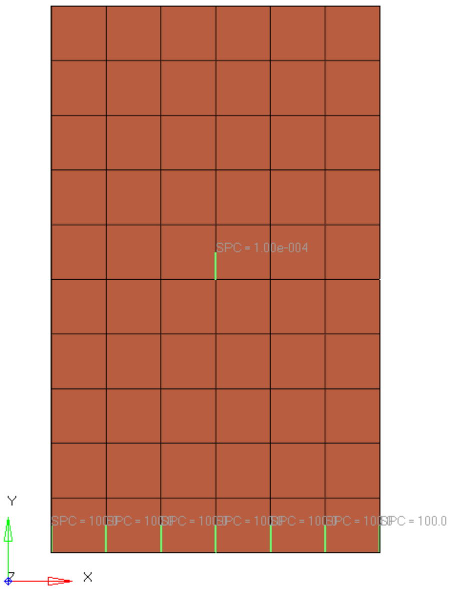
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

THERMAL, DLOAD, TSTEP, TEMPD, SPCD, TABLED1, TLOAD1, MAT4.

Proprietary Information of Altair Engineering. All Rights Reserved.

9. NAFEMS Test No T4 – Two-Dimensional Heat Transfer with Convection



Summary

This is the NAFEMS test problem T4 for steady state heat transfer analysis. The model is having zero internal heat generation and Optistruct examines the temperature at point E.

Benchmark Model

A 10×6 mesh configuration is created with QUAD4, QUAD8, TRIA3 and TRIA6 elements. One edge of the plate is having a prescribed temperature of 100°C, one end insulated and convection to the ambient temperature at the other two edges.

Material properties

Conductivity = 52.0 W/m°C.
Surface convective hear transfer coefficient = 750.0 W/m^2 °C.

Linear Static Analysis Results

All results are normalized with the target value (18.30°C)

Shell Element	**Material temperature at point C, x=0.08m, time t=32sec (°C)**	**Normalized with the target value**
Quad4	17.77	1.029825549
Quad8	17.1	1.070175439
Tria3	17.29	1.058415269
Tria6	16.83	1.087344029

Proprietary Information of Altair Engineering. All Rights Reserved.

File Location

<install_directory>/demos/hwsolvers/optistruct/T4Quad4.fem

<install_directory>/demos/hwsolvers/optistruct/T4Quad8.fem

<install_directory>/demos/hwsolvers/optistruct/T4Tria3.fem

<install_directory>/demos/hwsolvers/optistruct/T4Tria6.fem

Reference

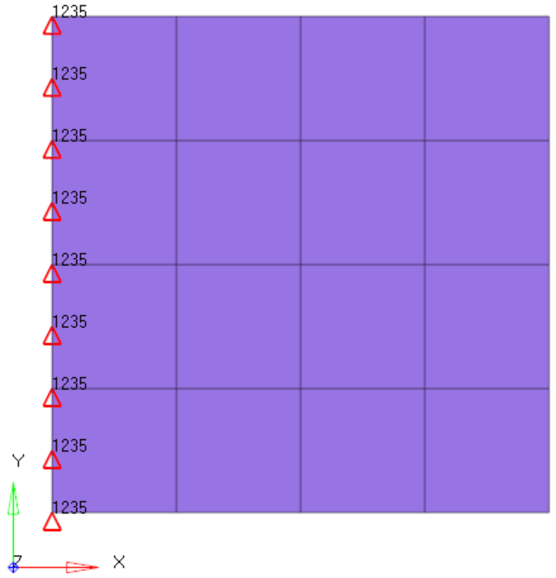
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

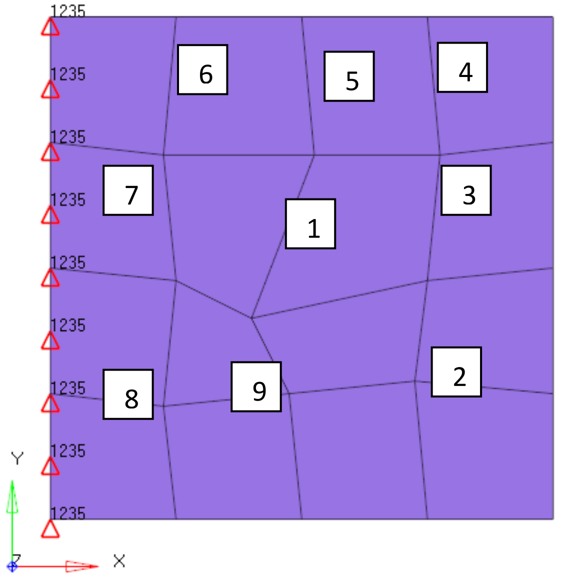
THERMAL, PCONV, CONV, CHBDYE, MAT4.

Proprietary Information of Altair Engineering. All Rights Reserved.

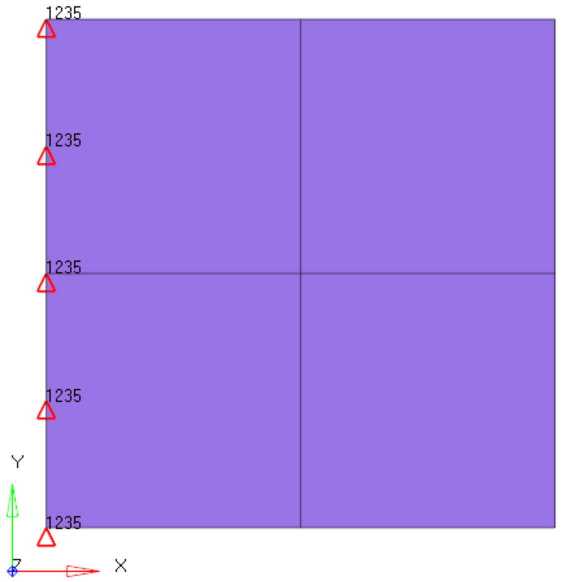
10. NAFEMS Test No FV16 – Cantilevered Thin Square Plate



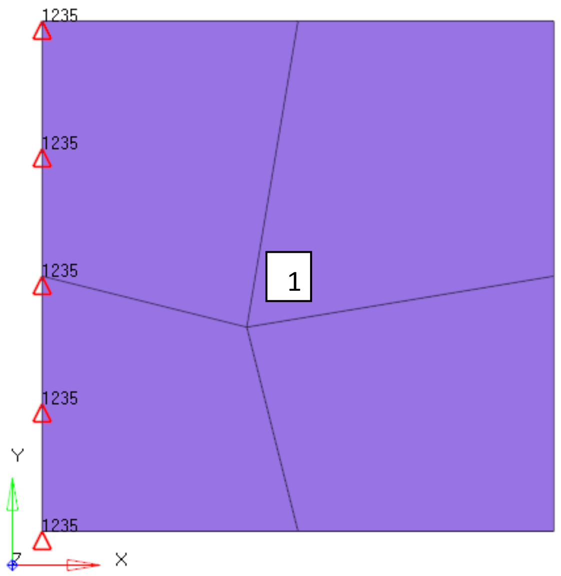
Test 1



Test 2



Test 3



Test 4

Summary

The NAFEMS problem Test No FV16 is a cantilevered thin square plate of 10m×10m dimension. Optistruct is used to investigate the frequencies at different modes for different test cases.

Test 2 Nodal Co-ordinates

Node No.	Co-ordinates(m)

Proprietary Information of Altair Engineering. All Rights Reserved.

	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

Test 4 Nodal Co-ordinates

Node No.	Co-ordinates(m)	
	x	y
1.	4.0	4.0

Benchmark Model

Quad8 elements are used to model different Test cases with specific grid point locations. The thickness of the plate is 0.05m. The x,y,z displacements and y rotations are zero along y-axis.

Material properties

Young's Modulus = 200×10^9 N/m^2
Poisson's Ratio = 0.3
Density = 8000 kg/m^3

Modal Analysis Results

The frequency of each targeted mode is normalized with the NAFEMS closed form solution.

Frequencies (Hz)	Mode 1	Mode 2	Mode 3	Mode 4	Mode 5	Mode 6
Reference Solution	0.421	1.029	2.582	3.306	3.753	6.555
Test 1	0.41	0.99	2.33	3.0	3.44	5.82
Normalized	1.026829	1.041498	1.109107	1.100899	1.090671	1.125708
Test 2	0.41	0.99	2.34	3	3.44	5.83
Normalized	1.026829	1.042553	1.103891	1.103103	1.091306	1.125129
Test 3	0.39	0.92	1.84	2.47	2.93	3.57

Proprietary Information of Altair Engineering. All Rights Reserved.

Normalized	1.085052	1.117264	1.403261	1.33792	1.280887	1.83871
Test 4	0.39	0.91	1.86	2.44	2.82	3.25
Normalized	1.087855	1.130769	1.391164	1.355474	1.333215	2.015063

File Location

<install_directory>/demos/hwsolvers/optistruct/FV16Test1.fem

<install_directory>/demos/hwsolvers/optistruct/FV16Test2.fem

<install_directory>/demos/hwsolvers/optistruct/FV16Test3.fem

<install_directory>/demos/hwsolvers/optistruct/FV16Test4.fem

Reference

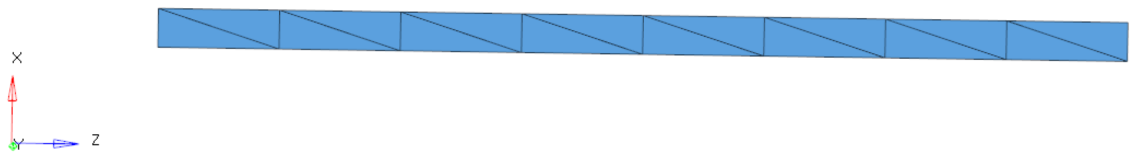
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

EIGRL.

Proprietary Information of Altair Engineering. All Rights Reserved.

11. NAFEMS Test No FV41 – Free Cylinder: Axi-symmetric Vibration



Summary

The NAFEMS problem Test No FV41 is a free cylinder with axi-symmetric vibration. Optistruct is used to investigate the rigid body mode, close eigenvalues, and the coupling between axial, radial and circumferential behavior.

Benchmark Model

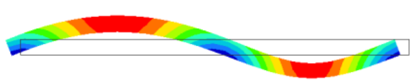
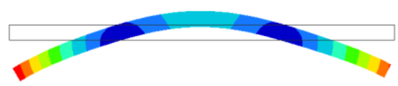
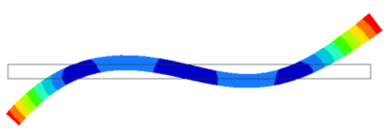
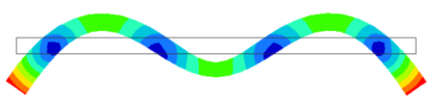
The 1st order and 2nd order triangular axi-symmetric elements (CTAXI) are used to model the cylinder. The cylinder is unsupported.

Material properties

Young's Modulus = 200×10^9 N/m^2
Poisson's Ratio = 0.3
Density = 8000 kg/m^3

Modal Analysis Results

Mode shapes predicted by Optistruct

Mode 2	**Mode 3**	**Mode 4**
		
Modes 5	**Mode 6**	
		

The frequency of each targeted mode is normalized with the NAFEMS closed form solution.

Proprietary Information of Altair Engineering. All Rights Reserved.

	Mode					
	1	**2**	**3**	**4**	**5**	**6**
NAFEMS	RBM	243.53 Hz	377.41 Hz	394.11 Hz	397.72 Hz	405.28 Hz
HOE (8×1)	RBM	243.34 Hz	372.14 Hz	374.18 Hz	378.22 Hz	396.00 Hz
Normalized	1	1.000780801	1.014161337	1.053263135	1.051557295	1.023434343
HOE (16×3)	RBM	243.46 Hz	377.15 Hz	393.2 Hz	394.6 Hz	400.97 Hz
Normalized	1	1.000287522	1.000689381	1.002314344	1.007906741	1.010748934
LOE (8×1)	RBM	242.45 Hz	376.02 Hz	379.86 Hz	389.83 Hz	426.92 Hz
Normalized	1	1.004454527	1.003696612	1.037513821	1.020239592	0.949311346
LOE (16×3)	RBM	243.28 Hz	377.69 Hz	391.42 Hz	395.54 Hz	423.00 Hz
Normalized	1	1.001027622	0.999258651	1.006872413	1.005511453	0.958108747

File Location

<install_directory>/demos/hwsolvers/optistruct/FV41HOEC.fem

<install_directory>/demos/hwsolvers/optistruct/FV41HOEF.fem

<install_directory>/demos/hwsolvers/optistruct/FV41LOEC.fem

<install_directory>/demos/hwsolvers/optistruct/FV41LOEF.fem

Reference

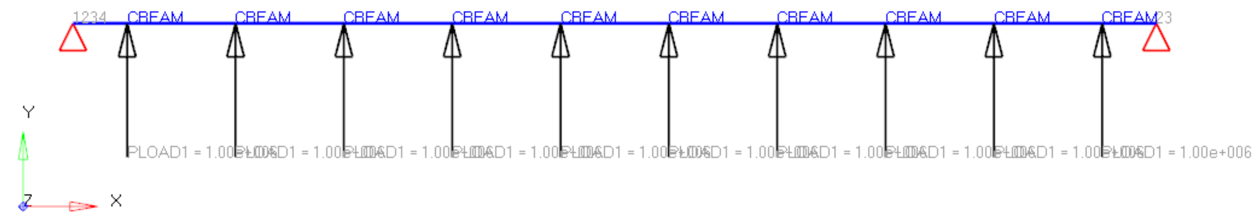
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

CTAXI, PAXI, PAXI.

Proprietary Information of Altair Engineering. All Rights Reserved.

12. NAFEMS Test 5H – Deep Simply-Supported Beam Harmonic Forced Vibration Response



Summary

The NAFEMS problem Test 5H is a simply-supported beam with Harmonic Forced Vibration Response. Optistruct is used to investigate the Peak Displacement in y-direction and extreme fiber bending stress at undamped Natural Frequency (at the mid-span node).

Benchmark Model

Timoshenko beam and Engineer's beam elements are used to model the simply-supported beam which consists of 10 elements. The displacements in x, y, and z direction, as well as the rotation in x direction are fixed at the end A. In addition, the displacements in y and z direction are constrained at end B. A steady state harmonic forced vibration $F=F_0 \sin \omega t$ is induced in the y-direction. ($F_0=10^6$ N/m uniformly distributed, $\omega=2\pi f$, f=0 to 4.16 Hz). For modal analysis solution, a damping ratio of 0.02 is applied in all 16 modes and for direct solution, Rayleigh damping factor $\alpha_1=5.36$ and $\alpha_2=7.46\times10^{-5}$ are given.

Material properties

Young's Modulus = 200×10^9 N/m^2
Poisson's Ratio = 0.3
Density = 8000 kg/m^3

Frequency Response Summary

The frequency of each targeted mode is normalized with the closed form solution.

f* = closed form solution

	Peak displacement (mm)	**Peak Stress (N/mm^2)**	**Frequency (Hz)**
Reference Solution	13.45	241.9	42.65
PBEAML			
Direct Solution	13.42	236.1	43.02
Normalized	1.002235469	1.024565862	0.991399349
Modal Solution	13.56	238.61	43.16

Proprietary Information of Altair Engineering. All Rights Reserved.

Normalized	0.991887906	1.01378819	0.988183503
PBEAM			
Direct Solution	12.27	238.21	45.28
Normalized	1.096169519	1.015490534	0.941916961
Modal Solution	12.3	238.89	45.34
Normalized	1.093495935	1.012599941	0.94067049

File Location

<install_directory>/demos/hwsolvers/optistruct/Test5HPBEAMLD.fem

<install_directory>/demos/hwsolvers/optistruct/Test5HPBEAMLM.fem

<install_directory>/demos/hwsolvers/optistruct/Test5HPBEAMD.fem

<install_directory>/demos/hwsolvers/optistruct/Test5HPBEAMM.fem

Reference

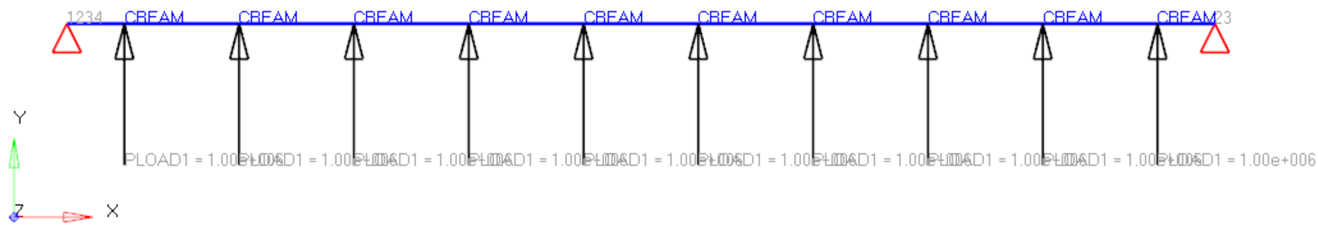
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

CONM2, CBEAM, PBEAM, FREQ2, TABLED1, PLOAD1, RLOAD2, PBEAML, DLOAD.

Proprietary Information of Altair Engineering. All Rights Reserved.

13. NAFEMS Test 5P – Deep Simply-Supported Beam Periodic Forced Vibration Response



Summary

The NAFEMS problem Test 5P is a simply-supported beam with Periodic Forced Vibration Response. Optistruct is used to investigate the Peak Displacement in y-direction and extreme fiber bending stress at the mid-span node.

Benchmark Model

Timoshenko beam and Engineer's beam elements are used to model the simply-supported beam which consists of 10 elements. The displacements in x, y, and z direction, as well as the rotation in x direction are fixed at the end A. In addition, the displacements in y and z direction are constrained at end B. A steady state periodic forced vibration $F=F_0$ (sin ωt-sin 3ωt) is induced in the y-direction. ($F_0=10^6$ N/m uniformly distributed, $\omega=2\pi f$, f=20 Hz). For modal analysis solution, a damping ratio of 0.02 is applied in all 16 modes and for direct solution, Rayleigh damping factor $\alpha_1=5.36$ and $\alpha_2=7.46\times10^{-5}$ are given.

Material properties

Young's Modulus = 200×10^9 N/m^2
Poisson's Ratio = 0.3
Density = 8000 kg/m^3

Frequency Response Summary

The frequency of each targeted mode is normalized with the closed form solution.

f* = closed form solution

	Peak displacement (mm)	**Peak Stress (N/mm^2)**
Reference Solution	0.951	17.10
HOE		
Direct Solution	0.982	17.367
Normalized	0.968431772	0.984626015
Modal Solution	0.982	17.369

Proprietary Information of Altair Engineering. All Rights Reserved.

Normalized	0.968431772	0.984512637
LOE		
Direct Solution	1	19.6
Normalized	0.951	0.87244898
Modal Solution	1	0.951
Normalized	19.6	0.87244898

File Location

<install_directory>/demos/hwsolvers/optistruct/Test5PPBEAMLD.fem

<install_directory>/demos/hwsolvers/optistruct/Test5PPBEAMLM.fem

<install_directory>/demos/hwsolvers/optistruct/Test5PPBEAMD.fem

<install_directory>/demos/hwsolvers/optistruct/Test5PPBEAMM.fem

Reference

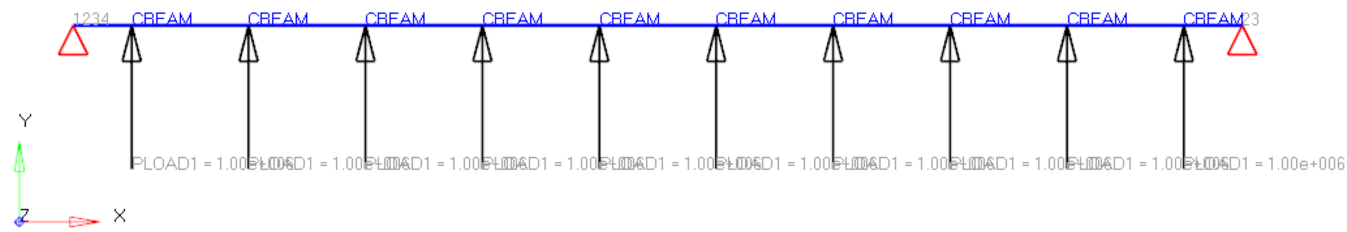
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

CONM2, CBEAM, PBEAM, FREQ, TABLED1, PLOAD1, RLOAD2, PBEAML, DLOAD.

Proprietary Information of Altair Engineering. All Rights Reserved.

14. NAFEMS Test 5T – Deep Simply-Supported Beam Transient Forced Vibration Response



Summary

The NAFEMS problem Test 5T is a simply-supported beam with Transient Forced Vibration Response. Optistruct is used to investigate the Peak Displacement in y-direction, the time at the peak displacement, extreme fiber bending stress at undamped Natural Frequency and the Static displacement at the mid-span node.

Benchmark Model

Timoshenko beam and Engineer's beam elements are used to model the simply-supported beam which consists of 10 elements. The displacements in x, y, and z direction, as well as the rotation in x direction are fixed at the end A. In addition, the displacements in y and z direction are constrained at end B. A suddenly applied step load $F_0=10^6$ N/m is induced in the y-direction. For modal analysis solution, a damping ratio of 0.02 is applied in all 16 modes at a time step of 0.0001 secs and for direct solution, Rayleigh damping factor $\alpha_1=5.36$ and $\alpha_2=7.46\times10^{-5}$ at a time step of 0.0001 secs are given.

Material properties

Young's Modulus = 200×10^9 N/m^2
Poisson's Ratio = 0.3
Density = 8000 kg/m^3

Frequency Response Summary

The frequency of each targeted mode is normalized with the closed form solution.

f* = closed form solution

	Peak displacement (mm)	**Time at Peak displacement (sec)**	**Peak Stress (N/mm^2)**	**Static Displacement (mm)**
Reference Solution	1.043	0.0117	18.76	0.538
HOE				
Direct Solution	1.036	0.0116	18.02	0.533

Proprietary Information of Altair Engineering. All Rights Reserved.

Normalized	1.006756757	1.00862069	1.041065483	1.009380863
Modal Solution	1.03	0.0115	17.99	0.533
Normalized	1.012621359	1.017391304	1.042801556	1.009380863
LOE				
Direct Solution	0.94	0.0109	18.02	0.48
Normalized	1.109574468	1.073394495	1.041065483	1.120833333
Modal Solution	0.939	0.01109	17.92	0.48
Normalized	1.110756124	1.055004509	1.046875	1.120833333

File Location

<install_directory>/demos/hwsolvers/optistruct/Test5TPBEAMLD.fem

<install_directory>/demos/hwsolvers/optistruct/Test5TPBEAMLM.fem

<install_directory>/demos/hwsolvers/optistruct/Test5TPBEAMD.fem

<install_directory>/demos/hwsolvers/optistruct/Test5TPBEAMM.fem

Reference

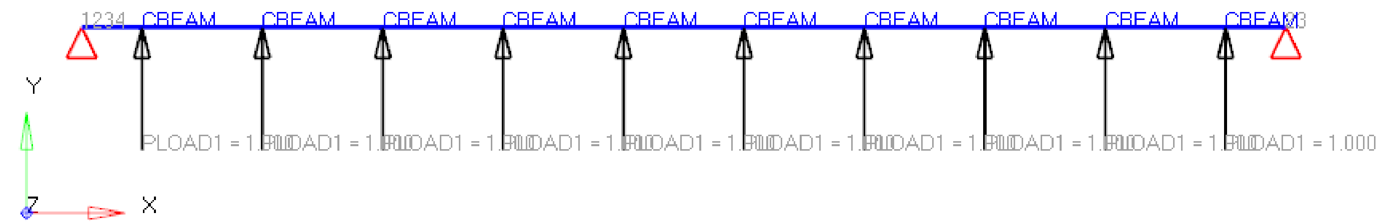
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

CONM2, CBEAM, PBEAM, TABLED1, RLOAD2, PBEAML, DLOAD, TSTEP, DAREA, TLOAD1.

Proprietary Information of Altair Engineering. All Rights Reserved.

15. NAFEMS Test 5R – Deep Simply-Supported Beam Random Forced Vibration Response



Summary

The NAFEMS problem Test 5R is a simply-supported beam with Random Forced Vibration Response. Optistruct is used to investigate the Peak Displacement in y-direction and extreme fiber bending stress at undamped Natural Frequency (at the mid-span node).

Benchmark Model

Timoshenko beam and Engineer's beam elements are used to model the simply-supported beam which consists of 10 elements. The displacements in x, y, and z direction, as well as the rotation in x direction are fixed at the end A. In addition, the displacements in y and z direction are constrained at end B. A steady state random forcing with uniform power spectral density (of force) PSD= $(10^6 \text{ N/m})^2$/Hz is induced in the y-direction. For modal analysis solution, a damping ratio of 0.02 is applied in all 16 modes and for direct solution, Rayleigh damping factor $\alpha_1=5.36$ and $\alpha_2=7.46\times10^{-5}$ are given.

Material properties

Young's Modulus = 200×10^9 N/m^2
Poisson's Ratio = 0.3
Density = 8000 kg/m^3

Frequency Response Summary

The frequency of each targeted mode is normalized with the closed form solution.

f* = closed form solution

	Peak displacement (mm)	**Peak Stress (N/mm^2)**	**Frequency (Hz)**
Reference Solution	180.90	58516	42.65
HOE			
Direct Solution	184.03	56999.67	43.14
Normalized	0.982991903	1.026602435	0.988641632
Modal Solution	183.93	56973.24	43.16

Proprietary Information of Altair Engineering. All Rights Reserved.

Normalized	0.983526342	1.027078678	0.988183503
LOE			
Direct Solution	150.9	56917.35	45.34
Normalized	1.198807157	1.028087218	0.94067049
Modal Solution	151.4	57105.24	45.34
Normalized	1.194848085	1.024704563	0.94067049

File Location

<install_directory>/demos/hwsolvers/optistruct/Test5RPBEAMLD.fem

<install_directory>/demos/hwsolvers/optistruct/Test5RPBEAMLM.fem

<install_directory>/demos/hwsolvers/optistruct/Test5RPBEAMD.fem

<install_directory>/demos/hwsolvers/optistruct/Test5RPBEAMM.fem

Reference

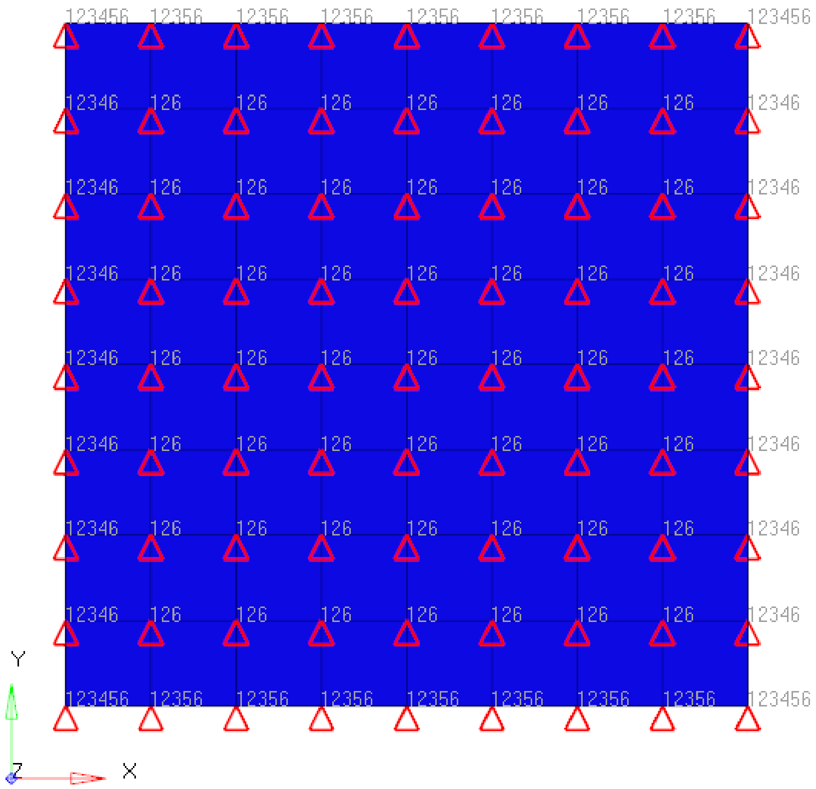
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

CONM2, CBEAM, PBEAM, TABLED1, RLOAD2, PBEAML, PLOAD1, TSTEP, RANDPS, FREQ3, TABDMP1, TABRND1.

Proprietary Information of Altair Engineering. All Rights Reserved.

16. NAFEMS Test 13 – Simply-Supported Thin Square Plate



Summary

The NAFEMS problem Test 13 is a simply-supported thin square plate. Optistruct is used to investigate the repeated eigenvalues.

Benchmark Model

The 2nd order and 1st order quad elements are used to model the square plate of thickness 0.05m. The z-rotation and x, y translations are fixed for all the nodes, z-translation is fixed along all four edges, x-rotation is fixed along the edge x=0 and x=10 and y-rotation is fixed along the edge y=0 and y=10.

Material properties

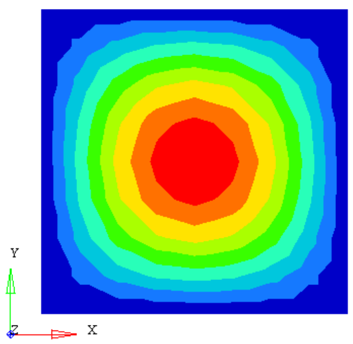
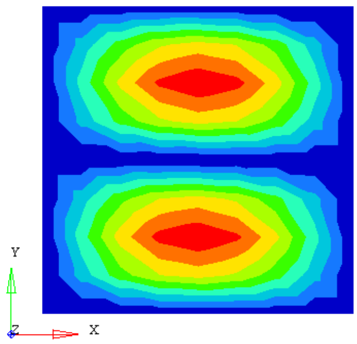
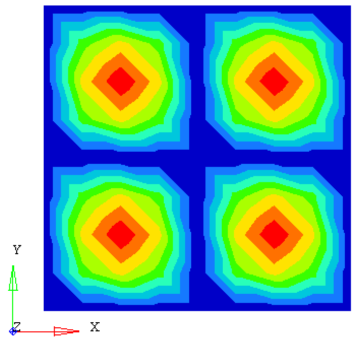
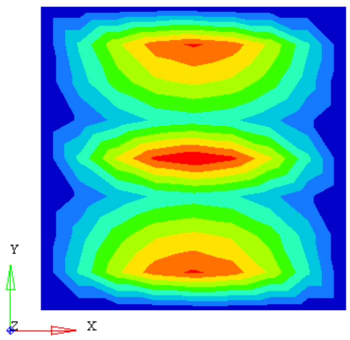
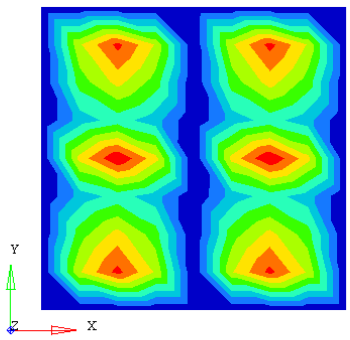
Young's Modulus = 200×10^9 N/m^2
Poisson's Ratio = 0.3
Density = 8000 kg/m^3

Modal Analysis Results

The frequency of each targeted mode is normalized with the closed form solution.

f* = closed form solution

Proprietary Information of Altair Engineering. All Rights Reserved.

Mode 1		Modes 2 & 3		Mode 4	
					
f*	2.377 Hz	f*	5.942 Hz	f*	9.507 Hz
HOE	1.021926053	HOE	1.066977913	HOE	1.076061121
LOE	1.013646055	LOE	1.015552897	LOE	1.053290494
Modes 5 & 6		**Mode 7 & 8**			
					
f*	11.884 Hz	f*	15.449 Hz		
HOE	1.176750173	HOE	1.175007606		
LOE	1.002869198	LOE	1.065889334		

File Location

<install_directory>/demos/hwsolvers/optistruct/Test13HOE.fem

<install_directory>/demos/hwsolvers/optistruct/Test13LOE.fem

Reference

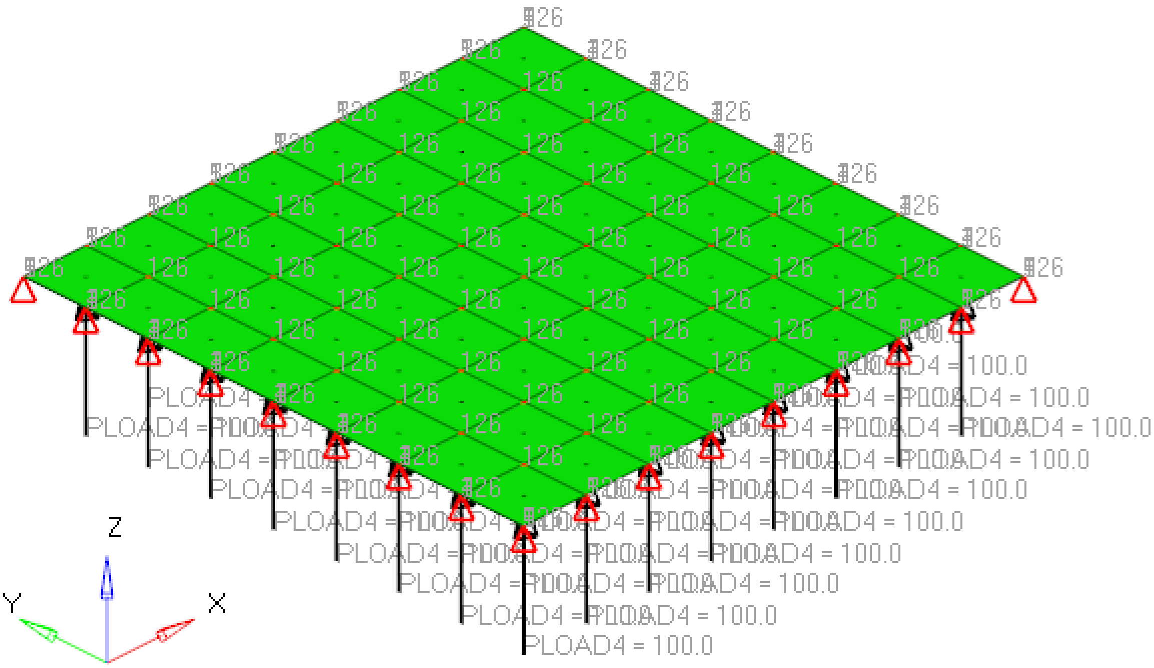
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

EIGRL

Proprietary Information of Altair Engineering. All Rights Reserved.

17. NAFEMS Test 13H – Simply-Supported Thin Square Plate Harmonic Forced Vibration Response



Summary

The NAFEMS problem Test 13H is a simply-supported thin square plate with Harmonic Forced Vibration Response. Optistruct is used to investigate the Peak Displacement in z-direction and extreme fiber bending stress at undamped Natural Frequency (at the center of the plate).

Benchmark Model

The 2nd order and 1st order quad elements are used to model the square plate of thickness 0.05m. The z-rotation and x, y translations are fixed for all the nodes, z-translation is fixed along all four edges, x-rotation is fixed along the edge x=0 and x=10 and y-rotation is fixed along the edge y=0 and y=10. A steady state harmonic forced vibration $F=F_0 \sin \omega t$ is induced in the z-direction. ($F_0=100$ N/m^2 over whole plate, $\omega=2\pi f$, f=0 to 4.16 Hz). For modal analysis solution, a damping ratio of 0.02 is applied in all 16 modes and for direct solution, Rayleigh damping factor $\alpha_1=0.299$ and $\alpha_2=1.339\times10^{-3}$ are given.

Material properties

Young's Modulus = 200 × 10^9 N/m^2
Poisson's Ratio = 0.3
Density = 8000 kg/m^3

Frequency Response Summary

The frequency of each targeted mode is normalized with the closed form solution.

Proprietary Information of Altair Engineering. All Rights Reserved.

f* = closed form solution

	Peak displacement (mm)	**Peak Stress (N/mm^2)**	**Frequency (Hz)**
Reference Solution	45.42	30.03	2.377
HOE			
Direct Solution	47.254	37.57	2.323
Normalized	0.961188471	0.799307958	1.023245803
Modal Solution	47.34	37.64	2.324
Normalized	0.959442332	0.797821467	1.022805508
LOE			
Direct Solution	45.22	30.84	2.349
Normalized	1.004422822	0.973735409	1.011919966
Modal Solution	45.45	30.98	2.345
Normalized	0.999339934	0.969335055	1.013646055

File Location

<install_directory>/demos/hwsolvers/optistruct/Test13HHOED.fem

<install_directory>/demos/hwsolvers/optistruct/Test13HHOEM.fem

<install_directory>/demos/hwsolvers/optistruct/Test13HLOED.fem

<install_directory>/demos/hwsolvers/optistruct/Test13HLOEM.fem

Reference

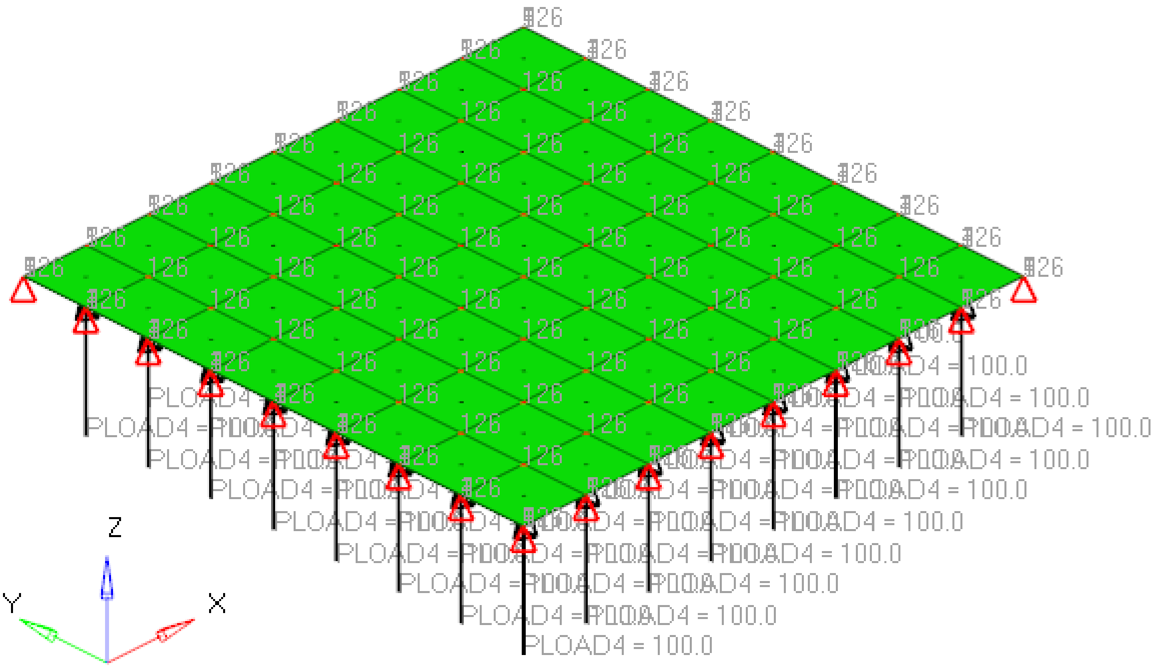
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

DLOAD, TABDMP1, SPCADD, FREQ2, TABLED1, RLOAD2, PLOAD4, EIGRL.

Proprietary Information of Altair Engineering. All Rights Reserved.

18. NAFEMS Test 13P – Simply-Supported Thin Square Plate Periodic Forced Vibration Response



Summary

The NAFEMS problem Test 13P is a simply-supported thin square plate with Periodic Forced Vibration Response. Optistruct is used to investigate the Peak Displacement in z-direction and extreme fiber bending stress at the center of the plate.

Benchmark Model

The 2nd order and 1st order quad elements are used to model the square plate of thickness 0.05m. The z-rotation and x, y translations are fixed for all the nodes, z-translation is fixed along all four edges, x-rotation is fixed along the edge x=0 and x=10 and y-rotation is fixed along the edge y=0 and y=10. A steady state harmonic forced vibration $F=F_0$ (sin ωt-sin 3ωt) is induced in the z-direction. (F_0=100 N/m^2 over whole plate, ω=2πf, f=1.2 Hz). For modal analysis solution, a damping ratio of 0.02 is applied in all 16 modes and for direct solution, Rayleigh damping factor α_1=0.299 and α_2=1.339×10^{-3} are given.

Material properties

Young’s Modulus = 200 × 10^9 N/m^2
Poisson’s Ratio = 0.3
Density = 8000 kg/m^3

Frequency Response Summary

The frequency of each targeted mode is normalized with the closed form solution.

Proprietary Information of Altair Engineering. All Rights Reserved.

f* = closed form solution

	Peak displacement (mm)	**Peak Stress (N/mm^2)**
Reference Solution	2.863	2.018
HOE		
Direct Solution	2.928	2.418
Normalized	0.977800546	0.834574028
Modal Solution	2.929	2.426
Normalized	0.977466712	0.831821929
LOE		
Direct Solution	2.825	1.956
Normalized	1.013451327	1.031697342
Modal Solution	2.826	1.961
Normalized	1.013092711	1.029066803

File Location

<install_directory>/demos/hwsolvers/optistruct/Test13PHOED.fem

<install_directory>/demos/hwsolvers/optistruct/Test13PHOEM.fem

<install_directory>/demos/hwsolvers/optistruct/Test13PLOED.fem

<install_directory>/demos/hwsolvers/optistruct/Test13PLOEM.fem

Reference

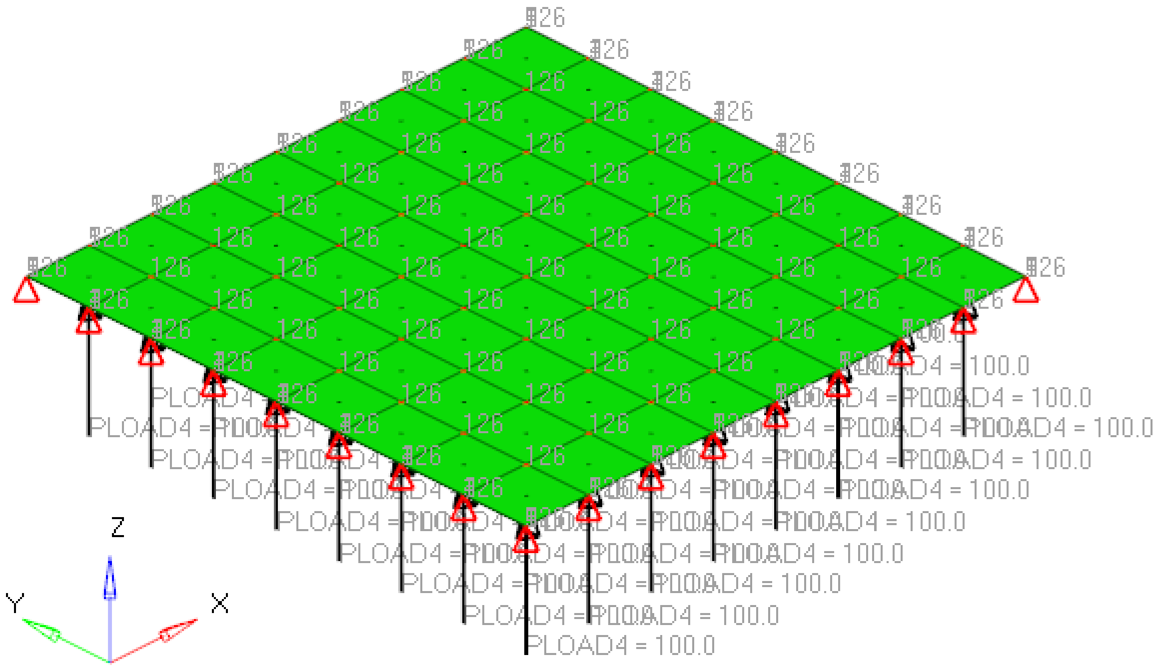
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

DLOAD, TABDMP1, SPCADD, FREQ, TABLED1, RLOAD2, PLOAD4, EIGRL.

Proprietary Information of Altair Engineering. All Rights Reserved.

19. NAFEMS Test 13T – Simply-Supported Thin Square Plate Transient Forced Vibration Response



Summary

The NAFEMS problem Test 13T is a simply-supported thin square plate with Transient Forced Vibration Response. Optistruct is used to investigate the Peak Displacement in z-direction, the time at the peak displacement, extreme fiber bending stress at undamped Natural Frequency and the Static displacement at the center of the plate.

Benchmark Model

The 2nd order and 1st order quad elements are used to model the square plate of thickness 0.05m. The z-rotation and x, y translations are fixed for all the nodes, z-translation is fixed along all four edges, x-rotation is fixed along the edge x=0 and x=10 and y-rotation is fixed along the edge y=0 and y=10. A suddenly applied step load F_0=100 N/m^2 is induced in the z-direction. For modal analysis solution, a damping ratio of 0.02 is applied in all 16 modes at a time step of 0.002 secs and for direct solution, Rayleigh damping factor α_1=0.299 and α_2=1.339×10^{-3} at a time step of 0.002 secs are given.

Material properties

Young's Modulus = 200 × 10^9 N/m^2
Poisson's Ratio = 0.3
Density = 8000 kg/m^3

Frequency Response Summary

The frequency of each targeted mode is normalized with the closed form solution.

Proprietary Information of Altair Engineering. All Rights Reserved.

f* = closed form solution

	Peak displacement (mm)	**Time at Peak displacement (sec)**	**Peak Stress (N/mm^2)**	**Static Displacement (mm)**
Reference Solution	3.523	0.210	2.484	1.817
HOE				
Direct Solution	3.637	0.212	2.784	1.832
Normalized	0.968655485	0.990566038	0.892241379	0.991812227
Modal Solution	3.643	0.210	2.820	1.832
Normalized	0.967060115	1	0.880851064	0.991812227
LOE				
Direct Solution	3.465	0.210	2.244	1.780
Normalized	1.016738817	1	1.106951872	1.020786517
Modal Solution	3.454	0.214	2.203	1.779
Normalized	1.019976838	0.981308411	1.127553336	1.021360315

File Location

<install_directory>/demos/hwsolvers/optistruct/Test13THOED.fem

<install_directory>/demos/hwsolvers/optistruct/Test13THOEM.fem

<install_directory>/demos/hwsolvers/optistruct/Test13TLOED.fem

<install_directory>/demos/hwsolvers/optistruct/Test13TLOEM.fem

Reference

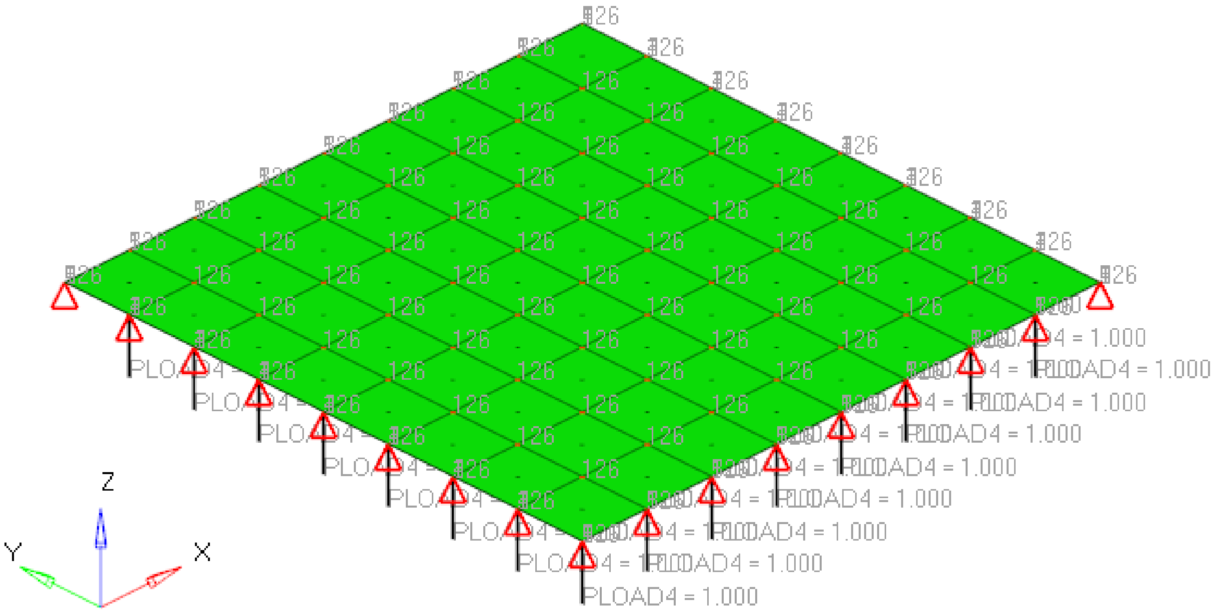
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

DLOAD, TABDMP1, SPCADD, TABLED1, RLOAD2, PLOAD4, EIGRL.

Proprietary Information of Altair Engineering. All Rights Reserved.

20. NAFEMS Test 13R – Simply-Supported Thin Square Plate Harmonic Forced Vibration Response



Summary

The NAFEMS problem Test 13R is a simply-supported thin square plate with Random Forced Vibration Response. Optistruct is used to investigate the Peak Displacement in z-direction and extreme fiber bending stress at undamped Natural Frequency (at the center of the plate).

Benchmark Model

The 2nd order and 1st order quad elements are used to model the square plate of thickness 0.05m. The z-rotation and x, y translations are fixed for all the nodes, z-translation is fixed along all four edges, x-rotation is fixed along the edge x=0 and x=10 and y-rotation is fixed along the edge y=0 and y=10. A steady state random forcing with uniform power spectral density (of force) PSD= $(100\ N/m^2)^2/Hz$ is induced in the z-direction. For modal analysis solution, a damping ratio of 0.02 is applied in all 16 modes and for direct solution, Rayleigh damping factor $\alpha_1=0.299$ and $\alpha_2=1.339\times10^{-3}$ are given.

Material properties

Young's Modulus = $200 \times 10^9\ N/m^2$
Poisson's Ratio = 0.3
Density = $8000\ kg/m^3$

Frequency Response Summary

The frequency of each targeted mode is normalized with the closed form solution.

Proprietary Information of Altair Engineering. All Rights Reserved.

f* = closed form solution

	Peak displacement PSD (mm^2/Hz)	**Peak Stress PSD (($N/mm^2)^2$/Hz)**	**Frequency (Hz)**
Reference Solution	2063.20	1025.44	2.377
HOE			
Direct Solution	2232.98	1411.14	2.322
Normalized	0.923967075	0.726674887	1.023686477
Modal Solution	2241.33	1416.89	2.324
Normalized	0.920524867	0.723725907	1.022805508
LOE			
Direct Solution	2045.23	951.00	2.349
Normalized	1.008786298	1.078275499	1.011919966
Modal Solution	2065.73	960.22	2.345
Normalized	0.998775251	1.067921935	1.013646055

File Location

<install_directory>/demos/hwsolvers/optistruct/Test13RHOED.fem

<install_directory>/demos/hwsolvers/optistruct/Test13RHOEM.fem

<install_directory>/demos/hwsolvers/optistruct/Test13RLOED.fem

<install_directory>/demos/hwsolvers/optistruct/Test13RLOEM.fem

Reference

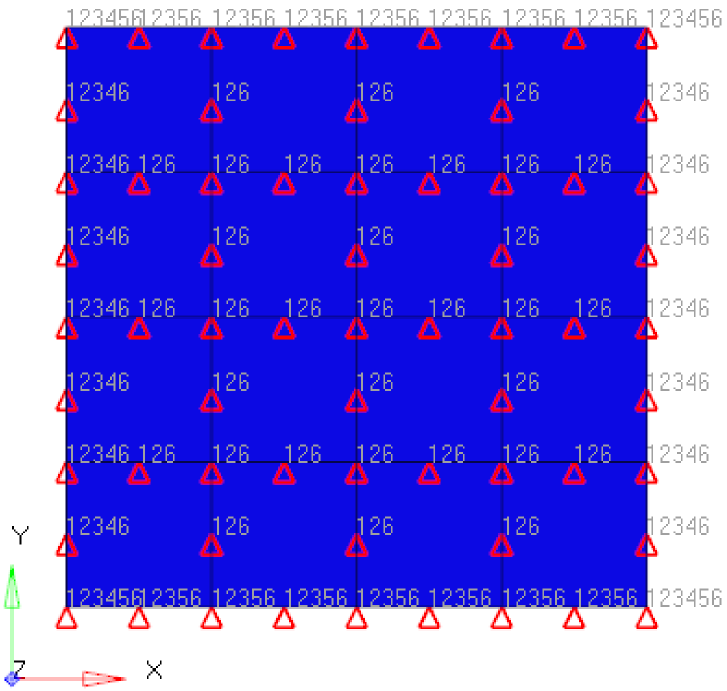
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

DLOAD, TABDMP1, SPCADD, TABLED1, RLOAD2, PLOAD4, EIGRL, FREQ2, RANDPS, TABRND1, FREQ3

Proprietary Information of Altair Engineering. All Rights Reserved.

21. NAFEMS Test 21 – Simply-Supported Thick Square Plate



Summary

The NAFEMS problem Test 21 is a simply-supported thick square plate. Optistruct is used to investigate the repeated eigenvalues and the effect of 'secondary' restrains.

Benchmark Model

The 2nd order and 1st order quad elements are used to model the square plate of thickness 1.0m. The z-rotation and x, y translations are fixed for all the nodes, z-translation is fixed along all four edges, x-rotation is fixed along the edge x=0 and x=10 and y-rotation is fixed along the edge y=0 and y=10.

Material properties

Young's Modulus = 200×10^9 N/m^2
Poisson's Ratio = 0.3
Density = 8000 kg/m^3

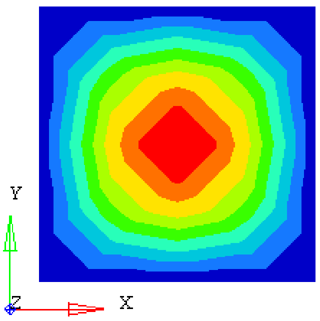
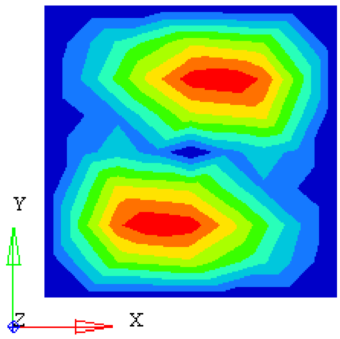
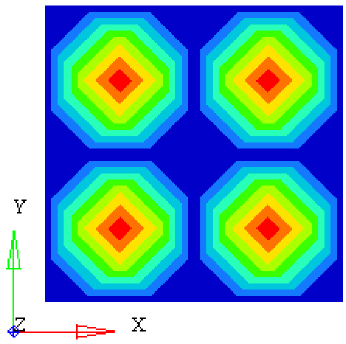
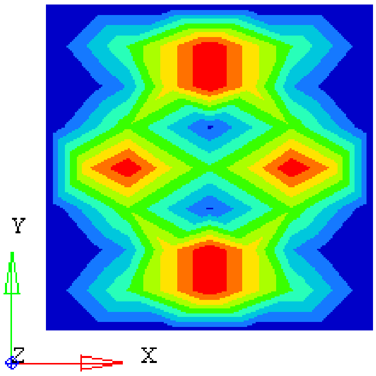
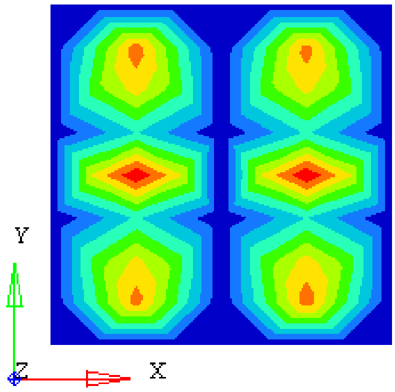
Modal Analysis Results

The frequency of each targeted mode is normalized with the closed form solution.

f* = closed form solution

Mode 1	Modes 2 & 3	Mode 4

Proprietary Information of Altair Engineering. All Rights Reserved.

					
f*	45.897 Hz	f*	109.44 Hz	f*	167.89 Hz
HOE	1.013827837	HOE	1.044863043	HOE	1.046793653
LOE	1.005851414	LOE	1.002363027	LOE	1.034589005
Modes 5 & 6		**Mode 7 & 8**			
					
*	204.51 Hz	f*	256.50 Hz		
HOE	1.125821617	HOE	1.094829757		
LOE	0.993823531	LOE	1.051061511		

File Location

<install_directory>/demos/hwsolvers/optistruct/Test21HOE.fem

<install_directory>/demos/hwsolvers/optistruct/Test21LOE.fem

Reference

NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

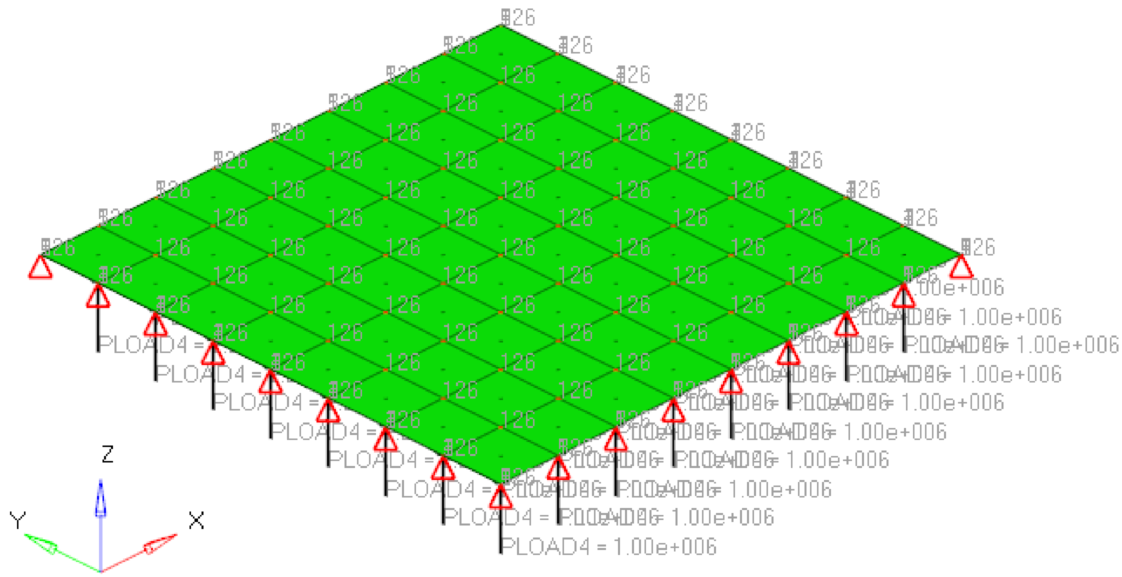
Keywords

EIGRL

Proprietary Information of Altair Engineering. All Rights Reserved.

Altair

22. NAFEMS Test 21H – Simply-Supported Thick Square Plate Harmonic Forced Vibration Response



Summary

The NAFEMS problem Test 21H is a simply-supported thick square plate with Harmonic Forced Vibration Response. Optistruct is used to investigate the Peak Displacement in z-direction and extreme fiber bending stress at undamped Natural Frequency (at the center of the plate).

Benchmark Model

The 2nd order and 1st order quad elements are used to model the square plate of thickness 1.0m. The z-rotation and x, y translations are fixed for all the nodes, z-translation is fixed along all four edges, x-rotation is fixed along the edge x=0 and x=10 and y-rotation is fixed along the edge y=0 and y=10. A steady state harmonic forced vibration $F=F_0 \sin \omega t$ is induced in the z-direction. ($F_0=10^6$ N/m^2 over whole plate, $\omega=2\pi f$, f=0 to 78.17 Hz). For modal analysis solution, a damping ratio of 0.02 is applied in all 16 modes and for direct solution, Rayleigh damping factor $\alpha_1=5.772$ and $\alpha_2=6.926\times10^{-5}$ are given.

Material properties

Young's Modulus = 200×10^9 N/m^2
Poisson's Ratio = 0.3
Density = 8000 kg/m^3

Frequency Response Summary

The frequency of each targeted mode is normalized with the closed form solution.

f* = closed form solution

Proprietary Information of Altair Engineering. All Rights Reserved.

	Peak displacement (mm)	**Peak Stress (N/mm^2)**	**Frequency (Hz)**
Reference Solution	58.33	800.8	45.90
HOE			
Direct Solution	62.633	943.67	45.21
Normalized	0.931298197	0.848601736	1.01526211
Modal Solution	62.67	944.57	45.23
Normalized	0.930748364	0.847793176	1.014813177
LOE			
Direct Solution	60	774.73	45.62
Normalized	0.972166667	1.033650433	1.006137659
Modal Solution	60.05	774.54	45.59
Normalized	0.971357202	1.033903995	1.006799737

File Location

<install_directory>/demos/hwsolvers/optistruct/Test21HHOED.fem

<install_directory>/demos/hwsolvers/optistruct/Test21HHOEM.fem

<install_directory>/demos/hwsolvers/optistruct/Test21HLOED.fem

<install_directory>/demos/hwsolvers/optistruct/Test21HLOEM.fem

Reference

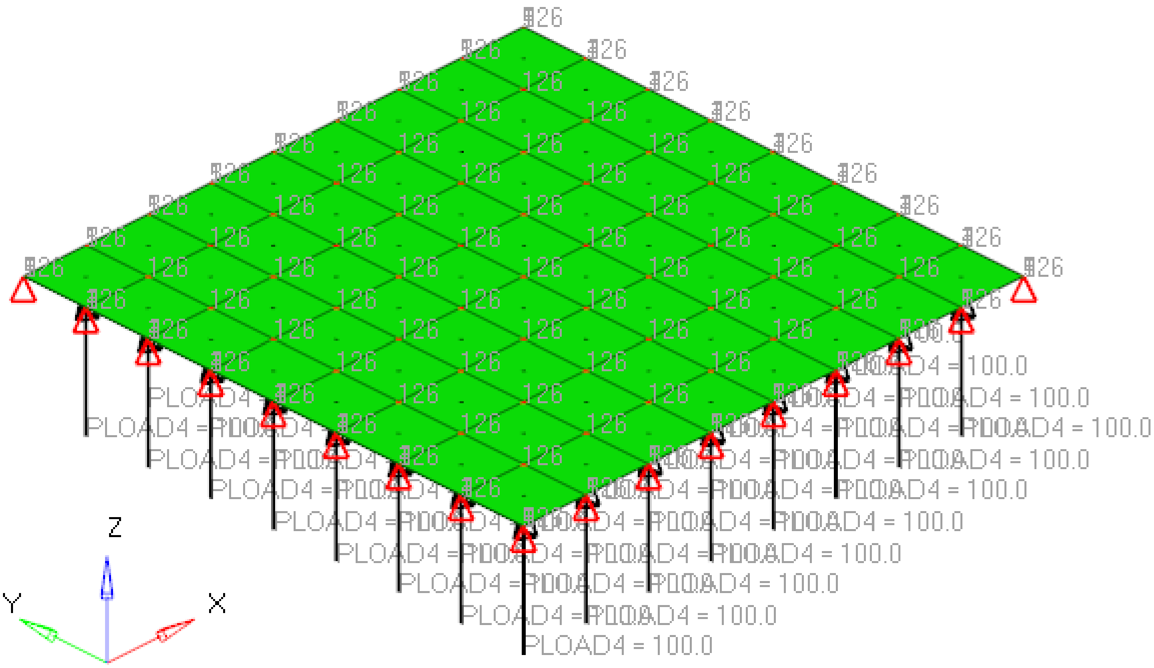
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

DLOAD, TABDMP1, SPCADD, FREQ2, TABLED1, RLOAD2, PLOAD4, EIGRL.

Proprietary Information of Altair Engineering. All Rights Reserved.

23. NAFEMS Test 21P – Simply-Supported Thick Square Plate Periodic Forced Vibration Response



Summary

The NAFEMS problem Test 21P is a simply-supported thin square plate with Periodic Forced Vibration Response. Optistruct is used to investigate the Peak Displacement in z-direction and extreme fiber bending stress at the center of the plate.

Benchmark Model

The 2nd order and 1st order quad elements are used to model the square plate of thickness 0.05m. The z-rotation and x, y translations are fixed for all the nodes, z-translation is fixed along all four edges, x-rotation is fixed along the edge x=0 and x=10 and y-rotation is fixed along the edge y=0 and y=10. A steady state harmonic forced vibration $F=F_0$ (sin ωt-sin 3ωt) is induced in the z-direction. ($F_0=10^6$ N/m^2 over whole plate, ω=2πf, f=20 Hz). For modal analysis solution, a damping ratio of 0.02 is applied in all 16 modes and for direct solution, Rayleigh damping factor $\alpha_1=5.772$ and $\alpha_2=6.929\times10^{-5}$ are given.

Material properties

Young's Modulus = 200×10^9 N/m^2
Poisson's Ratio = 0.3
Density = 8000 kg/m^3

Frequency Response Summary

The frequency of each targeted mode is normalized with the closed form solution.

Proprietary Information of Altair Engineering. All Rights Reserved.

f* = closed form solution

	Peak displacement (mm)	**Peak Stress (N/mm^2)**
Reference Solution	4.929	67.67
HOE		
Direct Solution	5.134	79.26
Normalized	0.960070121	0.853772395
Modal Solution	5.134	79.291
Normalized	0.960070121	0.8534386
LOE		
Direct Solution	5.016	65.48
Normalized	0.982655502	1.033445327
Modal Solution	5.018	65.595
Normalized	0.98226385	1.031633509

File Location

<install_directory>/demos/hwsolvers/optistruct/Test21PHOED.fem

<install_directory>/demos/hwsolvers/optistruct/Test21PHOEM.fem

<install_directory>/demos/hwsolvers/optistruct/Test21PLOED.fem

<install_directory>/demos/hwsolvers/optistruct/Test21PLOEM.fem

Reference

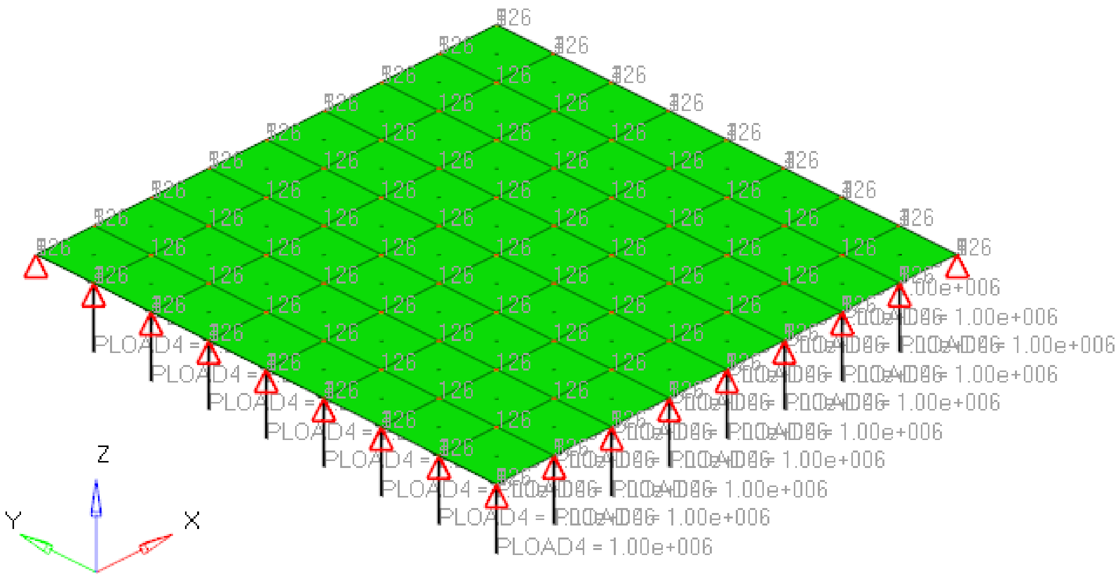
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

DLOAD, TABDMP1, SPCADD, FREQ, TABLED1, RLOAD2, PLOAD4, EIGRL.

Proprietary Information of Altair Engineering. All Rights Reserved.

24. NAFEMS Test 21T – Simply-Supported Thick Square Plate Transient Forced Vibration Response



Summary

The NAFEMS problem Test 21T is a simply-supported thick square plate with Transient Forced Vibration Response. Optistruct is used to investigate the Peak Displacement in z-direction, the time at the peak displacement, extreme fiber bending stress at undamped Natural Frequency and the Static displacement at the center of the plate.

Benchmark Model

The 2nd order and 1st order quad elements are used to model the square plate of thickness 1.0m. The z-rotation and x, y translations are fixed for all the nodes, z-translation is fixed along all four edges, x-rotation is fixed along the edge x=0 and x=10 and y-rotation is fixed along the edge y=0 and y=10. A suddenly applied step load $F_0=10^6$ N/m^2 is induced in the z-direction. For modal analysis solution, a damping ratio of 0.02 is applied in all 16 modes at a time step of 0.0001 secs and for direct solution, Rayleigh damping factor $\alpha_1=5.772$ and $\alpha_2=6.929\times10^{-5}$ at a time step of 0.0001 secs are given.

Material properties

Young's Modulus = 200 × 10^9 N/m^2
Poisson's Ratio = 0.3
Density = 8000 kg/m^3

Frequency Response Summary

The frequency of each targeted mode is normalized with the closed form solution.

f* = closed form solution

Proprietary Information of Altair Engineering. All Rights Reserved.

	Peak displacement (mm)	Time at Peak displacement (sec)	Peak Stress (N/mm^2)	Static Displacement (mm)
Reference Solution	4.524	0.0108	62.11	2.333
HOE				
Direct Solution	4.838	0.011	72.67	2.42
Normalized	0.935097148	0.981818182	0.854685565	0.964049587
Modal Solution	4.870	0.011	75.16	2.42
Normalized	0.928952772	0.981818182	0.82637041	0.964049587
LOE				
Direct Solution	4.604	0.0108	57.98	2.34
Normalized	0.982623805	1	1.071231459	0.997008547
Modal Solution	4.611	0.0107	58.44	2.341
Normalized	0.981132075	1.009345794	1.062799452	0.996582657

File Location

<install_directory>/demos/hwsolvers/optistruct/Test21THOED.fem

<install_directory>/demos/hwsolvers/optistruct/Test21THOEM.fem

<install_directory>/demos/hwsolvers/optistruct/Test21TLOED.fem

<install_directory>/demos/hwsolvers/optistruct/Test21TLOEM.fem

Reference

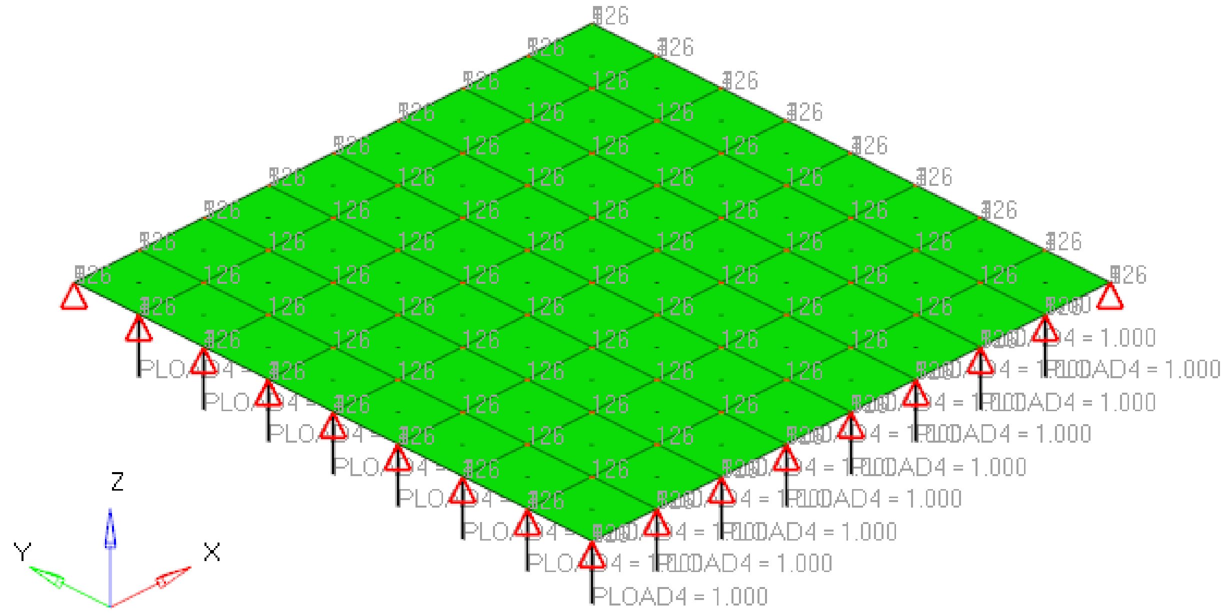
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

DLOAD, TABDMP1, SPCADD, TABLED1, RLOAD2, PLOAD4, EIGRL.

Proprietary Information of Altair Engineering. All Rights Reserved.

25. NAFEMS Test 21R – Simply-Supported Thick Square Plate Random Forced Vibration Response



Summary

The NAFEMS problem Test 21R is a simply-supported thick square plate with Random Forced Vibration Response. Optistruct is used to investigate the Peak Displacement in z-direction and extreme fiber bending stress at undamped Natural Frequency (at the center of the plate).

Benchmark Model

The 2nd order and 1st order quad elements are used to model the square plate of thickness 1.0m. The z-rotation and x, y translations are fixed for all the nodes, z-translation is fixed along all four edges, x-rotation is fixed along the edge x=0 and x=10 and y-rotation is fixed along the edge y=0 and y=10. A steady state random forcing with uniform power spectral density (of force) PSD= $(10^6\ N/m^2)^2/Hz$ is induced in the z-direction. For modal analysis solution, a damping ratio of 0.02 is applied in all 16 modes and for direct solution, Rayleigh damping factor $\alpha_1=5.772$ and $\alpha_2=6.929\times10^{-5}$ are given.

Material properties

Young's Modulus = $200 \times 10^9\ N/m^2$
Poisson's Ratio = 0.3
Density = $8000\ kg/m^3$

Frequency Response Summary

The frequency of each targeted mode is normalized with the closed form solution.

Proprietary Information of Altair Engineering. All Rights Reserved.

f* = closed form solution

	Peak displacement PSD (mm^2/Hz)	**Peak Stress PSD (($N/mm^2)^2$/Hz)**	**Frequency (Hz)**
Reference Solution	3401.81	641200.00	45.90
HOE			
Direct Solution	3929.62	892303.43	45.24
Normalized	0.865684214	0.718589639	1.014588859
Modal Solution	3928.88	892421.36	45.27
Normalized	0.865847264	0.718494681	1.013916501
LOE			
Direct Solution	3607.25	600979.1	45.62
Normalized	0.943048028	1.066925622	1.006137659
Modal Solution	3606.23	600094.2	45.63
Normalized	0.943314764	1.068498912	1.00591716

File Location

<install_directory>/demos/hwsolvers/optistruct/Test21RHOED.fem

<install_directory>/demos/hwsolvers/optistruct/Test21RHOEM.fem

<install_directory>/demos/hwsolvers/optistruct/Test21RLOED.fem

<install_directory>/demos/hwsolvers/optistruct/Test21RLOEM.fem

Reference

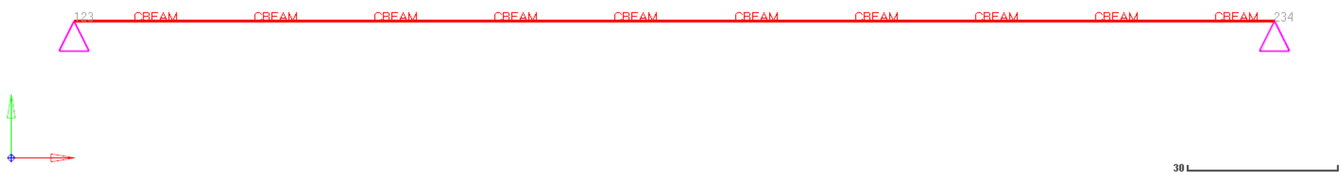
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

DLOAD, TABDMP1, SPCADD, TABLED1, RLOAD2, PLOAD4, EIGRL, FREQ2, RANDPS, TABRND1, FREQ3

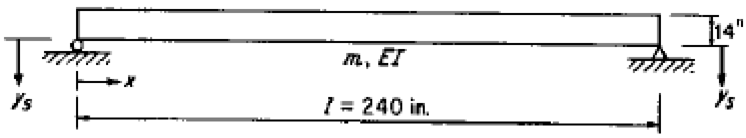
Proprietary Information of Altair Engineering. All Rights Reserved.

26. Response Spectrum Analysis of a Simply Supported Beam



Summary

The problem is a simply-supported beam analyzed by Biggs J M (1964), Introduction to Structural Dynamics, McGraw-Hill, page 256-263, 1964. Optistruct is used to investigate the peak Displacement, Stress and Moment in the mid-span node of the beam. The cross section and dimensions of the beam is shown in the figure. The width of the beam is 1.458 in.



$m = 0.2$ lb-sec^2/in^2
$E = 30 \times 10^6$ psi
$EI = 10^{10}$ lb-in^2

Benchmark Model

Rectangular cross-section CBEAM elements are used to model the simply-supported beam which consists of 10 elements. The displacements in x, y, and z direction are fixed at the end A and the displacements in y and z direction and the x rotations are constrained at end B. A Spectrum input in terms of acceleration, velocity and displacement is applied in the vertical direction at both supports. The damping is specified as zero for this problem. The Response Spectra input definition is mentioned below.

Frequency (Hz)	Displacement (inch)	Velocity (inch/sec)	Acceleration (inch/sec^2)
0	0	0	0
5	0.783	24.599	772.80
6	0.4531	17.083	644.01
6.08	0.4316	16.538	633.66
7	0.2854	12.550	552.01
8	0.2222	11.1659	561.44

Material properties

Proprietary Information of Altair Engineering. All Rights Reserved.

Young's Modulus = 30×10^6 psi
Density = 0.0098 lb-s^2/in^4

Response Spectrum Analysis Results

Model	Spectrum	Displacement (in)	Stress (lb/in^2)	Moment (lb-in)
Biggs		0.56	20100.00	959500.00
CBEAM	Displacement	0.55	19520.38	929716.81
Normalized		1.018	1.0296	1.0320
CBEAM	Velocity	0.55	19519.60	929679.68
Normalized		1.018	1.0297	1.0320
CBEAM	Acceleration	0.55	19517.91	929599.25
Normalized		1.018	1.0298	1.0321

File Location

<install_directory>/demos/hwsolvers/optistruct/RSAdisp.fem

<install_directory>/demos/hwsolvers/optistruct/RSAvel.fem

<install_directory>/demos/hwsolvers/optistruct/RSAaccl.fem

Reference

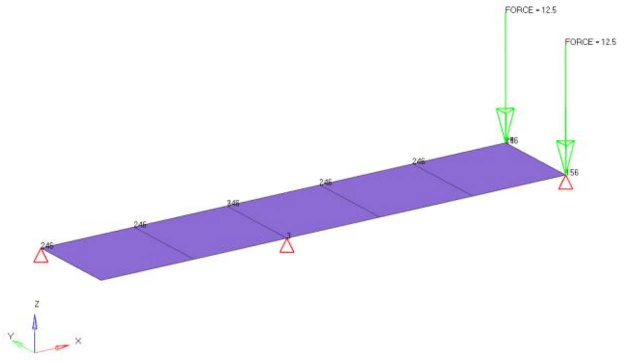
J. M. Biggs, *Introduction to Structural Dynamics*, McGraw-Hill Book Co., Inc., New York, NY, 1964, pg. 262, article 6.4.

Keywords

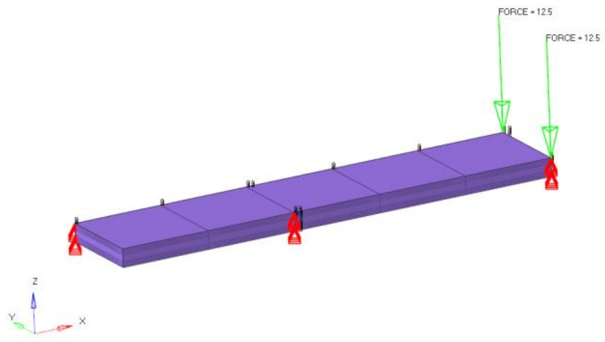
PARAM-COUPMASS, EFFMAS, PBEAML, EIGRL, DTI, TABDMP1, TABLED1, RSPEC.

Proprietary Information of Altair Engineering. All Rights Reserved.

27. NAFEMS Composite Benchmark – Laminated Strip



Quad4



Hexa8

Summary

This is the NAFEMS Composite Benchmark Problem Test No: R0031/1, Laminated Strip for linear static analysis. Optistruct examines the bending stress at E, Interlaminar shear stress at D and the z-deflection at E.

Benchmark Model

Quad4 and Hexa8 elements are used to create one quarter model of the laminated strip simply supported at A. For Quad4 elements Ply laminates are created and for Hexa8 elements PSOLID property is used. A line load of 10N/mm is applied at C.

Material properties

$E_1 = 1.0 \times 10^5$ MPa
$v_{12} = 0.4$
$v_{23} = 0.3$
$E_2 = 5.0 \times 10^3$ MPa
$G_{12} = 3.0 \times 10^3$ MPa
$G_{13} = G_{23} = 2.0 \times 10^3$ MPa
$v_{12}/E_1 = v_{21}/E_2$

Linear Static Analysis Results

	Target	OS Results	Normalized with the target value
Laminated Plate(Quad4)			
Bending Stress at E (MPa)	683.9	570.196	1.19941213
Interlaminar Shear Stress at D (MPa)	-4.1	-4.103	0.99926883
Z deflection at E (mm)	-1.06	-1.045	1.01435407
Stacked Brick(Hex8)			
Bending Stress at E (MPa)	683.9	572.81	1.19393865
Interlaminar Shear Stress at D	-4.1	-3.229	1.26974295

Proprietary Information of Altair Engineering. All Rights Reserved.

(MPa)			
Z deflection at E (mm)	-1.06	-1.028	1.0311284

File Location

<install_directory>/demos/hwsolvers/optistruct/complsq4.fem

<install_directory>/demos/hwsolvers/optistruct/complsh8.fem

Reference

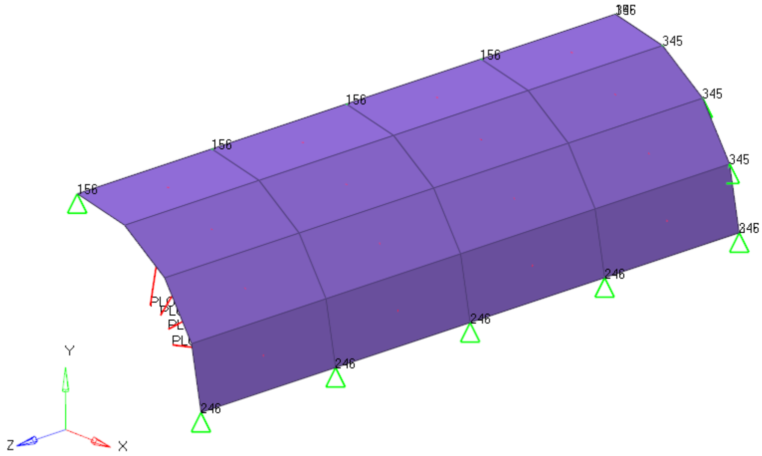
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

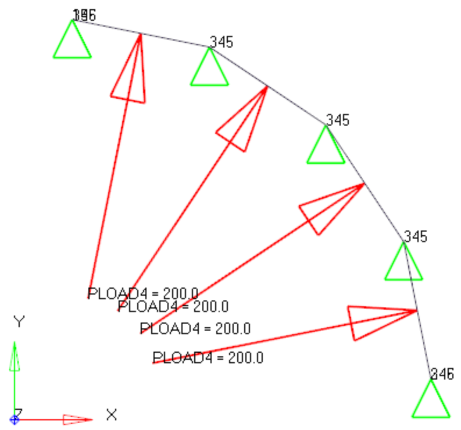
Keywords

PLY, STACK, PCOMPP, MAT8, CORDM, MAT9ORT.

Proprietary Information of Altair Engineering. All Rights Reserved.

28. NAFEMS Composite Benchmark – Wrapped Thick Cylinder





Summary

This is the NAFEMS Composite Benchmark Problem Test No: R0031/2, Wrapped Thick Cylinder for linear static analysis. Optistruct examines the hoop stress in the inner and outer cylinder at different radius.

Benchmark Model

Ply laminates are created using Quad4 elements for one quarter model of the cylinder. For Case 1 an internal pressure of 200MPa is applied and for the Case 2 together with the internal pressure, a temperature rise of 130°C is applied.

Material properties

Inner Cylinder(Isotropic):

$E = 2.1 \times 10^5$ MPa

$v = 0.3$

$\alpha = 2.0 \times 10^{-5}$ °C^{-1}

Outer Cylinder (Circumferentially wound):

$E_1 = 1.3 \times 10^5$ MPa

$v_{12} = 0.25$

$E_2 = 5.0 \times 10^3$ MPa

$\alpha_1 = 3.0 \times 10^{-6}$ °C^{-1}

$\alpha_2 = 2.0 \times 10^{-5}$ °C^{-1}

$G_{12} = 1.0 \times 10^4$ MPa

$G_{33} = 5.0 \times 10^3$ MPa

Linear Static Analysis Results

	Target (MPa)	OS Results (MPa)	Normalized with the target value
Case 1			

Proprietary Information of Altair Engineering. All Rights Reserved.

Hoop stress in inner cylinder at r = 23	1565.3	1659.4	0.94329276
Hoop stress in inner cylinder at r = 25	1429.7	1659.4	0.86157647
Hoop stress in outer cylinder at r = 25	874.7	792.6	1.10358314
Hoop stress in outer cylinder at r = 27	759.1	792.6	0.95773404
Case 2			
Hoop stress in inner cylinder at r = 23	1381.0	1392.05	0.99206207
Hoop stress in inner cylinder at r = 25	1259.6	1392.05	0.90485256
Hoop stress in outer cylinder at r = 25	1056.0	1059.92	0.99630161
Hoop stress in outer cylinder at r = 27	936.1	1059.92	0.88317986

File Location

<install_directory>/demos/hwsolvers/optistruct/compwtcq4c1.fem

<install_directory>/demos/hwsolvers/optistruct/compwtcq4c2.fem

Reference

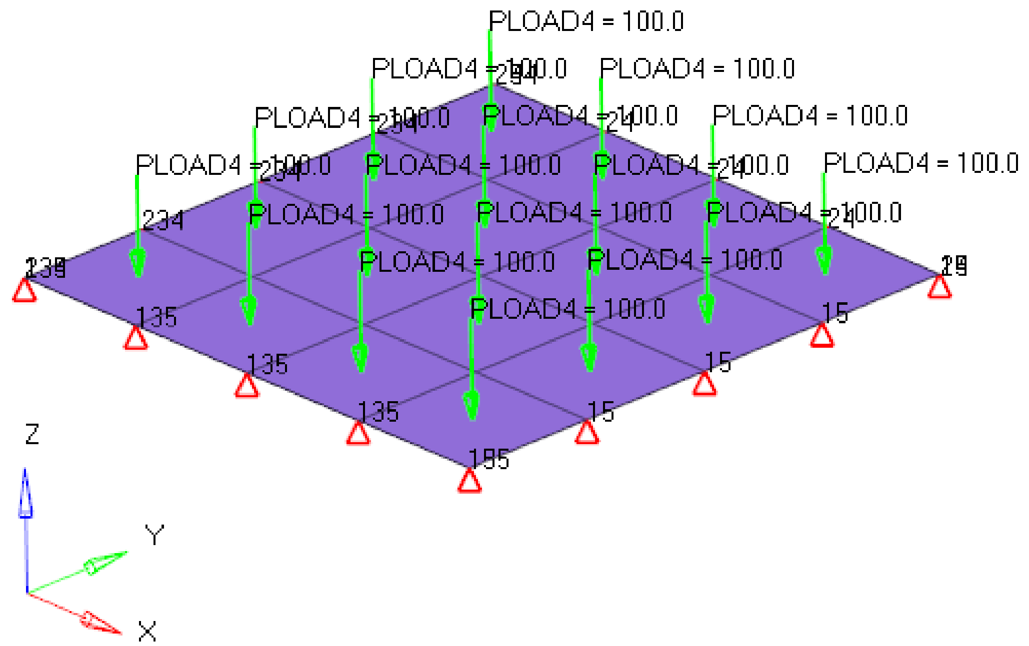
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

PLY, STACK, PCOMPP, MAT8, TEMPD, PLOAD4.

Proprietary Information of Altair Engineering. All Rights Reserved.

29. NAFEMS Composite Benchmark – Sandwich Shell



Summary

This is the NAFEMS Composite Benchmark Problem Test No: R0031/3, Sandwich Shell for linear static analysis. Optistruct examines the z deflection at C, Normal XX stress at C, Normal YY stress at C and Shear stress at E.

Benchmark Model

Quad4 elements are used to create Ply laminates and with SMCORE for the simply supported one quarter model. A uniform normal pressure of 200psi is applied on the plate.

Material properties

Face Sheets:
$E_x = 10.0 \times 10^6$ psi
$v_{xy} = 0.3$
$E_y = 4.0 \times 10^6$ psi
$G_{xy} = 1.875 \times 10^6$ psi

Core:
$E_x = 0.0$
$G_{xz} = 3.0 \times 10^4$ psi
$G_{yz} = 1.2 \times 10^4$ psi

Linear Static Analysis Results

	Target	OS Results	Normalized with the target value
Laminated Plate(PCOMPP)			

Proprietary Information of Altair Engineering. All Rights Reserved.

Z deflection at C (inches)	-0.123	-0.123	1
σ_{xx} at C	34449	34126.09	1.00946226
σ_{yy} at C	13390	13375.11	0.99812263
T_{xy} at E	-5067.5	-5080.6	0.99742156
SMCORE(PCOMPG)			
Z deflection at C (inches)	-0.123	-0.078	1.57692308
σ_{xx} at C	34449	32578.56	1.05741322
σ_{yy} at C	13390	14508.05	0.9201788
T_{xy} at E	-5067.5	-5327.68	0.95116448

File Location

<install_directory>/demos/hwsolvers/optistruct/compssq4.fem

<install_directory>/demos/hwsolvers/optistruct/compssq4smcore.fem

Reference

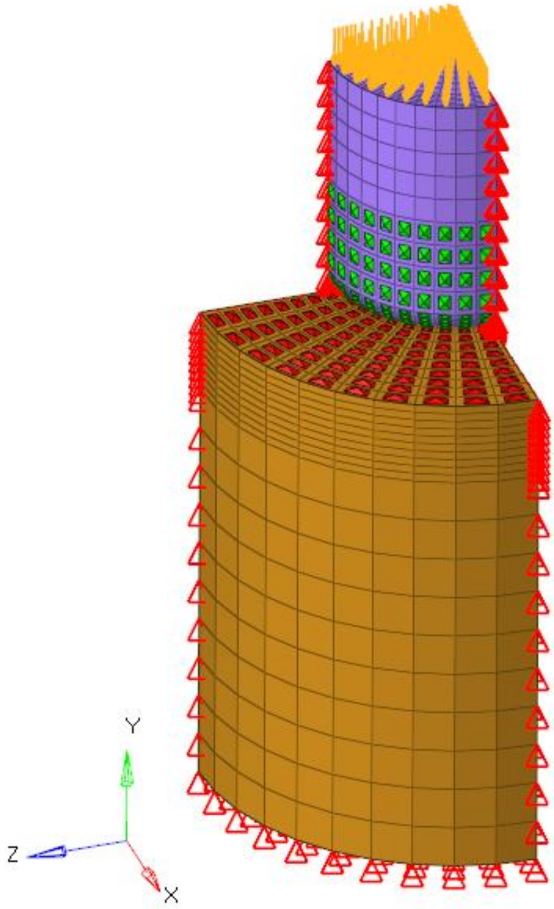
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

PLY, STACK, PCOMPP, MAT8, PLOAD4.

Proprietary Information of Altair Engineering. All Rights Reserved.

30. NAFEMS Contacts Benchmark 2– 3D Punch (Rounded Edges)



Summary

This is the NAFEMS Contact Benchmark - 2, 3D Punch (Rounded Edges) for Quasi-static analysis using Linear elastic material, geometric non-linearity and non-linear boundary conditions. OptiStruct FE results examine the plot of contact pressure, tangential stress against radial distance from the center of contact and relative tangential slip against distance from the center of contact. OptiStruct also examines the 3D contact, stick/slip behavior along the contact plane, compares the linear and quadratic elements and the plasticity.

Benchmark Model

Hexa8 and Hexa20 elements are used to create one quarter model with punch diameter 100mm, punch height 100mm, foundation diameter 200mm, foundation height 200mm and fillet radius at the edge of the punch contact is 10mm. A uniform pressure of 100N/mm^2 is applied at the top surface of the punch. The bottom surface of the foundation is fixed. Two different contact properties are used, one with coefficient of friction 0.0 and the second with coefficient of friction 0.1. the straight edge of the foundation is considered as the master surface and the nodes on the bottom edge of the punch are selected as the slave surface.

Material properties

E_{punch} = 210 kN/mm^2
v_{punch} = 0.3
$E_{foundation}$ = 70 kN/mm^2
$N_{foundation}$ = 0.3

Non-Linear Quasi Static Analysis Results

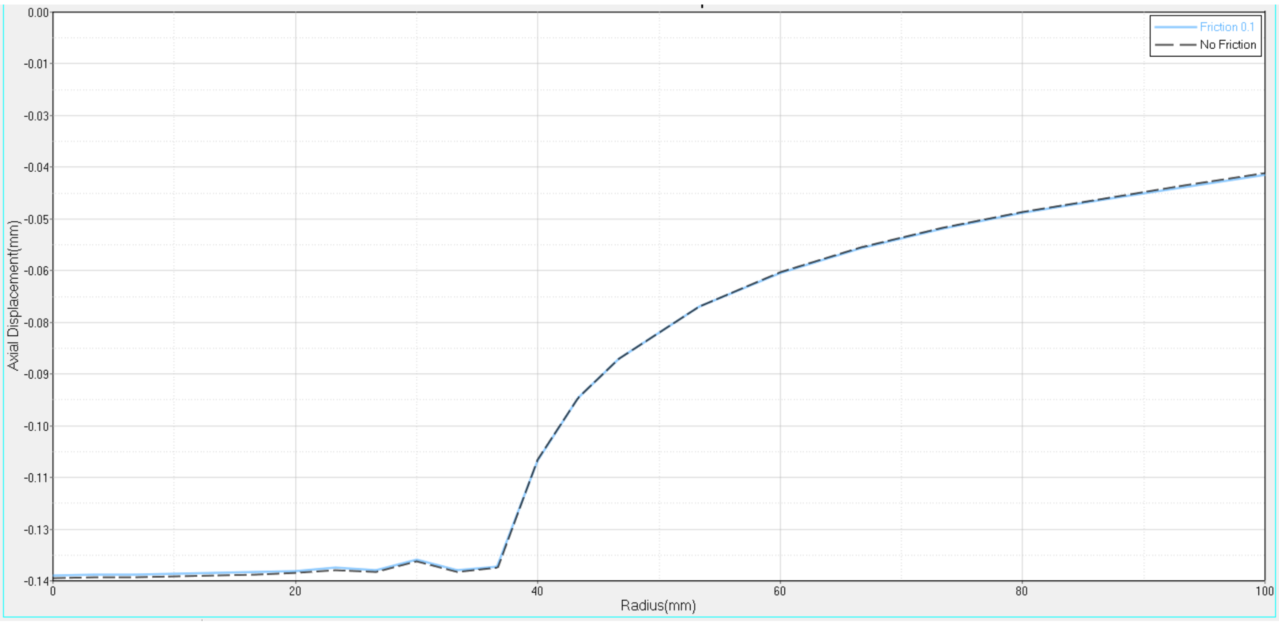
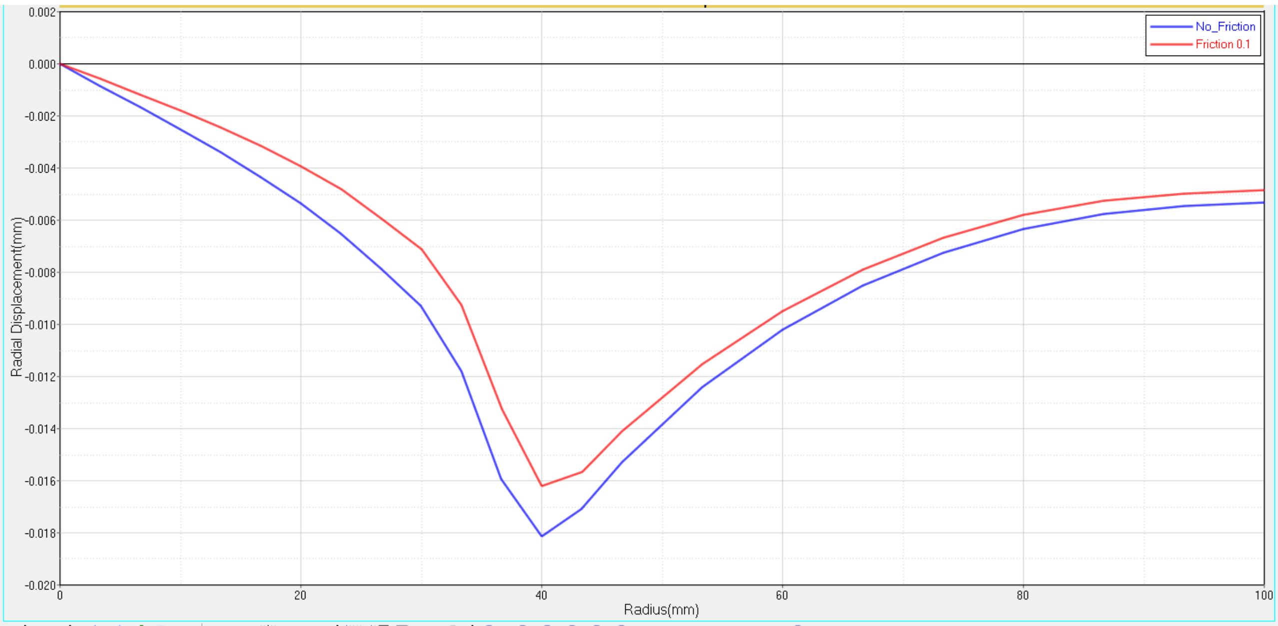


Figure-1:Axial displacement as a function of the radial coordinate (friction coefficient 0.0 and 0.1) obtained with linear elastic elements.



Proprietary Information of Altair Engineering. All Rights Reserved.

Figure-2:Radial displacement as function of the radial coordinate (friction coefficient 0.0 and 0.1) obtained with linear elastic elements.

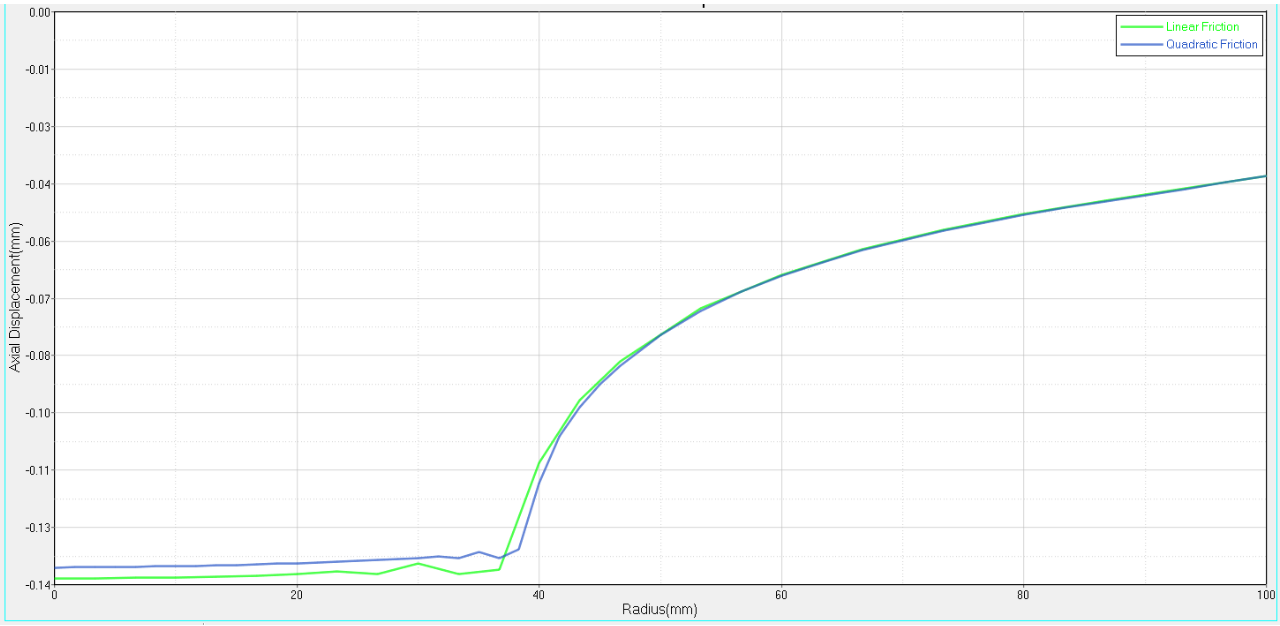


Figure-3:Axial displacement along top surface of foundation(with friction).

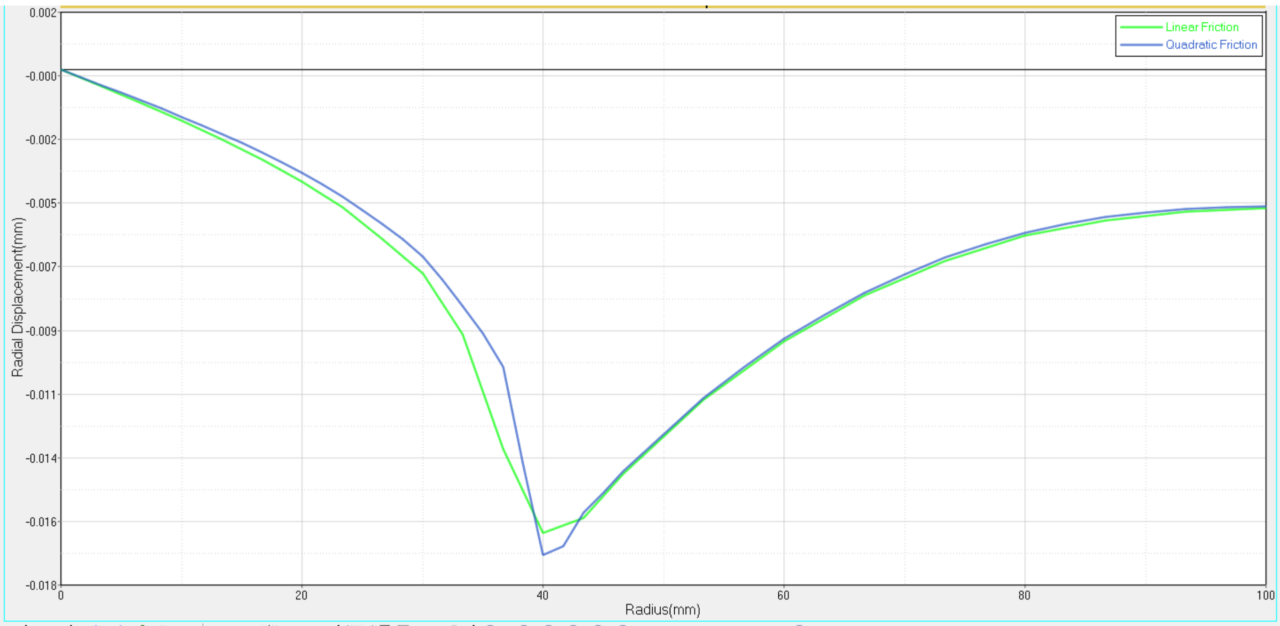


Figure-4:Radial displacement along top surface of foundation(with friction).

Proprietary Information of Altair Engineering. All Rights Reserved.

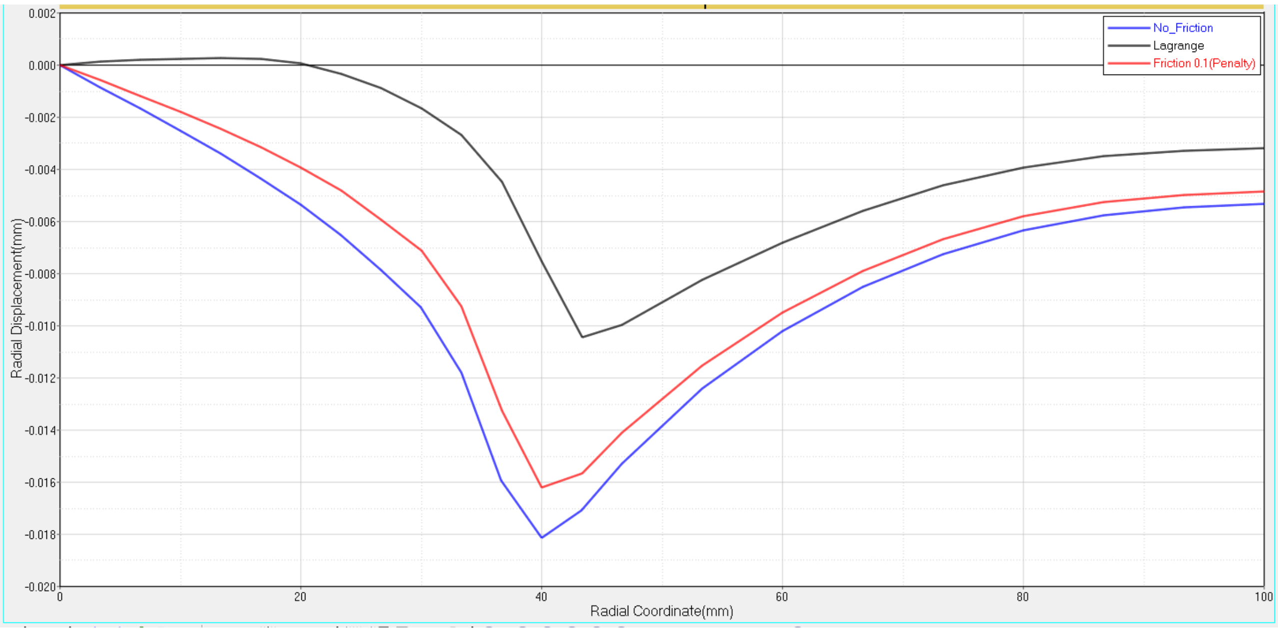


Figure-5:Effect of different friction coefficient and method of fiction handling on the radial displacement of the foundation edge(Linear Elements)

File Location

<install_directory>/demos/hwsolvers/optistruct/contb2H8.fem

<install_directory>/demos/hwsolvers/optistruct/contb2H8f.fem

<install_directory>/demos/hwsolvers/optistruct/contb2H20.fem

<install_directory>/demos/hwsolvers/optistruct/contb2H20f.fem

<install_directory>/demos/hwsolvers/optistruct/contb2H8L.fem

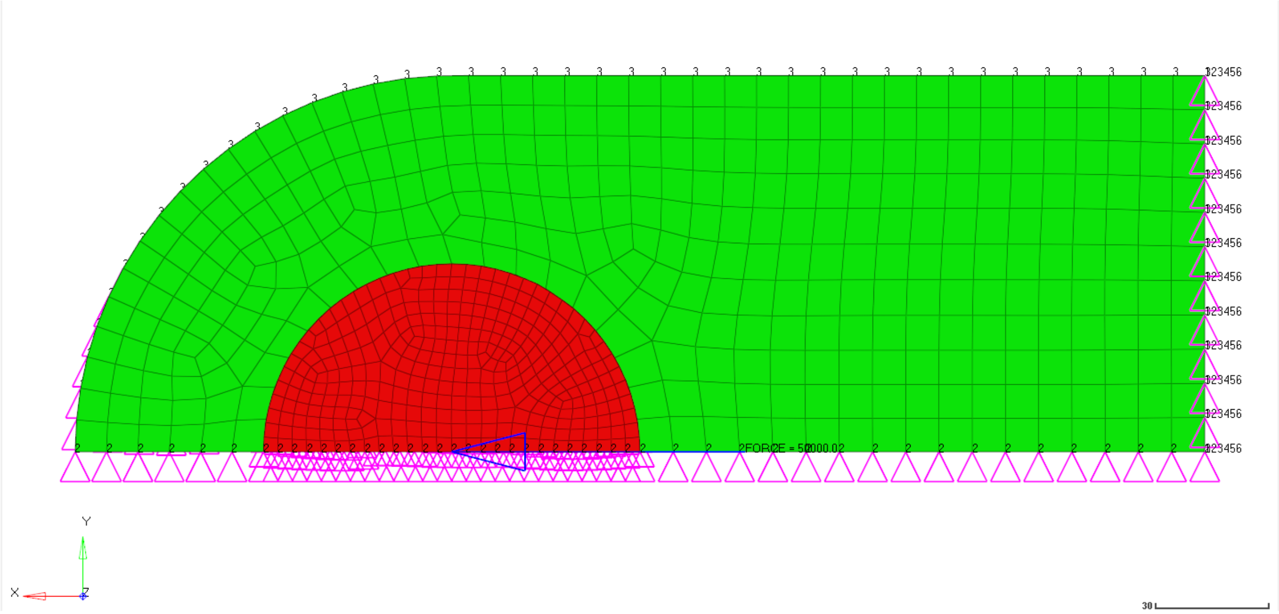
Reference

NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

CONTACT, NLPARM, NLADAPT, NLOUT, CNTSTB, PLOAD4, PCONT.

31. NAFEMS Contacts Benchmark 4– 3D Loaded Pin



Summary

This is the NAFEMS Contact Benchmark - 4, 3D Loaded Pin for Quasi-static analysis using Linear elastic material, geometric non-linearity and non-linear boundary conditions. OptiStruct FE results examine the plot of contact pressure, tangential stress and relative tangential slip against angle θ.

Benchmark Model

Hexa8 and Hexa20 elements are used to create one quarter model. The length of the sheet from left side to the center is 200mm, the inner radius of the sheet is 50mm, the outer radius of the sheet is 100mm, height of the sheet is 200mm, length of the pin is 20mm and the thickness of the sheet is 10mm. The outer surface of the pin and the inner surface of the sheet are in contact. Two equal point forces, resulting in a total force on the pin of 100kN is acting on both sides of the pin. The left side of the sheet is fixed. A frictional coefficient of 0.1 is acting between the contacts. The nodes along the pin boundary are selected as slave nodes, while the nodes along the strip are specified to be the master nodes.

Material properties

E_{pin} = 210 kN/mm^2
ν_{pin}= 0.3
E_{sheet} = 70 kN/mm^2
ν_{sheet} = 0.3

Non-Linear Quasi Static Analysis Results

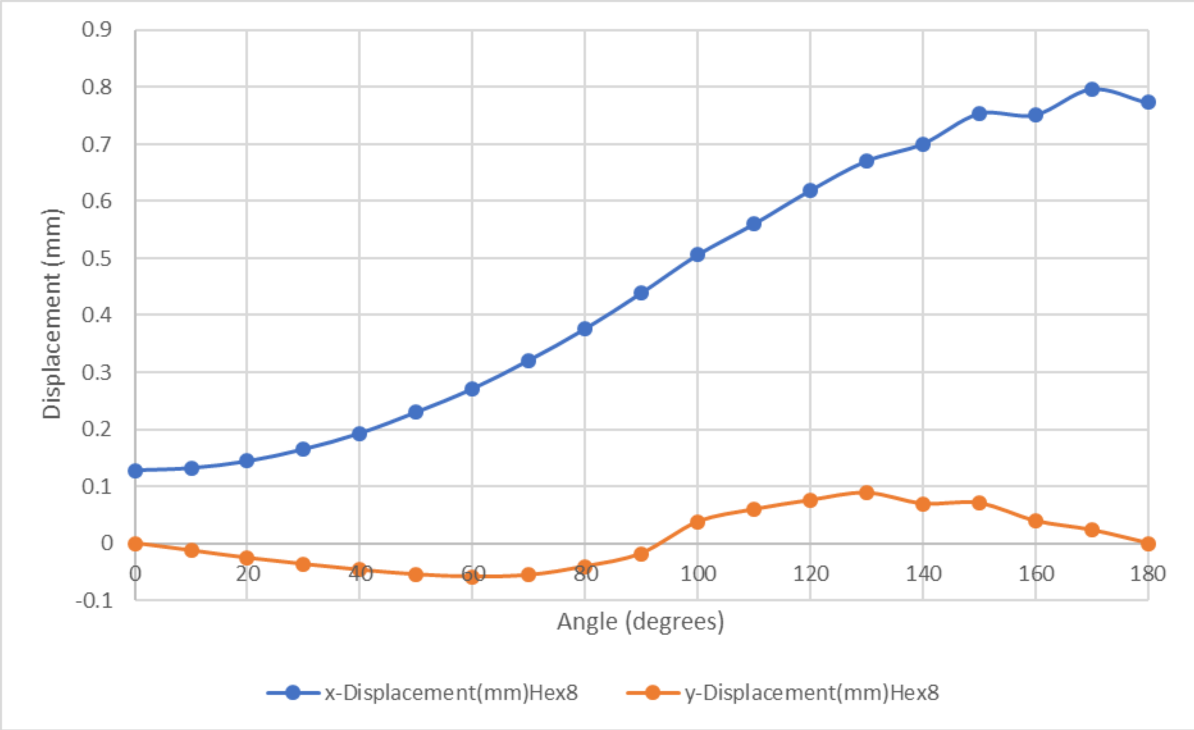
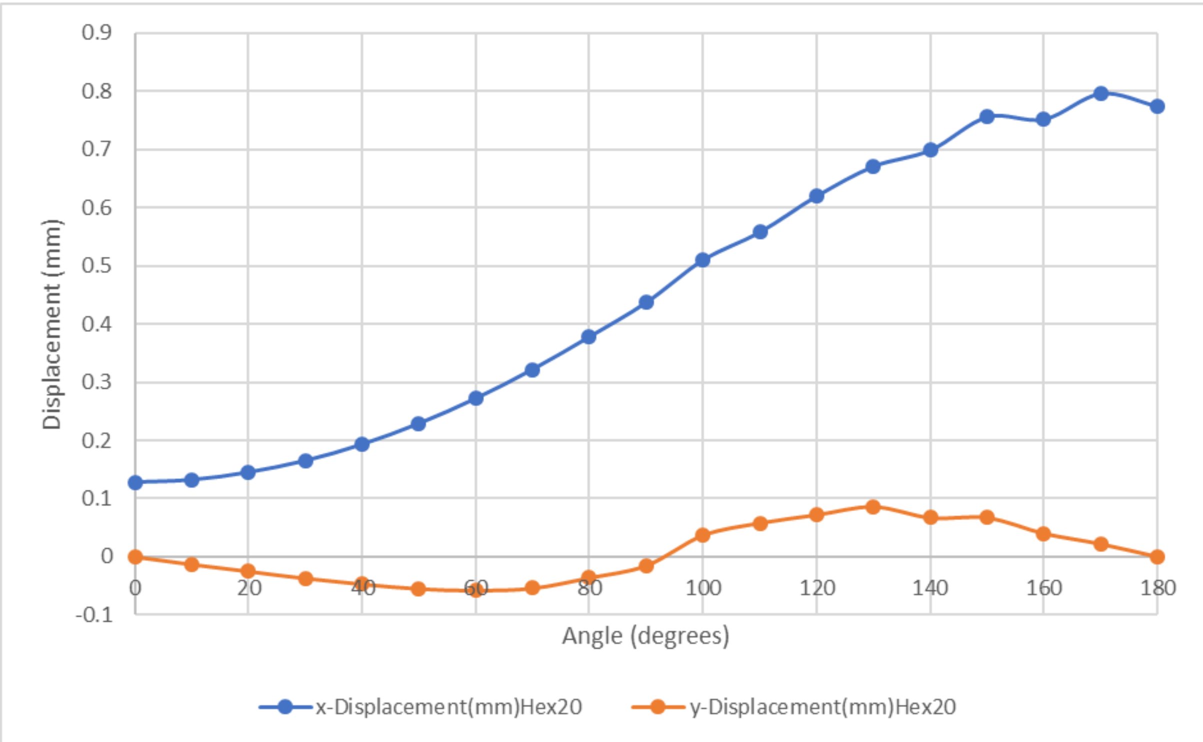


Figure-1:Displacement as a function of the angles obtained with first order elements for the nodes of the sheet contact surface.



Proprietary Information of Altair Engineering. All Rights Reserved.

Figure-2: Displacement as a function of the angles obtained with second order elements for the nodes of the sheet contact surface.

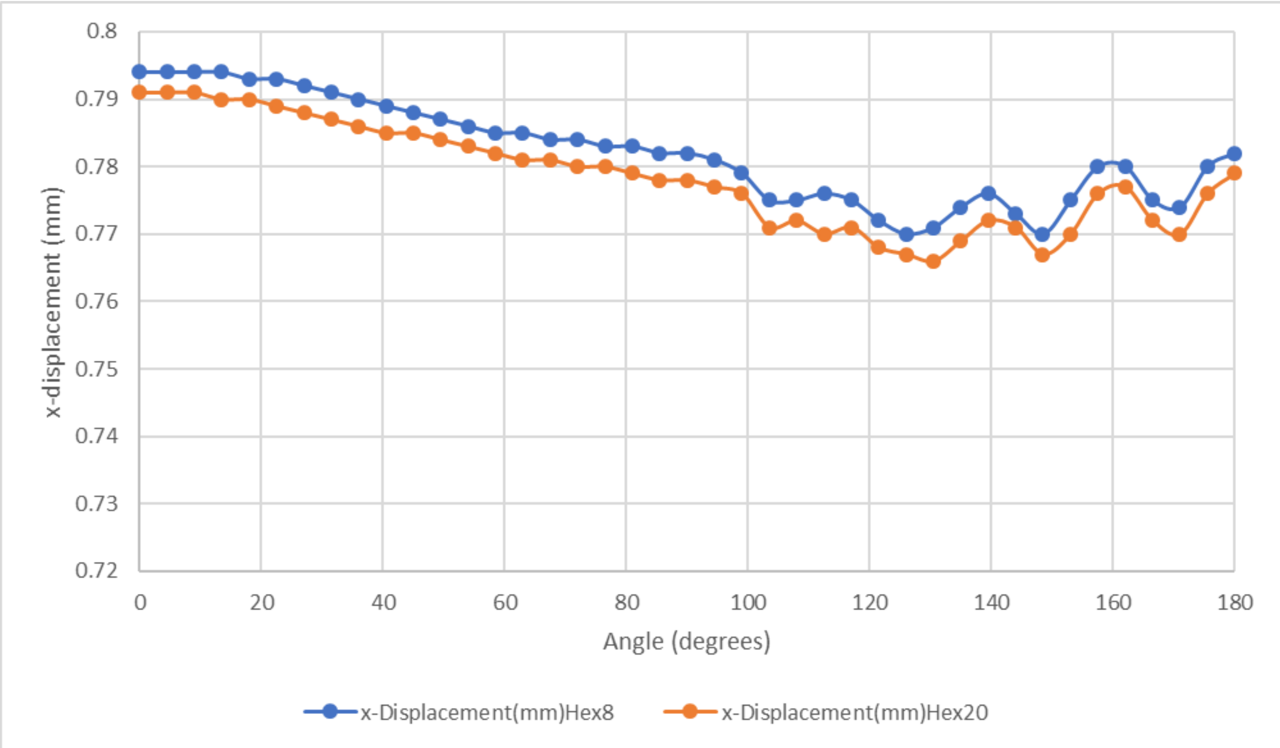


Figure-3:Displacement in x-direction for nodes along the pin as a function of the angle.

File Location

<install_directory>/demos/hwsolvers/optistruct/contb4H8.fem

<install_directory>/demos/hwsolvers/optistruct/contb4H20.fem

Reference

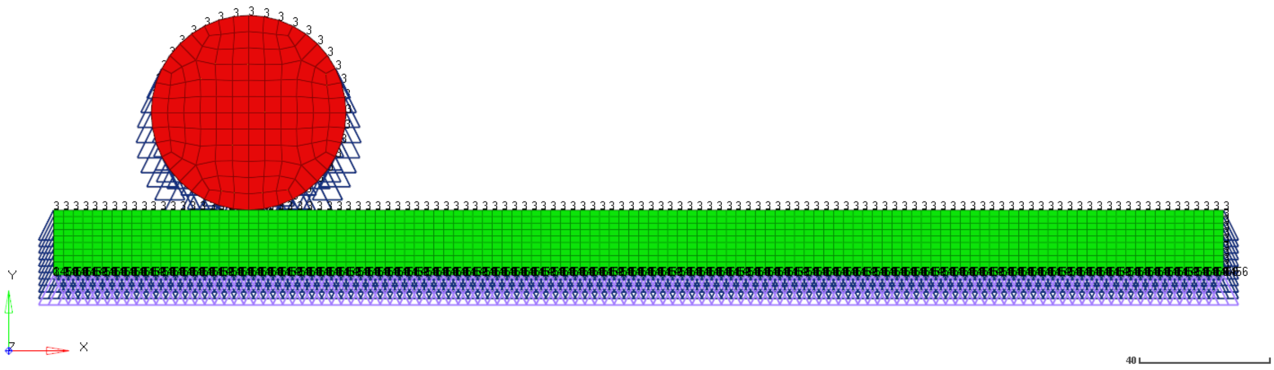
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

CONTACT, NLPARM, NLADAPT, NLOUT, CNTSTB, PLOAD4, PCONT.

Proprietary Information of Altair Engineering. All Rights Reserved.

32. NAFEMS Contacts Benchmark – 3D Steel Roller on Rubber



Summary

This is the NAFEMS Contact Benchmark - 5, 3D Steel Roller on Rubber for Quasi-static analysis using Linear elastic material, geometric non-linearity and non-linear boundary conditions. OptiStruct FE results examine the horizontal displacement of the point A after 360 degrees' motion. OptiStruct also examines the 3D deformable-deformable contact, Rolling contact and Incompressible material feature.

Benchmark Model

Hexa8 elements are used to create one half of the model. The Steel is of 20mm width and 30mm radius, the rubber mat is 22mm wide, 20mm in high and 360mm long. The steel roller starts rolling from a point 60mm from the left-hand side of the rubber mat. The center of the roller is fixed in horizontal and vertical direction, for a time period of 0-1 second the bottom surface of the rubber is displaced 3mm in the negative y direction, the sheet x-displacement is fixed and there is no roller rotation. For the time period of 1-2 second the bottom surface of the rubber sheet is held at 3mm y-displacement and rotation of 360 degrees is prescribed to the steel roller where the sheet is free to move in horizontal direction. There is no force applied on the system and the coefficient of friction between the two surface is 0.3. The nodes on the outer surface of the roll are selected as master nodes, while the nodes on the top surface of the mat are specified as the slave nodes.

Material properties

E_{steel} = 210 kN/mm^2

N_{steel}= 0.3

$C_{10,\ rubber}$ = 10 N/mm^2 (Neo Hookean material description)

$D1_{rubber}$ = 0.0001

Proprietary Information of Altair Engineering. All Rights Reserved.

Non-Linear Quasi Static Analysis Results

Horizontal Displacement (mm)	NAFEMS	OptiStruct	Normalized
3D first order elements	182.9	182.06	1.00461386

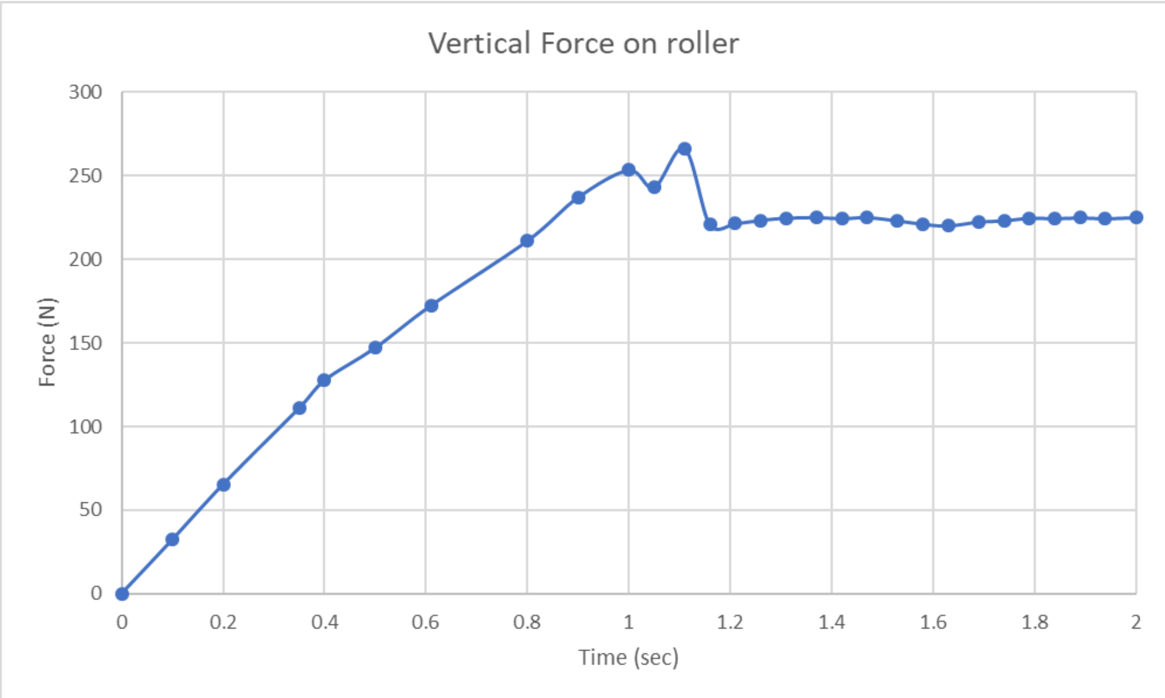
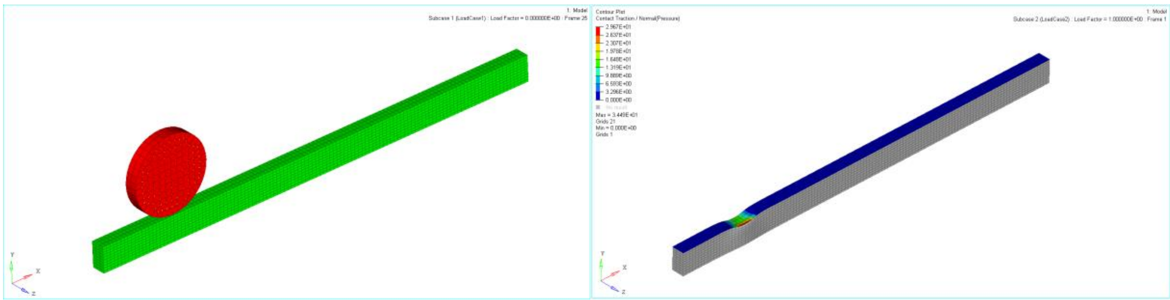


Figure-1:Vertical forces on the roller versus time.



Proprietary Information of Altair Engineering. All Rights Reserved.

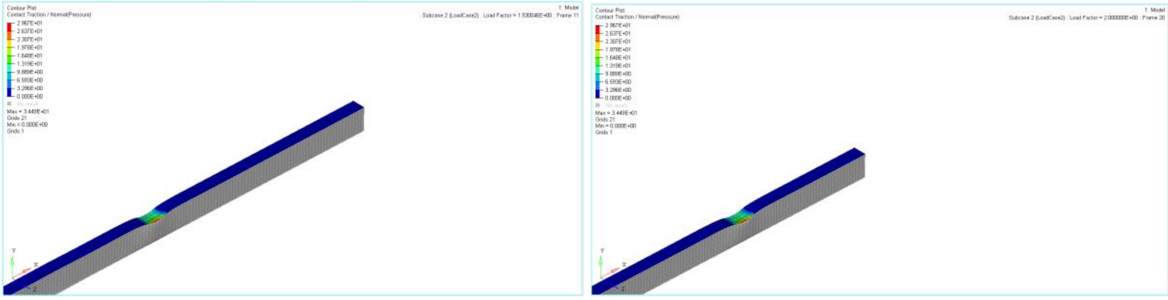


Figure-2:3D analysis – undeformed and contour plots of contact pressure on deformed structure.

File Location

<install_directory>/demos/hwsolvers/optistruct/contb5H8.fem

Reference

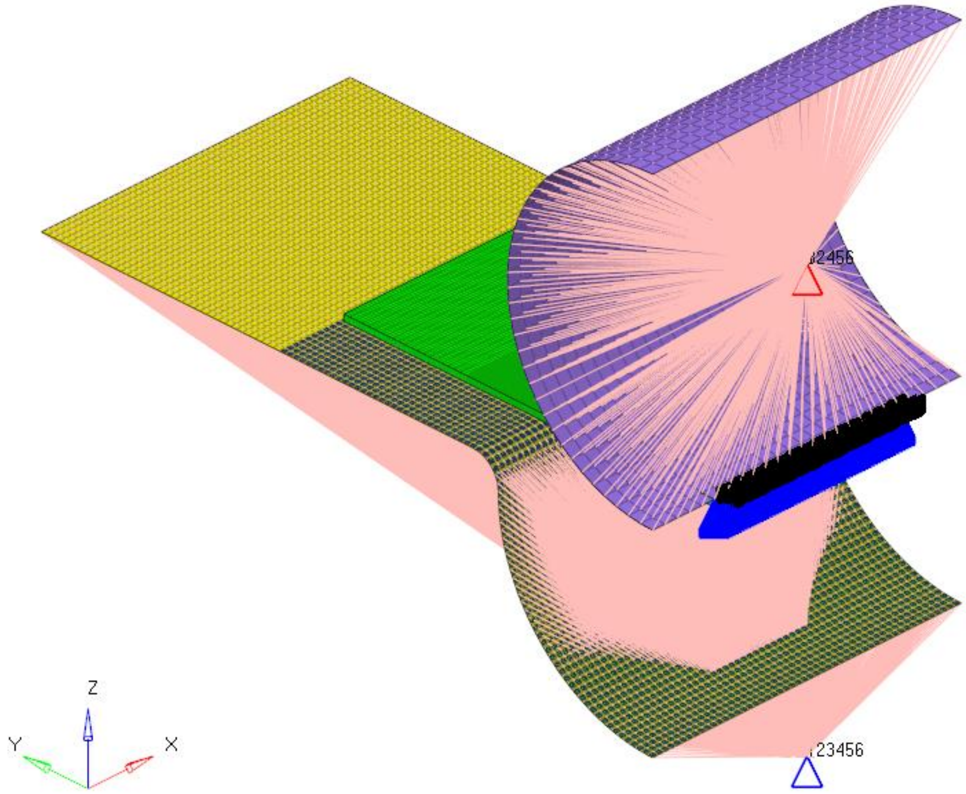
NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

CONTACT, NLPARM, NLADAPT, NLOUT, CNTSTB, PLOAD4, PCONT, MATHE NLMON, TABLED1.

Proprietary Information of Altair Engineering. All Rights Reserved.

33. NAFEMS Contacts Benchmark 3– 3D Sheet Metal Forming



Summary

This is the NAFEMS Contact Benchmark - 3, 3D Sheet Metal Forming for Quasi-static analysis using Elastic plastic material, geometric non-linearity and non-linear boundary conditions. OptiStruct FE results examine the forming angle and angle after the punch is released. OptiStruct also examines the contact features of the rigid and deformable bodies and sliding contact around the circular surfaces.

Benchmark Model

Hexa8 elements are used to create the half model of the sheet and Quad4 elements are used to model the punch and the die. The punch radius is 23.5mm, the die radius is 25mm, the die shoulder radius is 4mm, width of the tool is 50mm, length of sheet is 120mm, sheet thickness is 1mm and the width of the sheet is 30mm. The punch stroke is 28.5mm. The bottom surface is fixed. Two different contact properties are used, one with coefficient of friction 0.0 and the second with coefficient of friction 0.1342. For the contacts between the punch and the sheet, punch is considered as master surface and the sheet as slave and for the contacts between the die and the sheet die is considered as master and sheet as slave.

Material properties

E = 70.5 kN/mm^2

Proprietary Information of Altair Engineering. All Rights Reserved.

v= 0.342
σ_0 (Initial yield stress) = 194 N/mm^2
Hollomon hardening $\sigma=K\times\varepsilon^n$
K= 550.4 N/mm^2
n=0.223

Non-Linear Quasi Static Analysis Results

Characteristic angles during process

Frictional Coefficient=0	NAFEMS	OS results	Normalized
Forming angle	21.88	20.50	1.067317
Angle after release	48.38	45.53	1.062596
Frictional Coefficient=0.1348			
Forming angle	21.84	22.437	0.973392
Angle after release	54.45	43.22	1.259833

File Location

<install_directory>/demos/hwsolvers/optistruct/contb3f0.fem

<install_directory>/demos/hwsolvers/optistruct/contb3fc.fem

Reference

NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

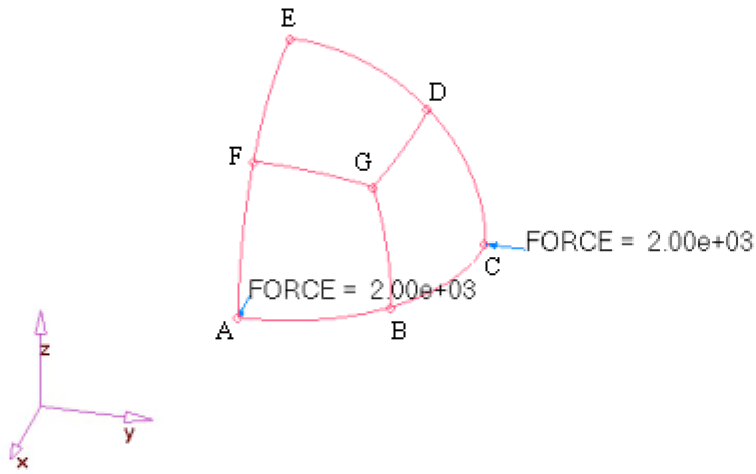
Keywords

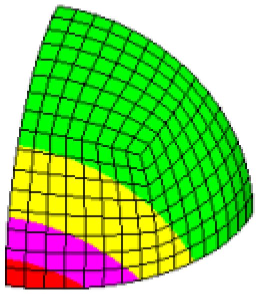
CONTACT, NLPARM, NLADAPT, NLOUT, CNTSTB, PLOAD4, PCONT, MATS1.

Proprietary Information of Altair Engineering. All Rights Reserved.

Altair

34. NAFEMS Test Problem (LE3) - Radial Point Load on a Hemisphere





Summary

This is the NAFEMS test problem LE3. The model is a hemispherical shell subjected to concentrated radial loads at its free edges. It examines the performance of the three-dimensional shell to model local bending behavior under conditions where the deformations are primarily due to bending.

Benchmark Model

4-node, first order CQUAD4 elements are benchmarked in LE3. The hemisphere is 10m in radius and 0.04m in radial thickness. Two pairs of identical loads, 4000N, are applied at the free edge of the hemisphere, and are at right angles to each other. One pair of the loads is directed inwards (toward the center) of the hemisphere, while the second pair is directed outward from the center, producing deformation of compression in one direction and elongation in another. Since both the geometry and loads are symmetrical, only a quarter of the hemisphere is modeled. Symmetric boundary constraints are applied on edges AE and CE. The z-translation at point E is fixed, and all displacements on edge AC are free. The test also requires the mesh of the hemisphere to have equally spaced nodes on edges AC, CE, EA, BG, DG, and FG. The target is x-translation at point A, with a target value of 0.185m.

Material properties

E = 68.25 GPa

$\nu = 0.3$.

Linear Static Analysis Results

All results are normalized with the target values of x translation at point A.

Element Type	**Mesh Configuration**		$\mathbf{n_{ac} \times n_{ce} \times n_{ea}}$		
CQUAD4	4 x 4 x 4	8 x 8 x 8	16 x 16 x 16	32 x 32 x 32	64 x 64 x 64

Proprietary Information of Altair Engineering. All Rights Reserved.

	0.9865	1.0200	1.0076	1.0032	1.0016

File Location

<install_directory>/demos/hwsolvers/optistruct/LE3.fem

Reference

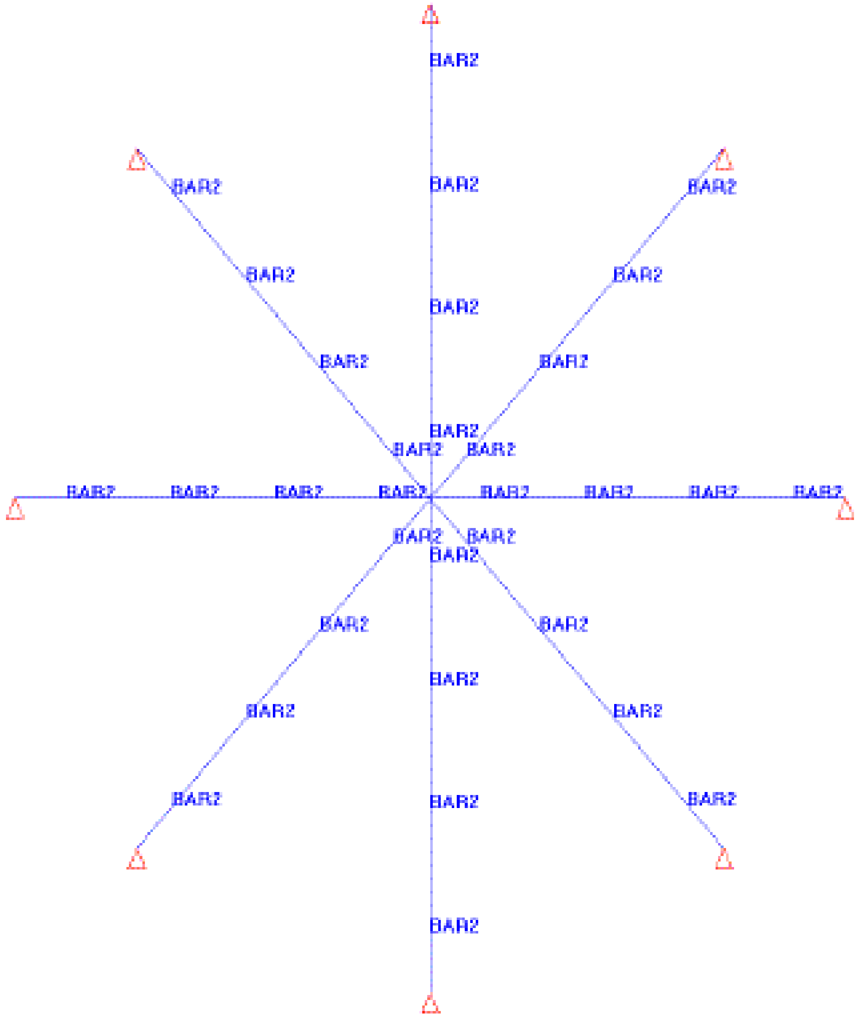
NAFEMS Ltd, *The Standard NAFEMS BENCHMARKS TNSB Rev. 3*, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

FORCE

Proprietary Information of Altair Engineering. All Rights Reserved.

35. Pin-ended Double Cross (Test No. FV2)



Summary

Test No. FV2 is a pin-ended double cross, in-plane vibration problem. OptiStruct is used to investigate the coupling between flexural and extensional behavior, as well as the repeated and closed eigenvalues of the double cross in normal modes analysis.

Benchmark Model

The 2-node simple beam elements are used to model the double cross. Each arm of the cross consists of four elements. The x- and y-translation displacements are fixed at the end of all arms.

Material properties

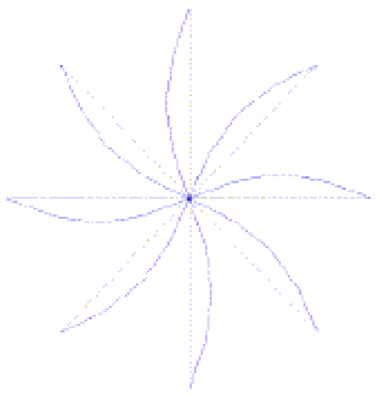
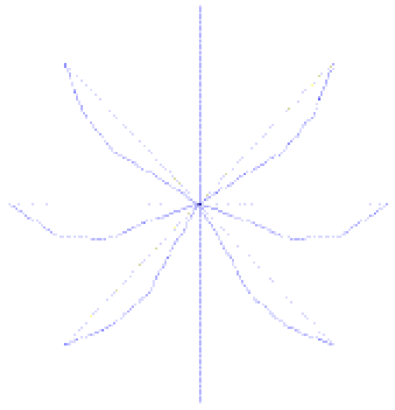
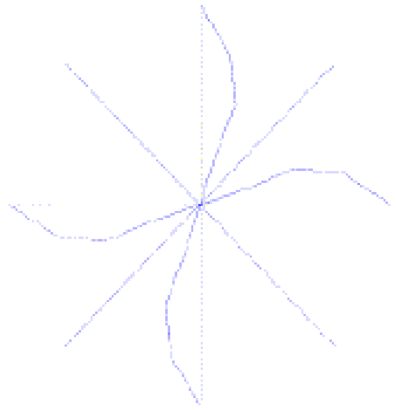
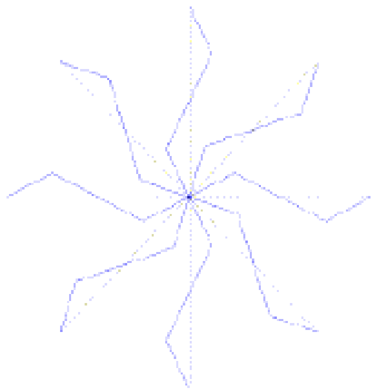
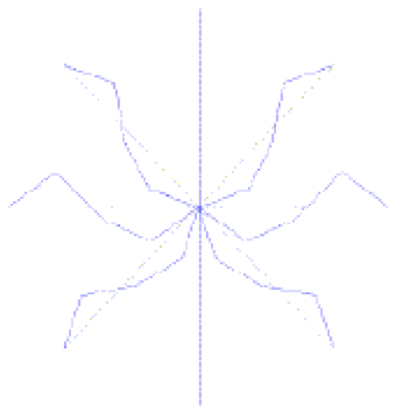
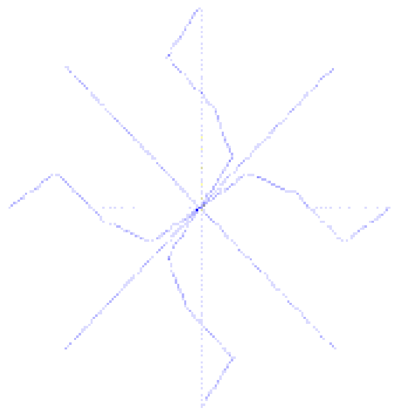
E = 200x10^9 N/m^2 and ρ = 8000 kg/m^3

Proprietary Information of Altair Engineering. All Rights Reserved.

Modal Analysis Results

The frequency of each targeted mode is normalized with the closed form solution.

`f* = closed form solution`

Mode 1		Modes 2 and 3		Modes 4, 5, 6, 7 and 8	
					
f* CBAR	11.336 Hz 0.9984	f* CBAR	17.709 Hz 0.9958	f* CBAR	17.709 Hz 0.9974
Mode 9		**Modes 10 and 11**		**Modes 12, 13, 14, 15 and 16**	
					
f* CBAR	45.345 Hz 1.0000	f* CBAR	57.390 Hz 0.9944	f* CBAR	57.390 Hz 0.9999

File Location

<install_directory>/demos/hwsolvers/optistruct/fv2.fem

Proprietary Information of Altair Engineering. All Rights Reserved.

Reference

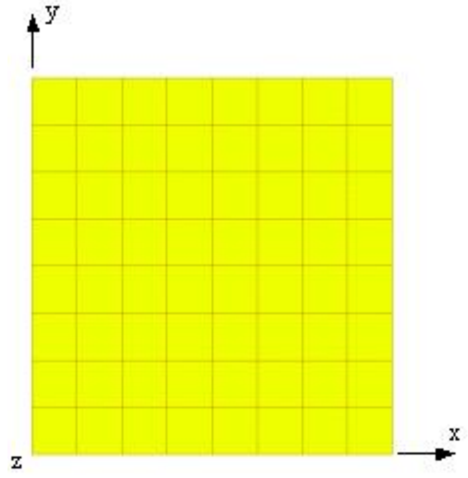
NAFEMS Ltd, *The Standard NAFEMS BENCHMARKS TNSB Rev. 3*, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

CBAR, PBAR, MAT8.

Proprietary Information of Altair Engineering. All Rights Reserved.

36. Free Thin Square Plate (Test No. FV12)



Summary

Test No. FV12 is a thin square plate model, which contains three rigid body modes. OptiStruct investigates the repeated eigenvalues and kinematically incomplete suppressions of the thin plate in normal modes analysis.

Benchmark Model

The 4-node quad elements with 8x8 mesh configuration are used to model the plate. The x-translation, y-translation displacements and z-rotational displacements are fixed for all nodes.

Material properties

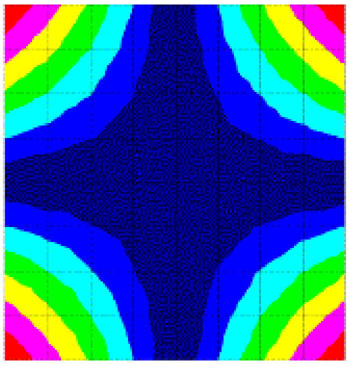
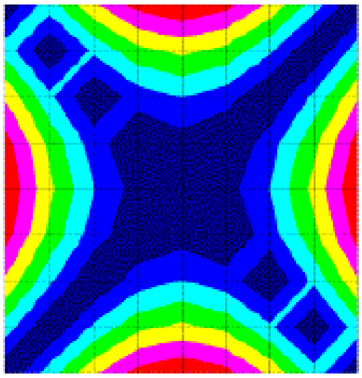
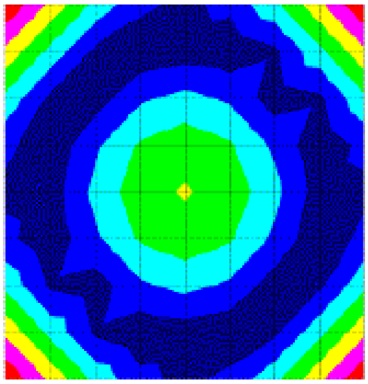
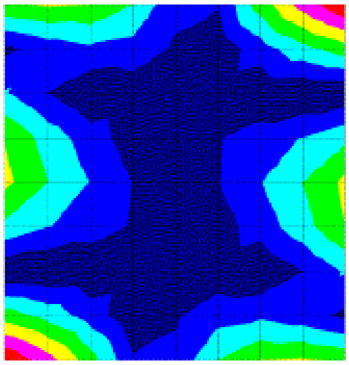
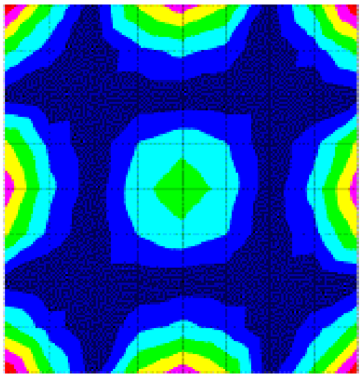
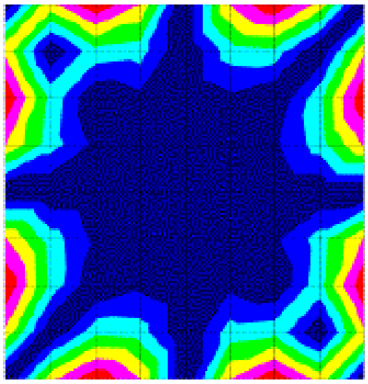
E = 200×10^9 N/m^2, ν = 0.3 and ρ = 8000 kg/m^3

Modal Analysis Results

The frequency of each targeted mode is normalized with the closed form solution. The closed form solution is not available for mode 10, use target value for H.O.E. instead.

`f* = closed form solution`

Proprietary Information of Altair Engineering. All Rights Reserved.

Mode 4		**Mode 5**		**Mode 6**	
					
F* CQUAD4	1.622 Hz 0.9964	f* CQUAD4	2.360 Hz 1.0145	f* CQUAD4	2.922 Hz 1.0231
Mode 7 and 8		**Mode 9**		**Mode 10**	
					
F* CQUAD4	4.233 Hz 0.9978	f* CQUAD4	7.416 Hz 1.0496	F[HOE] CQUAD4	7.392 Hz 1.1746

File Location

<install_directory>/demos/hwsolvers/optistruct/fv12.fem

Reference

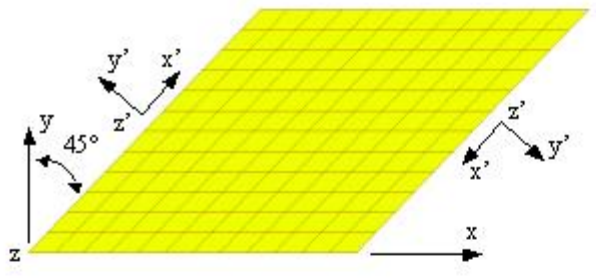
NAFEMS Ltd, *The Standard NAFEMS BENCHMARKS TNSB Rev. 3*, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Proprietary Information of Altair Engineering. All Rights Reserved.

Keywords

EIGRL

37. Clamped Thin Rhombic Plate (Test No. FV15)



Summary

Test No. FV15 is a thin rhombic plate problem. The behavior of distorted thin elements in normal modes analysis is examined using OptiStruct.

Benchmark Model

The 4-node quad elements with 12x12 mesh configuration are used to model the thin rhombic plate. The x-translation, y-translation displacements, and z-rotational displacements are fixed for all of the nodes. In addition, the z-translation, x-rotational and y-rotational are fixed for the nodes along the four edges.

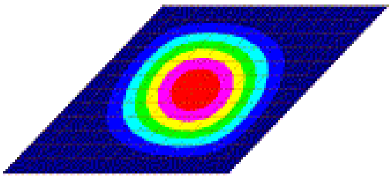
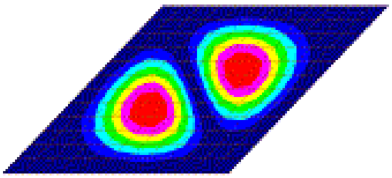
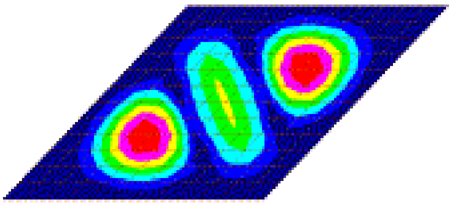
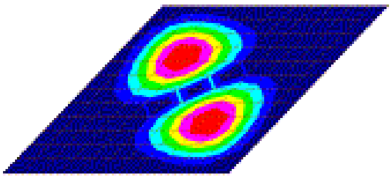
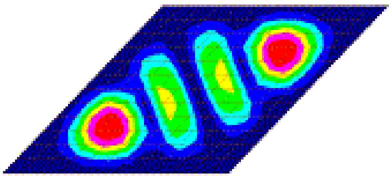
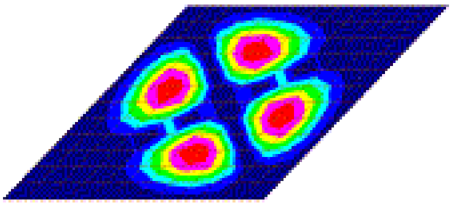
Material properties

E = 200x10^9 N/m^2, ν = 0.3 and ρ = 8000 kg/m^3

Modal Analysis Results

The frequency of each targeted mode is normalized with the closed form solution. If a closed form solution is not available, the target values for H.O.E. are used instead. All benchmarked results are normalized with target values.

`f* = closed form solution`

Mode 1		Mode 2		Mode 3	
					
F* CQUAD4	7.938 Hz 0.9798	f* CQUAD4	12.835 Hz 1.0079	f* CQUAD4	17.941 Hz 1.0182
Mode 4		**Mode 5**		**Mode 6**	
					
F* CQUAD4	19.133 Hz 0.9629	f* CQUAD4	24.009 Hz 1.0339	f* CQUAD4	27.922 Hz 1.0036

File Location

<install_directory>/demos/hwsolvers/optistruct/fv15.fem

Reference

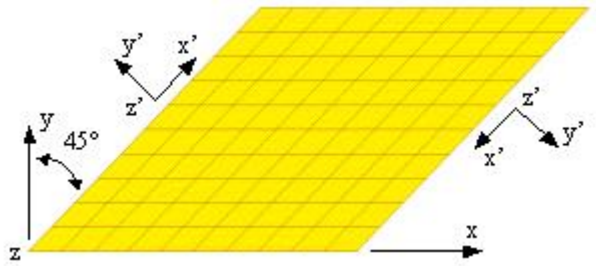
NAFEMS Ltd, *The Standard NAFEMS BENCHMARKS TNSB Rev. 3*, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

EIGRL, CORD2R.

Proprietary Information of Altair Engineering. All Rights Reserved.

38. Clamped Thick Rhombic Plate (Test No. FV22)



Summary

Test No. FV22 is a thick rhombic plate problem. The behavior of distorted, thick elements in normal modes analysis is examined using OptiStruct.

Benchmark Model

The 4-node quad elements with 10x10 mesh configuration are used to model the thick rhombic plate. The x-translation, y-translation displacements, and z-rotational displacements are fixed for all of the nodes. In addition, the z-translation, x-rotational and y-rotational are fixed for the nodes along the four edges.

Material properties

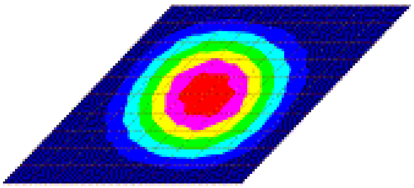
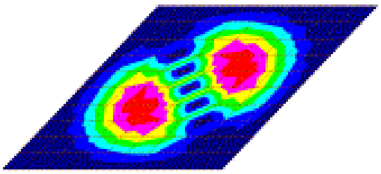
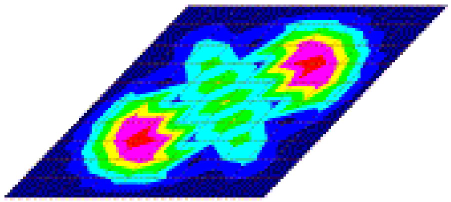
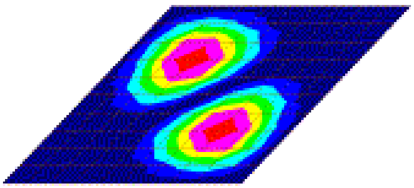
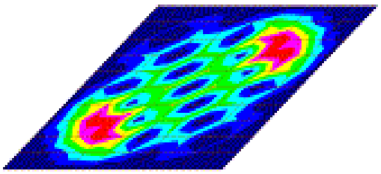
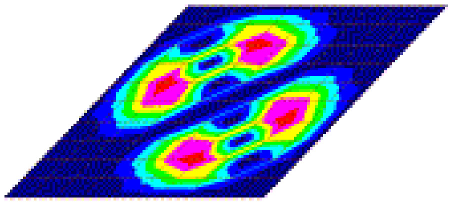
E = 200x10^9 N/m^2, ν = 0.3 and ρ = 8000 kg/m^3

Modal Analysis Results

The frequency of each targeted mode is normalized with the closed form solution. The closed form solution is not available for mode 6, the target value for H.O.E. should be used instead.

`f* = closed form solution`

Proprietary Information of Altair Engineering. All Rights Reserved.

Mode 1		Mode 2		Mode 3	
					
F* CQUAD4	133.95 Hz 0.9937	f* CQUA D4	201.41 Hz 1.0382	f* CQUA D4	265.81 Hz 1.0633
Mode 4		**Mode 5**		**Mode 6**	
					
F* CQUAD4	283.68 Hz 1.0006	f* CQUA D4	334.45 Hz 1.1073	F[HO E] CQUA D4	386.62 Hz 1.0403

File Location

`<install_directory>/demos/hwsolvers/optistruct/fv22.fem`

Reference

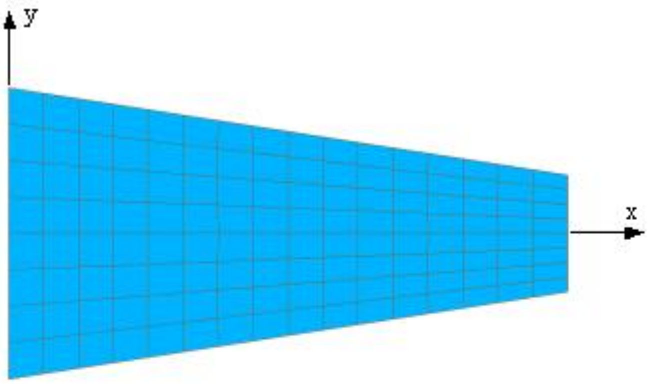
NAFEMS Ltd, *The Standard NAFEMS BENCHMARKS TNSB Rev. 3*, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

EIGRL, CORD2R.

Proprietary Information of Altair Engineering. All Rights Reserved.

39. Cantilevered Tapered Membrane (Test No. FV32)



Summary

Test No. FV32 is a tapered membrane problem with irregular mesh. The geometry and mesh symmetry are maintained along the x-direction. The shear behavior of membrane elements in the normal modes analysis is examined using OptiStruct.

Benchmark Model

The 4-node quad elements with 16x8 mesh configuration are used in test FV32. The displacements in z direction are fixed on all nodes and the x- and y-translation displacements are fixed for the nodes along the y-axis.

Material properties

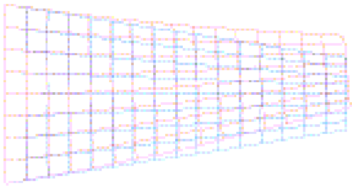
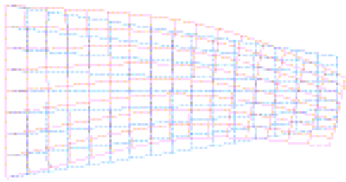
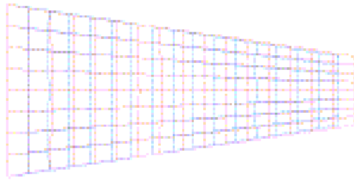
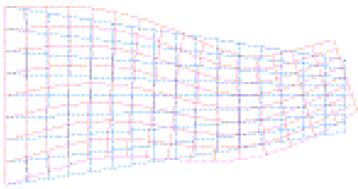
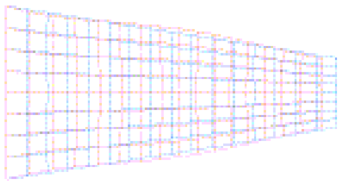
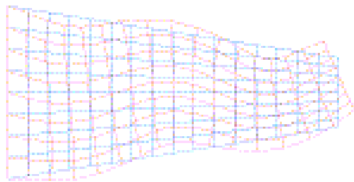
E = 200x10^9 N/m^2, ν = 0.3 and ρ = 8000 kg/m^3

Modal Analysis Results

The frequency of each targeted mode is normalized with the closed form solution.

```
f* = closed form solution
```

Proprietary Information of Altair Engineering. All Rights Reserved.

Mode 1		Mode 2		Mode 3	
					
f* CQUAD4	44.623 Hz 0.9092	f* CQUAD4	130.03 Hz 1.0265	f* CQUAD4	162.70 Hz 0.9104
Mode 4		**Mode 5**		**Mode 6**	
					
f* CQUAD4	246.05 Hz 1.0766	f* CQUAD4	379.90 Hz 1.0070	f* CQUAD4	391.44 Hz 1.0479

File Location

<install_directory>/demos/hwsolvers/optistruct/fv32.fem

Reference

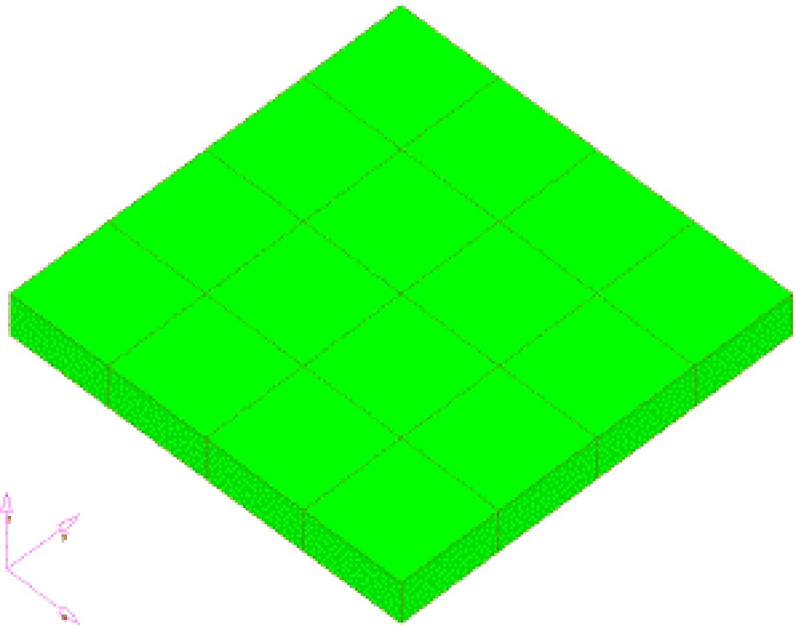
NAFEMS Ltd, *The Standard NAFEMS BENCHMARKS TNSB Rev. 3*, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

EIGRL, CORD2R.

Proprietary Information of Altair Engineering. All Rights Reserved.

40. Simply-Supported Solid Square Plate (Test No. FV52)



Summary

Test No. FV52 is a well-established solid square plate, which contains three rigid modes with the given boundary condition. The test examines the performance of OptiStruct normal modes analysis on 3D solid elements.

Benchmark Model

The HEXA 8-node and HEXA 20-node solid elements are used in test FV52. An 8x8x3 mesh configuration is used for HEXA 8-node elements, and a 4x4x1 mesh configuration is used with HEXA 20-node elements. The z-directional displacement is constrained at Z = -5m plane along the four edges of the plate.

Material properties

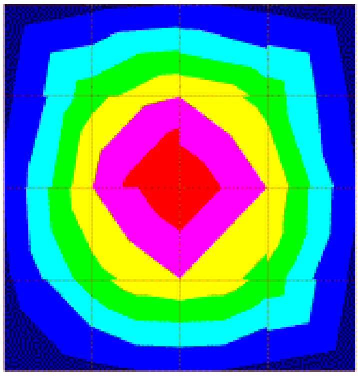
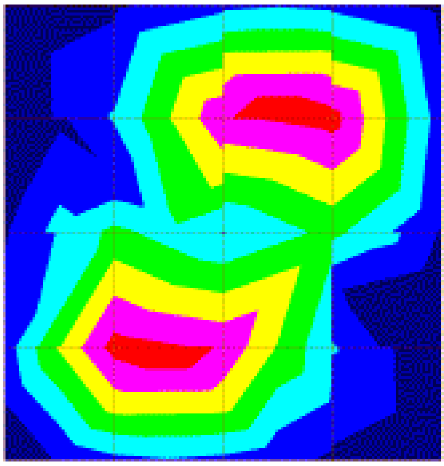
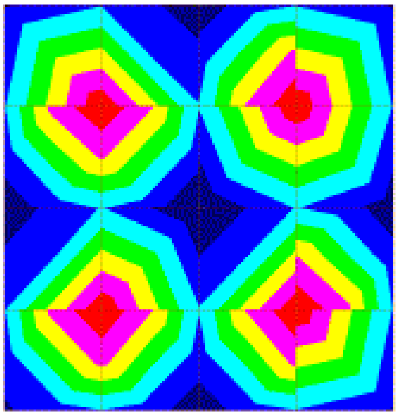
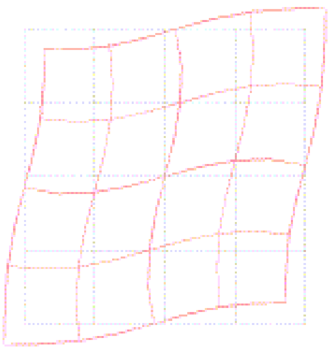
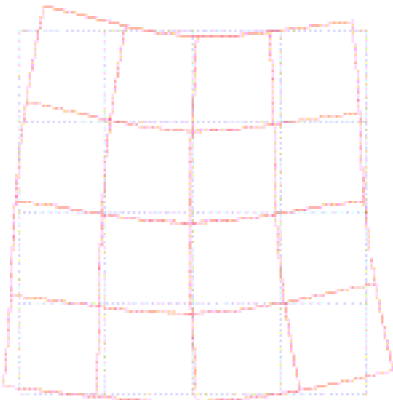
E = 200×10^9 N/m^2, ν = 0.3 and ρ = 8000 kg/m^3

Modal Analysis Results

The frequency of each targeted mode is normalized with the closed form solution.

`f* = closed form solution`

Proprietary Information of Altair Engineering. All Rights Reserved.

Mode 4		**Modes 5 and 6**		**Mode 7**	
					
f* HEXA20 HEXA8	45.897 Hz 0.9760 0.9874	f* HEXA20 HEXA8	109.44 Hz 1.0101 1.0413	f* HEXA2 0 HEXA8	167.89 Hz 1.0072 1.0322
Mode 8		**Modes 9 and 10**			
					
f* HEXA20 HEXA8	193.59 Hz 1.0017 1.0164	f* HEXA20 HEXA8	206.19 Hz 1.0022 1.0164		

File Location

```
<install_directory>/demos/hwsolvers/optistruct/fv52_HOE.fem
<install_directory>/demos/hwsolvers/optistruct/fv52_LOE.fem
```

Proprietary Information of Altair Engineering. All Rights Reserved.

Reference

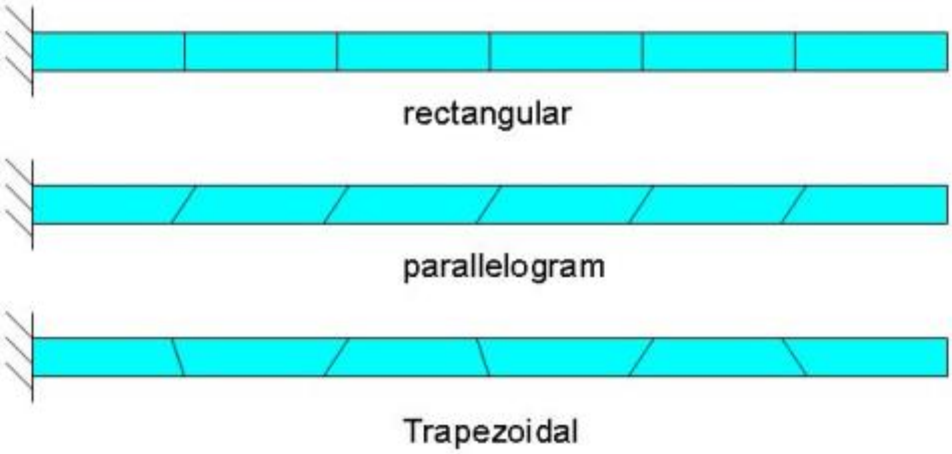
NAFEMS Ltd, *The Standard NAFEMS BENCHMARKS TNSB Rev. 3*, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

EIGRL.

Proprietary Information of Altair Engineering. All Rights Reserved.

41. MacNeal-Harder Test: Straight Cantilever Beam



Length = 6.0; Height = 0.2; Depth = 0.1; E = 1.0E7: Poisson's ratio = 0.3.

Summary

This is a straight cantilever beam solved with solid and shell elements. Three models (rectangular, parallelogram, trapezoidal) are made with each element's type to investigate the effect of distorted elements with a high aspect ratio.

Benchmark Model

Six types of elements are used for this problem. They are tria-shell, quad-shell, and hexa-solid elements, each with 1st and 2nd order. Four loading cases are used for each model; extension, in-plane bending, transverse bending, and twist. For the extension and bending load cases, unit loads are applied in a consistent fashion over all of the nodes at the tip of the beam. For the twist load cases, a unit moment is applied at the tip.

Theoretical solutions for the deflections at the tip, computed by beam theory, are as follows.

Load Type	**Component**	**Value**
extension	UX	0.00003
in-plane bending	UZ	0.1081
transverse bending	UY	0.4321
twist	ROTX	0.03208

Linear Static Analysis Results

All results are normalized with the target value.

(a) Rectangular				
	in-plane extension	**in-plane bending**	**Transverse bending**	**Twist**
QUAD4	1.000	0.992	0.981	0.941
QUAD8	1.006	1.000	1.016	0.953

Proprietary Information of Altair Engineering. All Rights Reserved.

(a) Rectangular				
TRI3	1.000	0.032	0.973	1.072
TRI6	1.006	0.994	1.001	0.950
HEX8	0.988	0.978	0.973	0.892
HEX20	1.008	0.992	0.992	0.905
(b) Parallelogram				
	in-plane extension	**in-plane bending**	**Transvers e bending**	**Twist**
QUAD4	1.000	0.712	0.981	0.905
QUAD8	1.008	0.999	1.015	0.937
TRI3	1.000	0.012	0.955	0.931
TRI6	1.005	0.962	0.995	0.982
HEX8	1.012	0.624	0.529	0.820
HEX20	1.008	0.976	0.977	0.905
(c) Trapezoidal				
	in-plane extension	**in-plane bending**	**Transvers e bending**	**Twist**
QUAD4	1.000	0.173	0.964	0.869
QUAD8	1.005	0.981	1.015	0.950
TRI3	1.000	0.019	0.965	1.175
TRI6	1.006	0.972	0.999	0.947
HEX8	1.010	0.047	0.030	0.563
HEX20	1.008	0.902	0.950	0.905

File Location

```
<install_directory>/demos/hwsolvers/optistruct/SCBhex8.fem
<install_directory>/demos/hwsolvers/optistruct/SCBhex20.fem
<install_directory>/demos/hwsolvers/optistruct/SCBquad4.fem
<install_directory>/demos/hwsolvers/optistruct/SCBquad8.fem
<install_directory>/demos/hwsolvers/optistruct/SCBtri3.fem
<install_directory>/demos/hwsolvers/optistruct/SCBtri6.fem
```

Reference

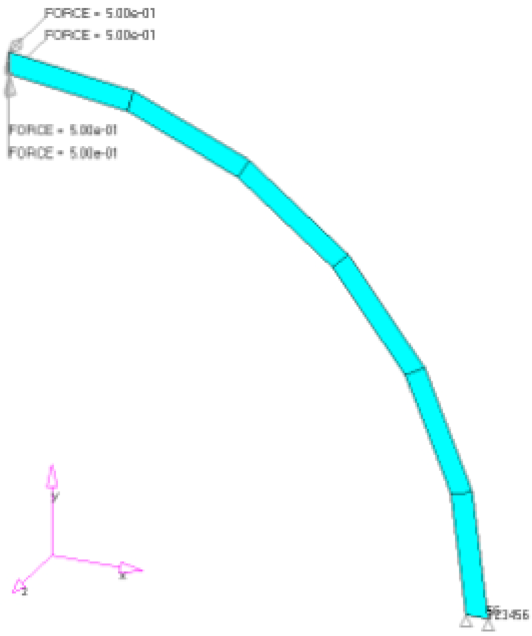
MacNeal, R.H., and Harder, R.L., A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Elements in Analysis and Design, 1 (1985) 3-20.

Keywords

FORCE.

Proprietary Information of Altair Engineering. All Rights Reserved.

42. MacNeal-Harder Test: Curved Cantilever Beam



E = 1.0E7; Poisson's ratio = 0.25; Loading; unit force at tip.

Summary

This is a curved cantilever beam solved with solid and shell elements. A model is made with each element's type to investigate the effect of distorted elements with a high aspect ratio.

Benchmark Model

Six types of elements are used for this problem. They are tria-shell, quad-shell, and hexa-solid elements, each with 1st and 2nd order. Two loading cases are used for each model; in-plane bending, transverse bending. For both load cases, unit loads are applied in a consistent fashion over all of the nodes at the tip of the beam.

Theoretical solutions for the deflections at the tip, computed by beam theory, are as follows.

Load Type	Component	Value
in-plane bending	UY	0.08734
transverse bending	UZ	0.5022

Linear Static Analysis Results

All results are normalized with the target value.

	In-plane Bending	Transverse Bending
QUAD4	0.952	0.955

Proprietary Information of Altair Engineering. All Rights Reserved.

QUAD8	1.015	0.984
TRI3	0.025	0.950
TRI6	1.005	0.961
HEX8	0.880	0.820
HEX20	1.009	0.946

File Location

```
<install_directory>/demos/hwsolvers/optistruct/CBhex8.fem
<install_directory>/demos/hwsolvers/optistruct/CBhex20.fem
<install_directory>/demos/hwsolvers/optistruct/CBquad4.fem
<install_directory>/demos/hwsolvers/optistruct/CBquad8.fem
<install_directory>/demos/hwsolvers/optistruct/CBtri3.fem
<install_directory>/demos/hwsolvers/optistruct/CBtri6.fem
```

Reference

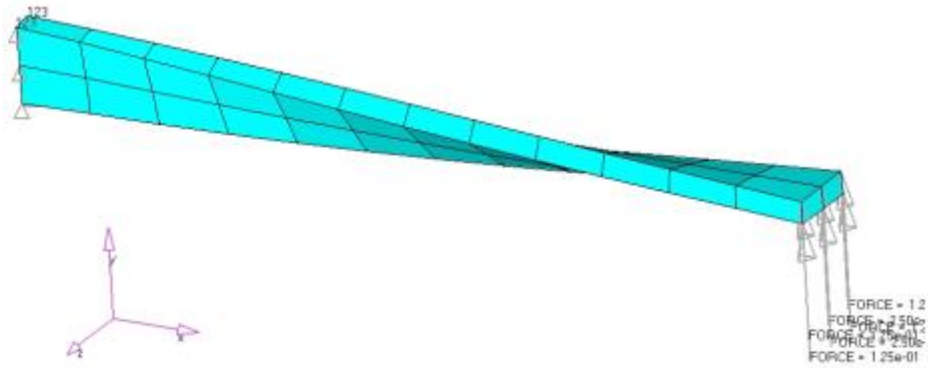
MacNeal, R.H., and Harder, R.L., A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Elements in Analysis and Design, 1 (1985) 3-20.

Keywords

CORD2C.

Proprietary Information of Altair Engineering. All Rights Reserved.

43. MacNeal-Harder Test: Twisted Cantilever Beam



Length = 12.0; Width = 1.1; Depth = 0.32; E = 29.0E6; Poisson's ratio = 0.22; Loading = unit forces at tip.

Summary

This is a twisted cantilever beam solved with solid and shell elements. A model is made with each element's type to investigate the effect of distorted elements with a high aspect ratio.

Benchmark Model

Six types of elements are used for this problem. They are tria-shell, quad-shell, and hexa-solid elements, each with 1st and 2nd order. Two loading cases are used for each model; in-plane bending, transverse bending. For both load cases, unit loads are applied in a consistent fashion over all of the nodes at the tip of the beam.

Theoretical solutions for the deflections at the tip, computed by beam theory, are as follows.

Load Type	Component	Value
in-plane bending	UY	0.001754
transverse bending	UZ	0.005424

Proprietary Information of Altair Engineering. All Rights Reserved.

Linear Static Analysis Results

All results are normalized with the target value.

	In-plane Bending	**Transverse Bending**
QUAD4	0.988	0.992
QUAD8	1.014	1.062
TRI3	0.839	0.984
TRI6	1.161	1.215
HEX8	0.986	1.005
HEX20	1.017	1.052

File Location

```
<install_directory>/demos/hwsolvers/optistruct/TBhex8.fem
<install_directory>/demos/hwsolvers/optistruct/TBhex20.fem
<install_directory>/demos/hwsolvers/optistruct/TBquad4.fem
<install_directory>/demos/hwsolvers/optistruct/TBquad8.fem
<install_directory>/demos/hwsolvers/optistruct/TBtri3.fem
<install_directory>/demos/hwsolvers/optistruct/TBtri6.fem
```

Reference

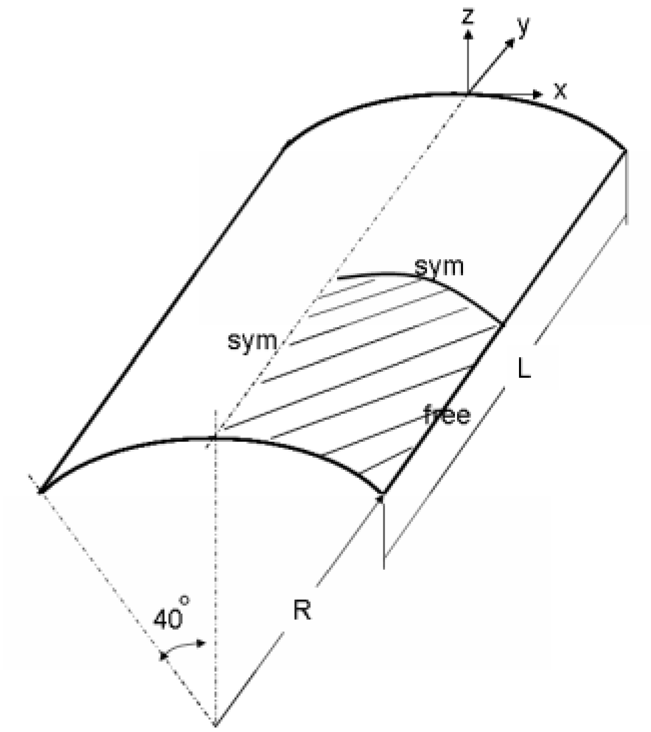
MacNeal, R.H., and Harder, R.L., A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Elements in Analysis and Design, 1 (1985) 3-20.

Keywords

FORCE

Proprietary Information of Altair Engineering. All Rights Reserved.

44. MacNeal-Harder Test: Scordelis-Lo Roof



R = 25.0; Length = 50.0, t = 0.25; E = 4.32E8; Poisson's ratio = 0.0; Ux = Uz = 0 on curved edge.

Summary

The Scordelis-Lo Roof is a classical benchmark problem for shell elements. Analytical and experimental investigations were initially performed by Scordelis and Lo.

Benchmark Model

The roof structure is supported on both ends, and loaded by self weight of 90 pounds per square foot, with only one quadrant modeled. Six types of elements are used for this problem. They are tria-shell, quad-shell, and hexa-solid elements, each with 1st and 2nd order. Each element type was benchmarked with different mesh density to check the convergence. As the original study was related to concrete structure, Poisson's ratio is set to zero. The structure is curved and its solution will exhibit membrane and bending behavior. Target solution for the vertical deflection at the midpoint of the free edge is 0.3024.

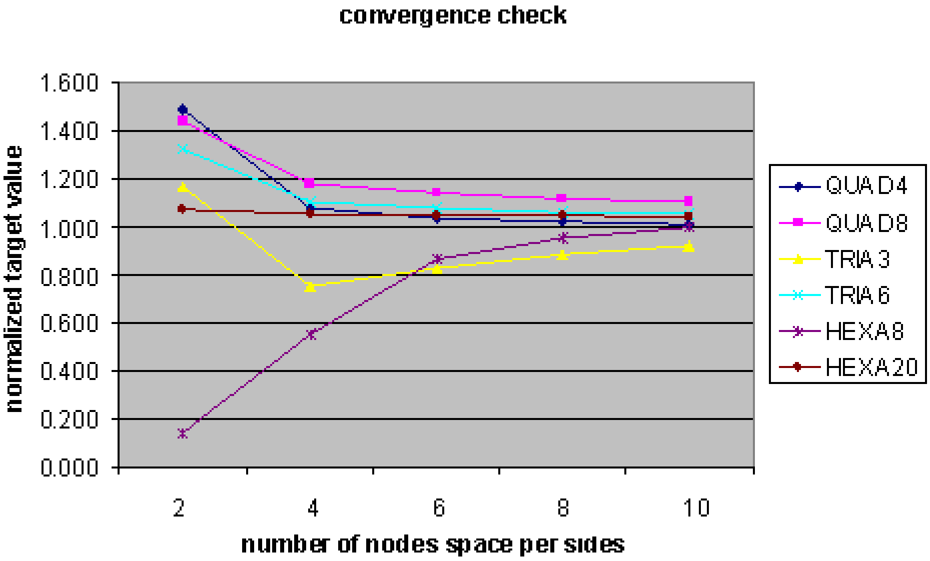
Linear Static Analysis Results

All results are normalized with the target value.

	Number of Nodes - space per sides				
	2	**4**	**6**	**8**	**10**
QUAD4	1.487	1.082	1.035	1.020	1.013
QUAD8	1.436	1.179	1.139	1.116	1.102
TRI3	1.169	0.753	0.829	0.886	0.920

Proprietary Information of Altair Engineering. All Rights Reserved.

TRI6	1.326	1.107	1.077	1.063	1.055
HEX8	0.142	0.550	0.868	0.951	1.000
HEX20	1.071	1.052	1.046	1.045	1.043



File Location

```
<install_directory>/demos/hwsolvers/optistruct/Roof1.fem
<install_directory>/demos/hwsolvers/optistruct/Roof2.fem
```

Reference

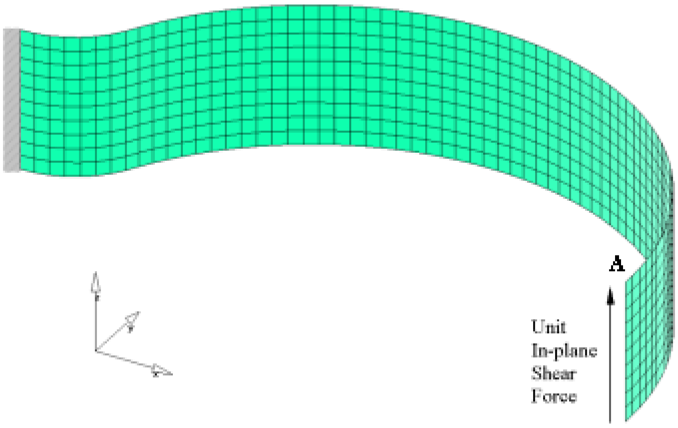
MacNeal, R.H., and Harder, R.L., A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Elements in Analysis and Design, 1 (1985) 3-20.

Keywords

PLOAD4

Proprietary Information of Altair Engineering. All Rights Reserved.

45. Raasch Challenge: Curved strip hook



Summary

Raasch challenge is a curved strip hook problem with a tip in-plane shear load, posed in 1990 by Ingo Raasch of BMW in Germany. The problem poses a significant challenge to shell elements because of the inherent coupling between three modes of deformation: bending, extension, and twist. OptiStruct is benchmarked against the Raasch challenge to assure its shell elements performance on linear static analysis.

Benchmark Model

Three types of elements are used for linear static analysis of the Raasch's hook. They are tria-shell, quad-shell, and hexa-solid elements. For each type of element, five different mesh densities (1x9, 3x17, 5x34, 10x68, and 20x136) are used to investigate the convergence of the solution. Two elements are used in the thickness direction for solid elements, thus, the mesh densities become 1x9x2, 3x17x2, 5x34x2, 10x68x2, and 20x136x2. The target is the z-directional displacement at point A, the upper end corner of the hook. The target value is 4.9366 inches.

Linear Static Analysis Results

All results are normalized with the target value.

Element Type	Mesh of elements n_w x n_l				
	1 x 9	**3 x 17**	**5 x 34**	**10 x 68**	**20 x 136**
CTRIA3	0.9191	0.9977	0.9906	0.9991	1.0086
CQUAD4	0.9792	1.0078	0.9968	1.0039	1.0116
CHEXA	0.5407	0.8966	0.9695	0.9893	1.0015

File Location

Proprietary Information of Altair Engineering. All Rights Reserved.

```
<install_directory>/demos/hwsolvers/optistruct/Rhex.fem
<install_directory>/demos/hwsolvers/optistruct/Rquad.fem
<install_directory>/demos/hwsolvers/optistruct/Rtri.fem
```

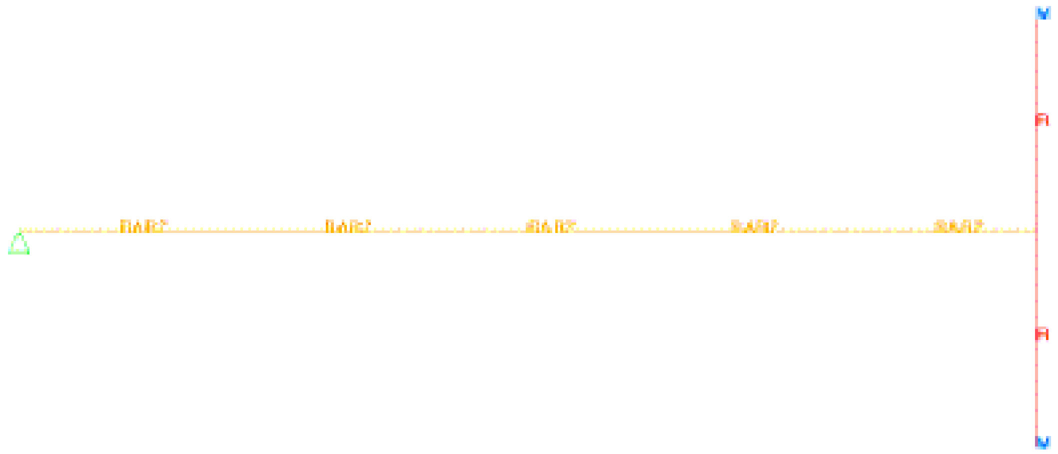
Reference

Knight, Jr. N. F., *Raasch Challenge for Shell Elements*, AIAA Journal, Vol. 35, No. 2, February 1997.

Keywords

FORCE.

46. Cantilever with Off-Center Point Masses (Test No. FV4)



Summary

Test No. FV4 is a cantilever beam with off-center point masses. The problem is set up to test the behavior of coupling between torsional and flexural, off-center inertial axis, and close eigenvalues. OptiStruct is used in this normal modes analysis.

Benchmark Model

The 2-node simple beam elements are used to model the problem. The cantilever beam consists of five elements. Two rigid elements are used to connect mass 1, 10000 kg, and mass 2, 1000 kg, at the end of the cantilever beam. All degrees of freedom at end A are constrained. The material properties for the cantilever beam are:

E = 200x10^9 N/m^2 and ρ = 8000 kg/m^3

Modal Analysis Results

The frequency of each targeted mode is normalized with the closed form solution.

```
f* = closed form solution
```

Proprietary Information of Altair Engineering. All Rights Reserved.

Mode 1		Mode 2		Mode 3	
f* CBAR	1.723 Hz 1.0054	f* CBAR	1.727 Hz 1.0048	f* CBAR	7.413 Hz 1.008
Mode 4		**Mode 5**		**Mode 6**	
f* CBAR	9.972 Hz 1.0055	f* CBAR	18.155 Hz 1.0067	f* CBAR	26.957 Hz 1.0068

File Location

`<install_directory>/demos/hwsolvers/optistruct/fv4.fem`

Reference

NAFEMS Ltd, *The Standard NAFEMS BENCHMARKS TNSB Rev. 3*, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

EIGRL.

Proprietary Information of Altair Engineering. All Rights Reserved.

47. Deep Simply-Supported Beam (Test No. FV5)

Summary

Test No. FV5 is a simply-supported beam problem. OptiStruct tests the repeated eigenvalues, shear deformation, and rotary inertia. The possibility of missing extensional modes is also investigated.

Benchmark Model

The 2-node beam elements are used to model the simply-supported beam which consists of five elements. The displacements in x, y, and z direction, as well as the rotation in x direction are fixed at the end A. In addition, the displacements in y and z direction are constrained at end B. The material properties for the beam are:

E = 200x10^9 N/m^2, ν = 0.3 and ρ = 8000 kg/m^3

Modal Analysis Results

The frequency of each targeted mode is normalized with the closed form solution.

```
f* = closed form solution
```

Proprietary Information of Altair Engineering. All Rights Reserved.

Modes 1 and 2 Flexural		Mode 3 Torsional		Mode 4 Extensional	
f* CBEAM	42.649 Hz 1.0124	f* CBEAM	77.542 Hz 0.9157	F* CBEAM	125.00 Hz 1.0000
Modes 5 and 6 Flexural		**Mode 7 Torsional**		**Modes 8 and 9 Flexural**	
f* CBEAM	148.31 Hz 1.0409	f* CBEAM	233.10 Hz 0.8840	f* CBEAM	284.55 Hz 1.0725

File Location

<install_directory>/demos/hwsolvers/optistruct/fv5.fem

Reference

NAFEMS Ltd, The Standard NAFEMS BENCHMARKS TNSB Rev. 3, NAFEMS Ltd, Scottish Enterprise Technology Park, Whitworth Building, East Kilbride, Glasgow, United Kingdom, 1990.

Keywords

EIGRL.

Proprietary Information of Altair Engineering. All Rights Reserved.