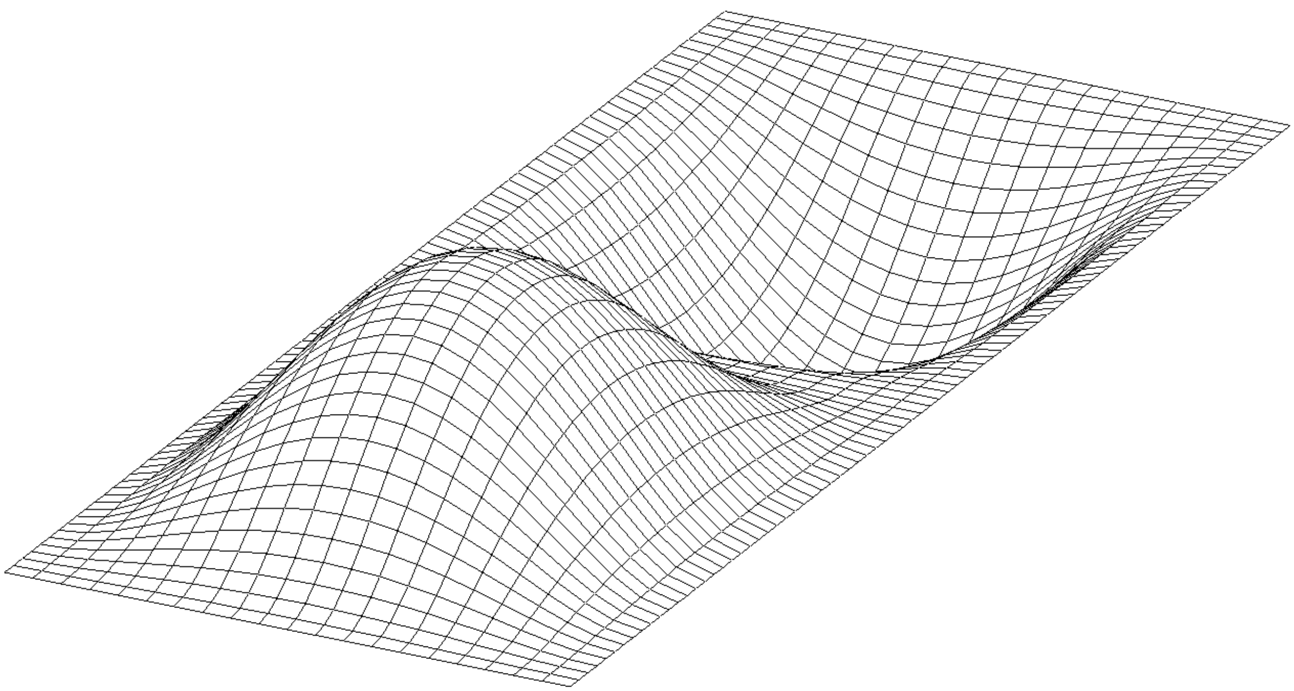
Autodesk Inventor Nastran 2022

Verification Manual



Autodesk® Inventor® Nastran® 2022

TABLE OF CONTENTS

1. Introduction

This guide contains verification test cases for the Autodesk® Inventor® Nastran® Finite Element Analysis solver. These test cases verify the functionality of Autodesk Inventor Nastran and encompass the different analysis types using theoretical and benchmark solutions from well-known engineering test cases. Each test case contains all test data needed to reproduce the given results. This guide contains test cases for:

- Linear Statics verification using theoretical solutions
- Normal Modes/Eigenvalue verification using theoretical solutions
- Normal Modes/Eigenvalue verification using standard NAFEMS benchmarks
- Verification Test Cases from the Société Francaise des Méchaniciens

1.1 Model Files Location

All the Autodesk Inventor Nastran model files (.nas) used for these test cases are located in this folder:

C:\Users\Public\Public Documents\Autodesk\Inventor Nastran 2021\Verification Models\en-us

2. Linear Statics Verification Using Theoretical Solutions

The purpose of these linear static test cases is to verify the functionality of Autodesk Inventor Nastran using theoretical solutions of well-known engineering linear static problems. The test cases are basic in form and have closed-form theoretical solutions.

The theoretical solutions given in these examples are from reputable engineering texts. For each case, a specific reference is cited. All theoretical reference texts are listed in Appendix A.

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

For most cases, discrepancies between Autodesk Inventor Nastran computed and theoretical results are minor and can be considered negligible. To produce exact results, for most cases, a larger number of elements would need to be used. Element quantity is chosen to achieve reasonable engineering accuracy in a reasonable amount of time.

2.1 Nodal Loads on a Cantilever Beam

Problem Description

Figure 1 shows the cantilever beam with a load acting on the free end. A static analysis is performed on the model. Beam deflection at the free end of the beam, and shear stress at the constrained end of the beam are determined. All dimensions are in inches.

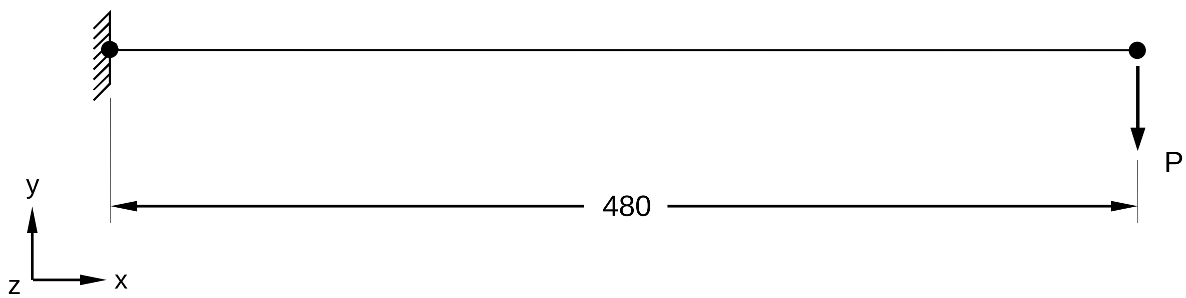


Figure 1. Cantilever Beam with Nodal Load

Autodesk Inventor Nastran Analysis Model Filename

- vm2_1.nas

Model Data

Finite Element Modeling

- Mesh (4 x 1): 5 nodes, 4 bar elements

Units

inch/pound/second

Model Geometry

Length: L = 480 in

Cross Sectional Properties

Area: A = 900 in^2
Square Cross Section = (30 in x 30 in)
Moment of Inertia: $I_y = I_z$ = 67500 in^4

Material Properties

Young's Modulus: E = 30.0 E+6 psi

Boundary Conditions

One end of the beam is constrained in all translations and rotations. A load P = 50,000 lb force in the negative Y-direction is set at the free end of the beam.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Beam Constrained End Z Shear Force Stress (psi)	5333.3	5333.3	0.0
Beam Free End T2 Translation (in)	0.9130*	0.9130	0.0

***Note:** The original theoretical value of 0.9102 neglects the shear deformation. The value for shear deformation is calculated below and added to the original theoretical value.

Post Processing

Shear Deformation $= \dfrac{VL}{AG}$

where: V = Shear Load, L = Length of the beam, A = Shear area = 0.8333 x cross section area, and G = modulus of rigidity

Shear deformation = 0.0028

Adding the shear deformation to the theory, T2 Translation = (Theory value + shear deformation)

T2 Translation = (0.9102 + 0.0028) = 0.9130 in

References

1. Beer and Johnston, *Mechanics of Materials*. New York: McGraw-Hill, Inc., 1992. p.716.

2.2 Axial Distributed Load on a Linear Beam

Problem Description

Figure 1 shows the model of the linear beam. A static analysis is performed using an axially distributed load. The beam axial stress at the constrained end (A), deflection at the free end (B), and the constraint force at the constrained end of the beam are determined. All dimensions are in inches.

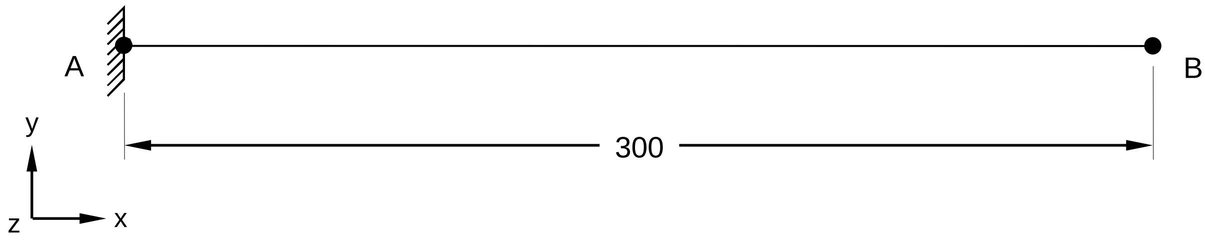


Figure 1. Linear Beam with Axial Distributed Load

Autodesk Inventor Nastran Analysis Model Filename

- vm2_2.nas

Model Data

Finite Element Modeling

- Mesh (30 x 1): 31 nodes, 30 bar elements

Units

inch/pound/second

Model Geometry

Length: L = 300 in

Cross Sectional Properties

Area: A = 9 in^2
Square Cross Section = (3 in x 3 in)
Moment of Inertia: I = 6.75 in^4

Material Properties

Young's Modulus: E = 30.0 E+6 psi

Boundary Conditions

One end of the beam is constrained in all translations and rotations (point A). An axially distributed load (force per unit length) is set to 1,000 lb/in in the negative Y-direction for the 10-inch long element furthest from the constrained end (at point B). See Figure 2.

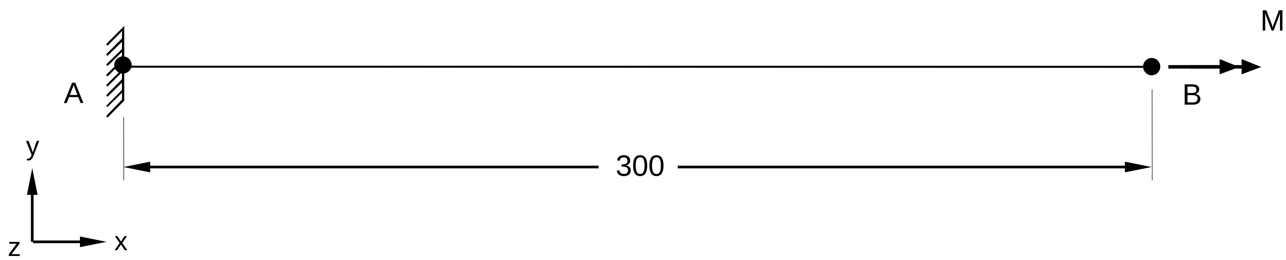


Figure 2. Boundary Conditions

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Beam Constrained End Axial Stress (psi)	1,111.1	1,111.1	0.0
Beam Free End T1 Translation (in)	0.011111	0.010926	1.7
Beam T1 Constraint Force (lb)	-10,000	-10,000	0.0

References

1. Beer and Johnston, *Mechanics of Materials*. New York: McGraw-Hill, Inc., 1992.

2.3 Distributed Loads on a Cantilever Beam

Problem Description

Figure 1 shows the model of a cantilever beam with a distributed load acting in the negative Y-direction. A static analysis is performed on the model. The beam torque stress at the constrained end of the beam (A), the deflection of the free end (B), and the total beam constraint force are determined. All dimensions are in inches.

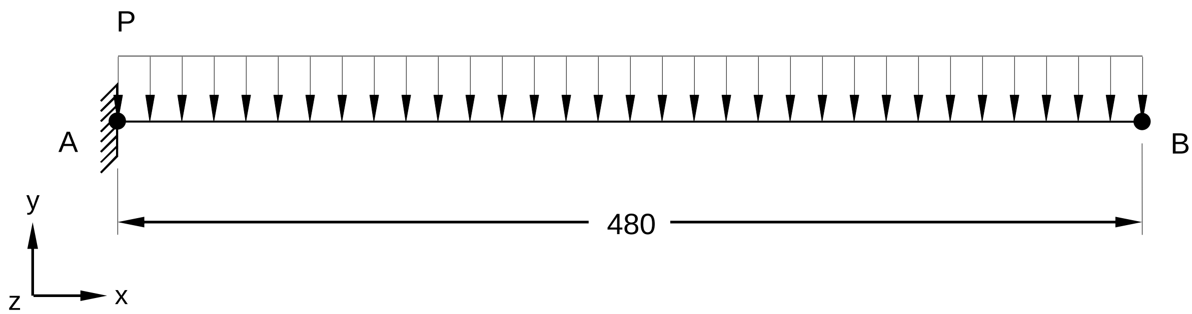


Figure 1. Cantilever Beam with Distributed Load

Autodesk Inventor Nastran Analysis Model Filename

- vm2_3.nas

Model Data

Finite Element Modeling

- Mesh (8 x 1): 9 nodes, 8 bar elements

Units

inch/pound/second

Model Geometry

Length: L = 480 in

Cross Sectional Properties

Area: A = 900 in^2
Square Cross Section = (30 in x 30 in)
Moment of Inertia: $I_y = I_z$ = 67500 in^4

Material Properties

Young's Modulus: E = 30.0 E+6 psi

Boundary Conditions

One end of the beam is constrained in all translations and rotations. A distributed load P = 250 lb/in in the negative Y-direction is defined on all elements.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Beam Constrained End Torque Stress (psi)	6,400	6,400	0.00
Beam Free End Total Translation (in)	0.8192*	0.8192	0.00
Beam Total Constraint Force (lb)	120,000	120,000	0.00

***Note:** The theoretical value neglects the shear deformation.

References

1. Beer and Johnston, *Mechanics of Materials*. New York: McGraw-Hill, Inc., 1992.

2.4 Moment Load on a Cantilever Beam

Problem Description

Figure 1 shows the model of a cantilever beam with moment load acting on the free end of the beam. A static analysis is performed on the model. The beam bending stress at the constrained end of the beam, the deflection at the free end, and the reaction force at the constrained end are determined. All dimensions are in inches.

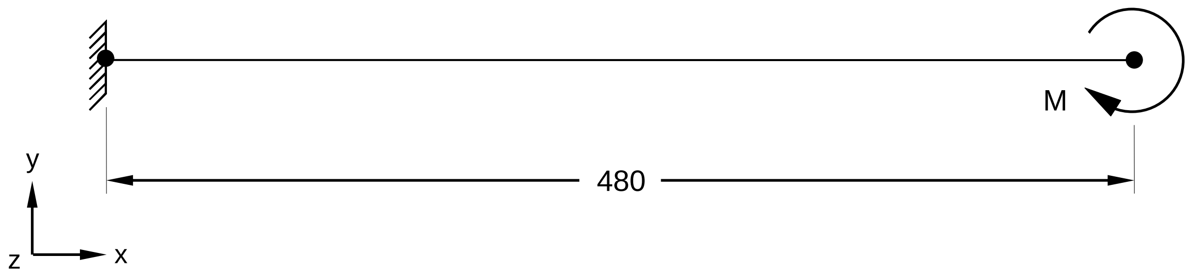


Figure 1. Cantilever Beam with Moment Load

Autodesk Inventor Nastran Analysis Model Filename

- vm2_4.nas

Model Data

Finite Element Modeling

- Mesh (8 x 1): 9 nodes, 8 bar elements

Units

inch/pound/second

Model Geometry

Length: L = 480 in

Cross Sectional Properties

Area: A = 900 in^2
Square Cross Section = (30 in x 30 in)
Moment of Inertia: $I_y = I_z$ = 67500 in^4

Material Properties

Young's Modulus: E = 30.0 E+6 psi

Boundary Conditions

One end of the beam is constrained in all translations and rotations. A M = 2.5 E+6 in-lb is set at the free end of the beam in the Z-direction.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Beam Constrained End Z Shear Force Stress (psi)	555.6	555.6	0.0
Beam Free End Total Translation (in)	0.1422	0.1422	0.0
Beam Constrained End Total Constraint Moment (lb)	2,500,000	2,500,000	0.0

References

1. Beer and Johnston, *Mechanics of Materials*. New York: McGraw-Hill, Inc., 1992.

2.5 Thermal Strain, Displacement, and Stress on a Heated Beam

Problem Description

Figure 1 shows the model of a heated beam. A static analysis is performed on the model. In one case of this example, the beam is constrained at one end. For the second case, the beam is constrained at both ends (see Figure 2). The beam axial stress at the constrained end of the beam, the deflection at the free end, and the constraint force at the constrained end of the beam are determined. All dimensions are in meters.

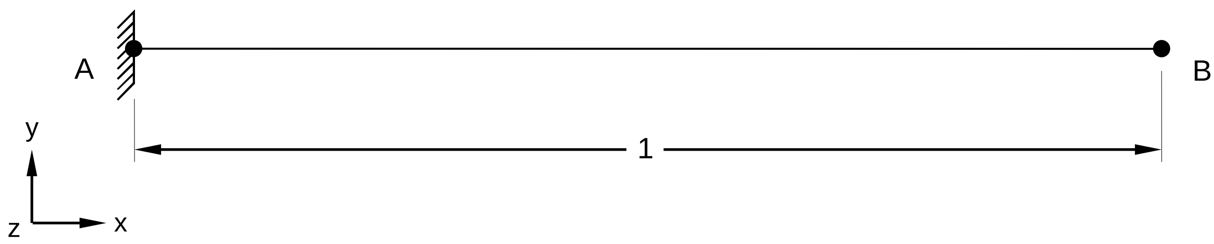


Figure 1. Heated Beam with Thermal Strain, Displacement, and Stress (Case 1)

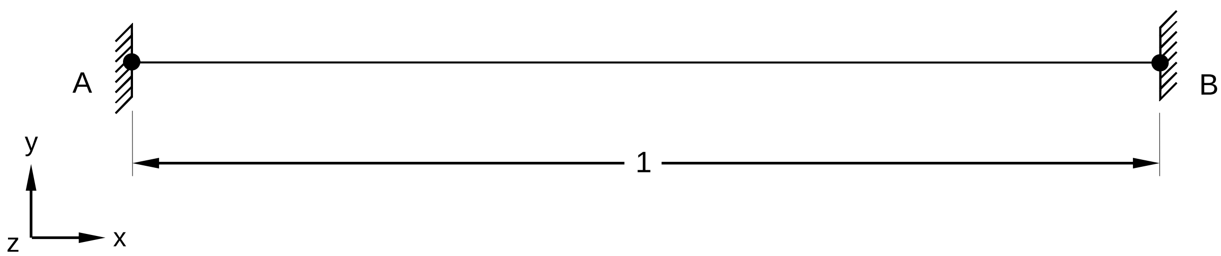


Figure 2. Heated Beam with Thermal Strain, Displacement, and Stress (Case 2)

Autodesk Inventor Nastran Analysis Model Filename

- vm2_5a.nas – Case 1 (beam constrained at one end)
- vm2_5b.nas – Case 2 (beam constrained at both ends)

Model Data

Finite Element Modeling

- Mesh (10 x 1): 11 nodes, 10 bar elements

Units

meter/Newton/second

Model Geometry

Length: L = 1 m

Cross Sectional Properties

Area: $A = 0.01\ m^2$

Material Properties

Young's Modulus: E = 2.068 E+11 Pa
Thermal Expansion Coefficient: α = 1.2 E-05 m/(m °C)
Poisson's Ratio: ν = 0.3

Boundary Conditions

Case 1: One end of the beam is constrained in all translations and rotations. The temperature on all nodes is set to 25°C. The reference temperature is set to -50°C.

Case 2: Both ends of the beam are constrained in all translations and rotations. The temperature on all nodes is set to 25°C. The reference temperature is set to -50°C.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1 and Table 2.

Table 1. Results for Case 1

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Beam Free End Total Translation (m)	0.0009	0.0009	0.0
Beam Constrained End Axial Strain	0.0009	0.0009	0.0

Table 2. Results for Case 2

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Beam Total Translation (m)	0.0	0.0	0.0
Beam Constrained End Total Constraint Force (kN)	1,860	1,860	0.0
Beam Constrained End Axial Stress (MPa)	-186	-186	0.0

References

1. Beer and Johnston, *Mechanics of Materials.* New York: McGraw-Hill, Inc., 1992.

2.6 Uniformly Distributed Load on a Linear Beam

Problem Description

Figure 1 shows the model of a beam with distributed loads acting on both ends and constraints acting on the second node in from either end (B and D). A static analysis is performed on the model. The beam torque stress at the center of the beam and the deflection at the center of the beam are determined. All dimensions are in inches.

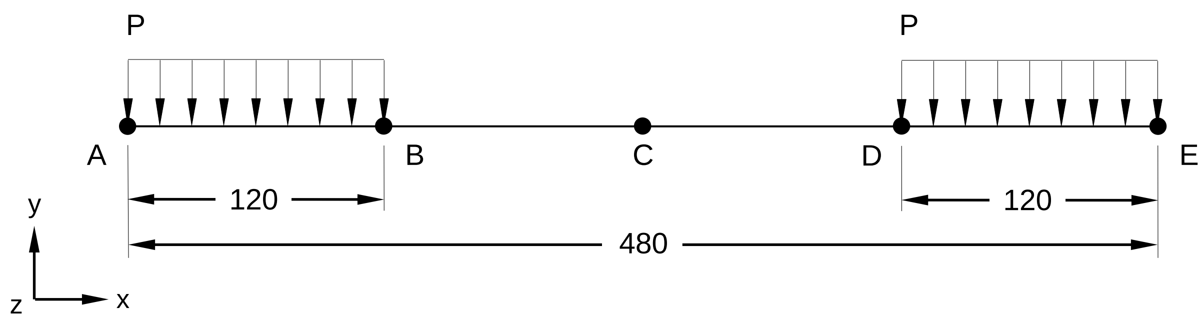


Figure 1. Linear Beam with Uniformly Distributed Load

Autodesk Inventor Nastran Analysis Model Filename

- vm2_6.nas

Model Data

Finite Element Modeling

- Mesh (4 x 1): 5 nodes, 4 bar elements

Units

inch/pound/second

Model Geometry

Length: L = 480 in

Cross Sectional Properties

Area: A = 50.5908 in^2
Rectangular Cross Section = (1.17 in x 43.24 in)
Moment of Inertia: I_z = 7892 in^4

Material Properties

Young's Modulus: E = 30.0 E+6 psi

Boundary Conditions

The second node in from either end of the beam (B and D, as shown in Figure 1) is constrained in all translations and X and Y-rotations. Rotation about Z is not constrained. A distributed load P = 833 lb/in is defined in the negative Y-direction on the end elements of the beam.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Beam Center Total Translation (in)	0.182	0.182	0.0
Beam Center Torque Stress (psi)	16,439	16,439	0.0

References

1. Timoshenko, S., *Strength of Materials, Part 1, Elementary Theory and Problems*, 3rd Edition. New York: D. VanNostrand Co., Inc., 1955. p.98.

2.7 Membrane Loads on a Plate

Problem Description

Figure 1 shows the plate with membrane loads acting in the plane of the plate. Static analysis is performed on the model. The change in length of diameter AB and of diameter CD are determined. All dimensions are in inches.

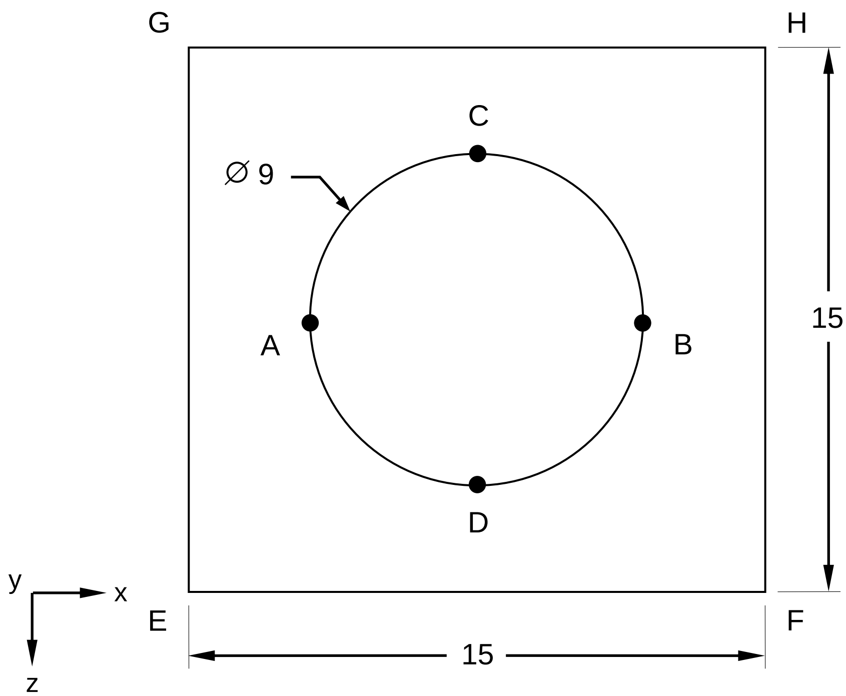


Figure1. Plate with Membrane Loads

Autodesk Inventor Nastran Analysis Model Filename

- vm2_7.nas

Model Data

Finite Element Modeling

- A ¼ of the model is created (the bottom right-hand corner that contains points B and D). Symmetry boundary conditions are applied. The answer is then multiplied by 2 for correct results.

Units

inch/pound/second

Model Geometry

Length: L = 15 in
Diameter: *AB, CD* = 9 in
Thickness: t = 3/4 in

Material Properties

Young's Modulus: E = 10.0 E+6 psi
Poisson's Ratio: ν = 0.3
F(x)/l = 9,000 lb/in
F(z)/l = 15,000 lb/in

Boundary Conditions

The model is constrained using symmetry boundary conditions (on edges EG and GH, as shown in Figure 1). The elemental edge load is set to 9,000 lb/in in the X-direction (on edge FH) and 15,000 lb/in in the negative Y-direction (on edge EF).

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodek Nastran**	**Error** (%)
T1 Translation (in)	0.0048	0.0048	0.0
T3 Translation (in)	0.0144	0.0144	0.0

References

1. Beer and Johnston, *Mechanics of Materials*. New York: McGraw-Hill, Inc., 1992. p.85.

2.8 Thin Wall Cylinder in Pure Tension

Problem Description

Figure 1 shows the thin wall cylinder with uniform axial loads. Static analysis is performed on the model. The stress and deflection of the thin wall cylinder is determined. All dimensions are in inches. The wall thickness is 0.01 inches.

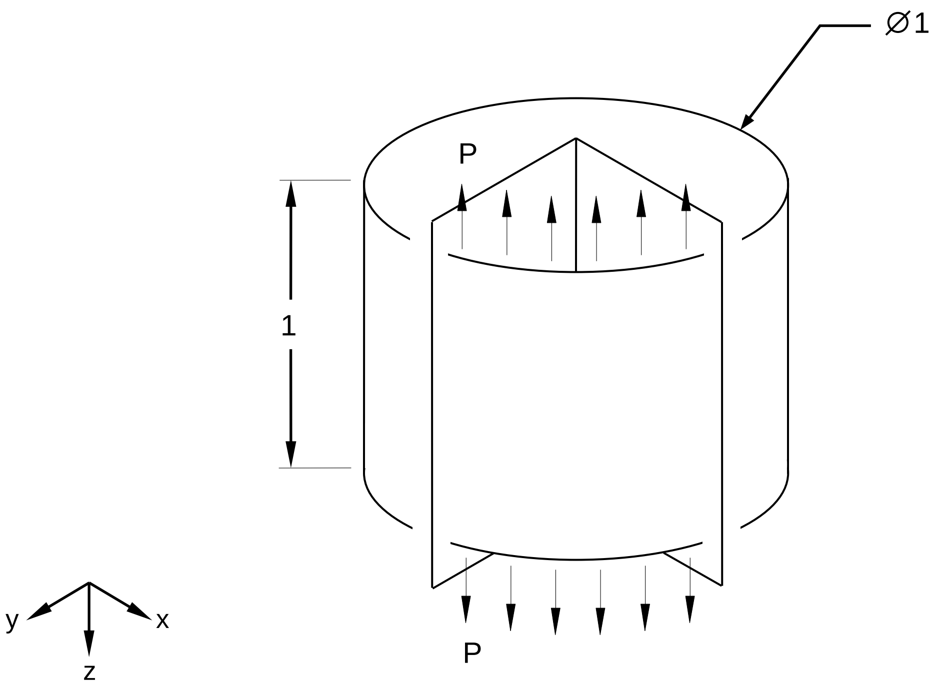


Figure 1. Thin Wall Cylinder in Pure Tension

Autodesk Inventor Nastran Analysis Model Filename

- vm2_8.nas

Model Data

Finite Element Modeling

- 25 nodes. A ¼ model is created with 16 5-DOF/node quadrilateral plate elements and symmetry boundary conditions.

Units

inch/pound/second

Model Geometry

Radius: R = 0.5 in
Thickness: t = 0.01 in
Height: h = 1.0 in

Material Properties

Young's Modulus: E = 10.0 E+3 psi
Poisson's Ratio: ν = 0.3

Boundary Conditions

The quarter model is constrained using symmetry boundary conditions. The nodes on the edge opposite to the loaded edge are constrained in the Z-translation. The nodal force P/(π*D) = 3.1831, where P = 10 psi, is applied to nodes 21, 25 as 0.9757 lbs and nodes 22, 23, 24 as 1.9509 lbs.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	Theory	Autodesk Inventor Nastran	Error (%)
Top Y Normal Stress (psi)	1,000.0	1,000.0	0.0
T3 Translation (in)	0.1	0.1	0.0
T1 Translation (in)	-0.015	-0.015	0.0

References

1. Roark, R. and Young, W., *Formulas for Stress and Strain*, 6th Edition. New York: McGraw-Hill Book Co., 1989. p. 518, Case 1a.

2.9 Thin Shell Beam Wall in Pure Bending

Problem Description

Figure 1 shows the thin shell beam wall. A distributed load is acting on the free end (edge CD). Static analysis is performed on the model. Maximum stress at the constrained end, maximum deflection at the free end, and the beams strain energy are determined. All dimensions are in inches.

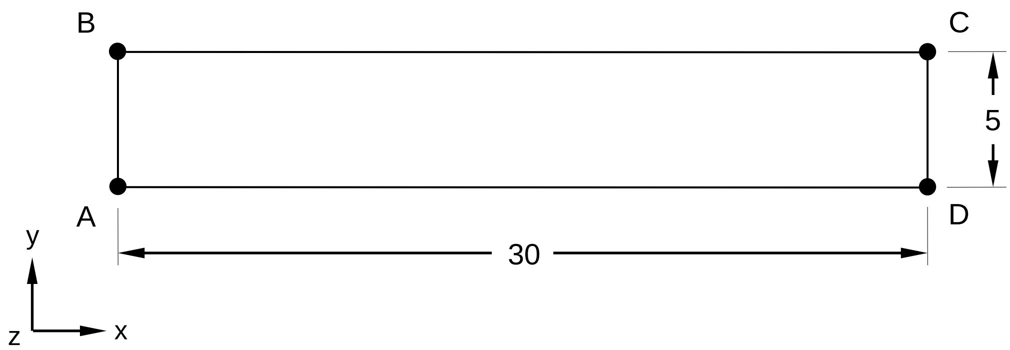


Figure 1. Thin Shell Beam Wall in Pure Bending

Autodesk Inventor Nastran Analysis Model Filename

- vm2_9.nas

Model Data

Finite Element Modeling

- 14 nodes, 6 5-DOF/node quadrilateral plate elements

Units

inch/pound/second

Model Geometry

Length: L = 30 in
Width: w = 5 in
Thickness: t = 0.1 in

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Poisson's Ratio: ν = 0.3

Boundary Conditions

The nodes at end AB (nodes 7 and 14) are constrained in all translations and rotations. Distributed load as nodal forces of p/w = 1.2 lbs/in, where p = 6.0 lb, are applied to the opposite end of the beam, at end CD (nodes 1 and 8), acting in the negative Z-direction.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	Theory	Autodesk Inventor Nastran	Error (%)
T3 Translation (in)	4.320	4.255	1.5
Plate Bottom Major Stress (psi)	21,600	20,535	4.9
Total Strain Energy (lb-in)	12.96	12.77	1.5

References

1. Shigley, J. and Mitchel L., *Mechanical Engineering Design*, 4th Edition. New York: McGraw-Hill, Inc., 1983. pp. 134, 804.

2.10 Strain Energy of a Truss

Problem Description

Figure 1 shows the truss with a load acting on the center, at point D. Static analysis is performed on the truss. The strain energy of the truss is determined. All dimensions are in inches.

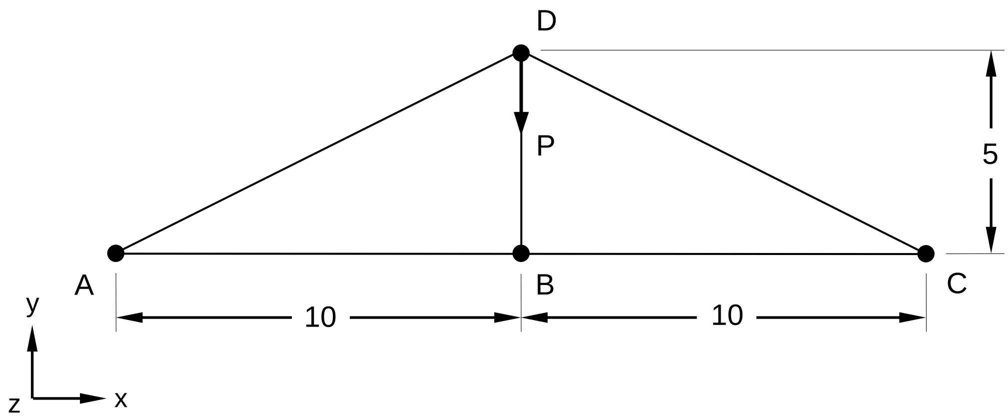


Figure 1. Truss Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_10.nas

Model Data

Finite Element Modeling

- 4 nodes, 5 rod elements

Units

inch/pound/second

Model Geometry

Length: L = 10 in

Cross Sectional Properties

Area: A = 0.01 in^2

Material Properties

Young's Modulus: E = 30.0 E+6 psi

Boundary Conditions

The bottom left corner (end A) of the truss is constrained in the X, Y, and Z-translations and the X and Y rotations. The bottom right corner (end C) of the truss is constrained in the Y and Z-translations and the X and Y-rotations. The top of the truss (point D) is loaded with a nodal force P = 300 lb in the negative Y-direction.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	Theory	Autodesk Inventor Nastran	Error (%)
Truss Total Strain Energy (lb-in)	5.846	5.846	0.0

References

1. Beer and Johnston, *Mechanics of Materials*. New York: McGraw –Hill, Inc., 1992. p. 588.

2.11 Flat Square Plate

Problem Description

Figure 1 shows the flat square plate. Static analysis is performed on the plate using two different plate thicknesses. The maximum displacement is determined. All dimensions are in inches.

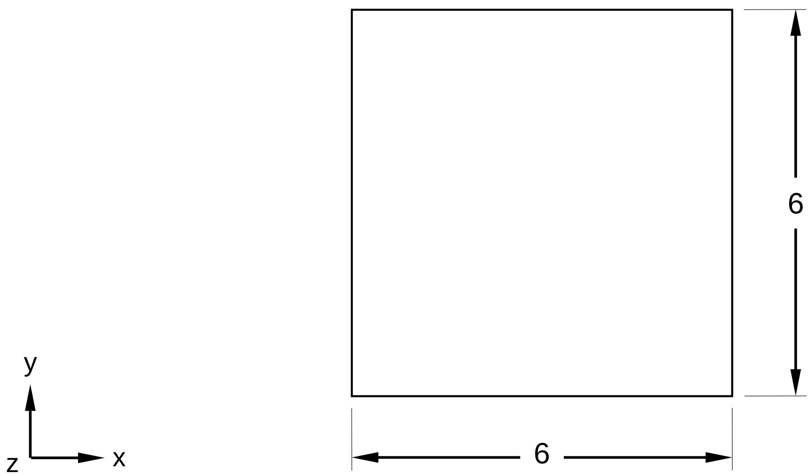


Figure 1. Flat Square Plate Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_11a.nas – thick plate
- vm2_11b.nas – thin plate

Model Data

Finite Element Modeling

- A ¼ model is created with 16 5-DOF/node quadrilateral plate elements and symmetry boundary conditions where applicable.

Units

inch/pound/second

Model Geometry

Length: L = 6 in
Width: w = 6 in
Thickness: t = 0.25 in (vm2_11a) and t = 0.0025 in (vm2_11b)

Material Properties

Young's Modulus: E = 29.0 E+6 psi

Boundary Conditions

The quarter model is constrained using symmetry boundary conditions. A pressure load of 100 psi is applied on the thick plate (vm2_11a) and a pressure load of 0.0001 psi is applied on the thin plate (vm2_11b).

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1 for the thick plate example and in Table 2 for the thin plate example.

Table 1. Results Thick Plate

Description	Theory	Autodesk Inventor Nastran	Error (%)
Max Displacement in Z-direction (in)	3.94E-3	4.01E-3	1.8

Table 2. Results Thin Plate

Description	Theory	Autodesk Inventor Nastran	Error (%)
Max Displacement in Z-direction (in)	3.94E-3	3.78E-3	4.1

References

1. Roark, R. and Young, W., *Formulas for Stress and Strain*, 6th Edition. New York: McGraw-Hill Book Co., 1989. p. 508.

2.12 The Raasch Challenge Problem for Shell Elements

Problem Description

Figure 1 shows the clamped Raasch Hook with a tip in-plane shear load. The structure consists of two cylindrical shells with different curvatures. Static analysis is performed on the plate using five different mesh sizes to check the results convergence and to be consistent with the meshes presented by Knight (1997). The maximum displacement is determined at the tip in the direction of the load. All dimensions are in inches.

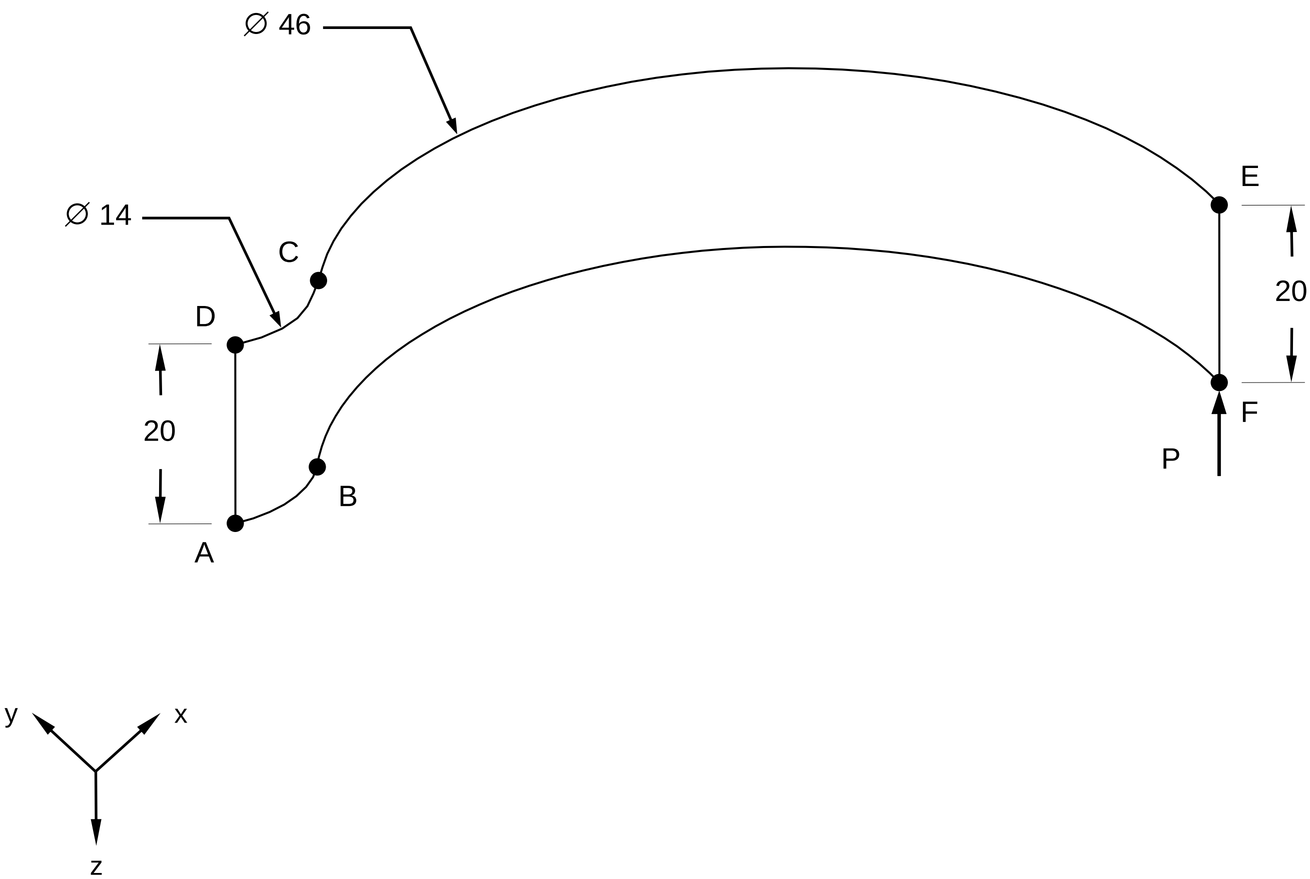


Figure 1. Raasch Hook Plate Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_12a4.nas – 5-DOF/node, CQUAD4 elements
- vm2_12aR.nas – 6-DOF/node, CQUADR elements
- vm2_12b4.nas – 5-DOF/node, CQUAD4 elements
- vm2_12bR.nas – 6-DOF/node, CQUADR elements
- vm2_12c4.nas – 5-DOF/node, CQUAD4 elements
- vm2_12cR.nas – 6-DOF/node, CQUADR elements
- vm2_12d4.nas – 5-DOF/node, CQUAD4 elements
- vm2_12dR.nas – 6-DOF/node, CQUADR elements
- vm2_12e4.nas – 5-DOF/node, CQUAD4 elements
- vm2_12eR.nas – 6-DOF/node, CQUADR elements

Model Data

Finite Element Modeling

- Test vm2_12a4: 1x9 mesh, 20 nodes, 9 5-DOF/node quadrilateral plate elements
- Test vm2_12aR: 1x9 mesh, 20 nodes, 9 6-DOF/node quadrilateral plate elements
- Test vm2_12b4: 3x17 mesh, 90 nodes, 51 5-DOF/node quadrilateral plate elements
- Test vm2_12bR: 3x17 mesh, 90 nodes, 51 6-DOF/node quadrilateral plate elements
- Test vm2_12c4: 5x34 mesh, 248 nodes, 170 5-DOF/node quadrilateral plate elements
- Test vm2_12cR: 5x34 mesh, 248 nodes, 170 6-DOF/node quadrilateral plate elements
- Test vm2_12d4: 10x68 mesh, 828 nodes, 680 5-DOF/node quadrilateral plate elements
- Test vm2_12dR: 10x68 mesh, 828 nodes, 680 6-DOF/node quadrilateral plate elements
- Test vm2_12e4: 20x136 mesh, 3014 nodes, 2720 5-DOF/node quadrilateral plate elements
- Test vm2_12eR: 20x136 mesh, 3014 nodes, 2720 6-DOF/node quadrilateral plate elements

Units

inch/pound/second

Model Geometry

Radius: R_1 = 14 in and R_2 = 46 in
Width: w = 20 in
Thickness: t = 2 in

Material Properties

Young's Modulus: E = 3300.0 psi
Poisson's Ratio: ν = 0.35

Boundary Conditions

One end is fixed in all translations and rotations (edge AB from Figure 1). A unit load (1 lb) is applied at the other end as tip in-plane shear using a rigid body element (RBE2).

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

		Bench Value*	Autodesk Inventor Nastran	
Mesh Size	**Element**	**Displacement z** (in)	**Displacement z** (in)	**Error** (%)
1x9	CQUAD4	5.02	4.45	11.4
	CQUADR	5.02	4.39	12.5
3x17	CQUAD4	5.02	4.97	1.0
	CQUADR	5.02	4.92	1.2
5x34	CQUAD4	5.02	5.01	0.2
	CQUADR	5.02	4.97	1.0
10x68	CQUAD4	5.02	5.06	0.8
	CQUADR	5.02	5.03	0.2
20x136	CQUAD4	5.02	5.09	1.4
	CQUADR	5.02	5.08	1.2

***Note:** The bench value of *5.02 in* is cited by Knight (1997) and is based on tests performed using various types of finite elements. An analytical solution for the shell model of the hook is not available. According to Wlassow (1964), the analytical solution is based on the theory of curved beams and it gives a stiffer solution for the deflection, *4.7561 in*.

References

1. Knight, N. F., *The Raasch Challenge for Shell Elements*, AIAA Journal, Volume 35, No.2, 1997. pp. 375-381.
2. Wlassow, W.S., Dunnwandige Elastische Stabe Band 2, VEB Verlag fur das Bauwesen Berlin, 1964.

2.13 Twisted Beam Static Load

Problem Description

Figure 1 shows the twisted beam. Static analysis is performed on the plate using two different load cases. The maximum displacement is determined at the tip of the load in the direction of the load in-plane and out-of-plane, respectively. All dimensions are in inches.

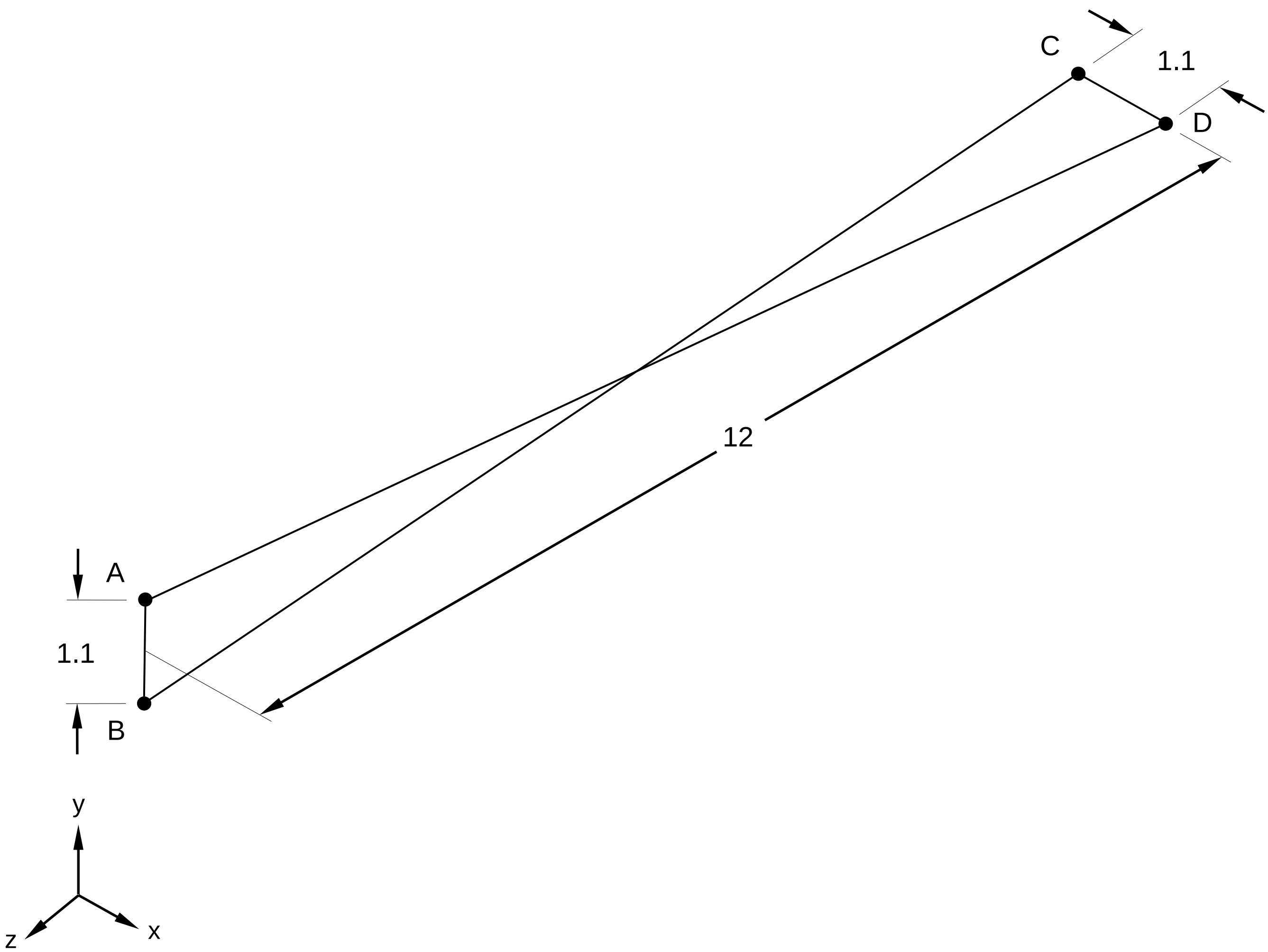


Figure 1. Twisted Beam Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_13.nas

Model Data

Finite Element Modeling

- 441 nodes, 384 5-DOF/node quadrilateral plate elements

Units

inch/pound/second

Model Geometry

Length: L = 12.00 in
Width: w = 1.10 in
Thickness: t = 0.32 in
Twist: θ = 90 (root to tip)

Material Properties

Young's Modulus: E = 29.0 E+6 psi
Poisson's Ratio: ν = 0.22

Boundary Conditions

The model is constrained at one end in all translations and all rotations (edge CD). A unit load of 1 lb is applied at tip (edge AB) in-plane (Y-direction) and out-of-plane (in the X-direction), respectively.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Max Displacement in Y-direction (in) – loading in-plane	5.424E-3	5.513E-3	1.6
Max Displacement in X-direction (in) – loading out-of-plane	1.754E-3	1.779E-3	1.4

References

1. MacNeal, R. H. and Harder, R. C., *A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Element in Analysis and Design 1*. North Holland, 1985.

2.14 Plane Frame with Beam Span Loads

Problem Description

Figure 1 shows the plane frame. Static analysis is performed on the 2D frame using two different load cases. The bending moments and shear forces at mid-point on the first bay for each load case are determined. All dimensions are in feet.

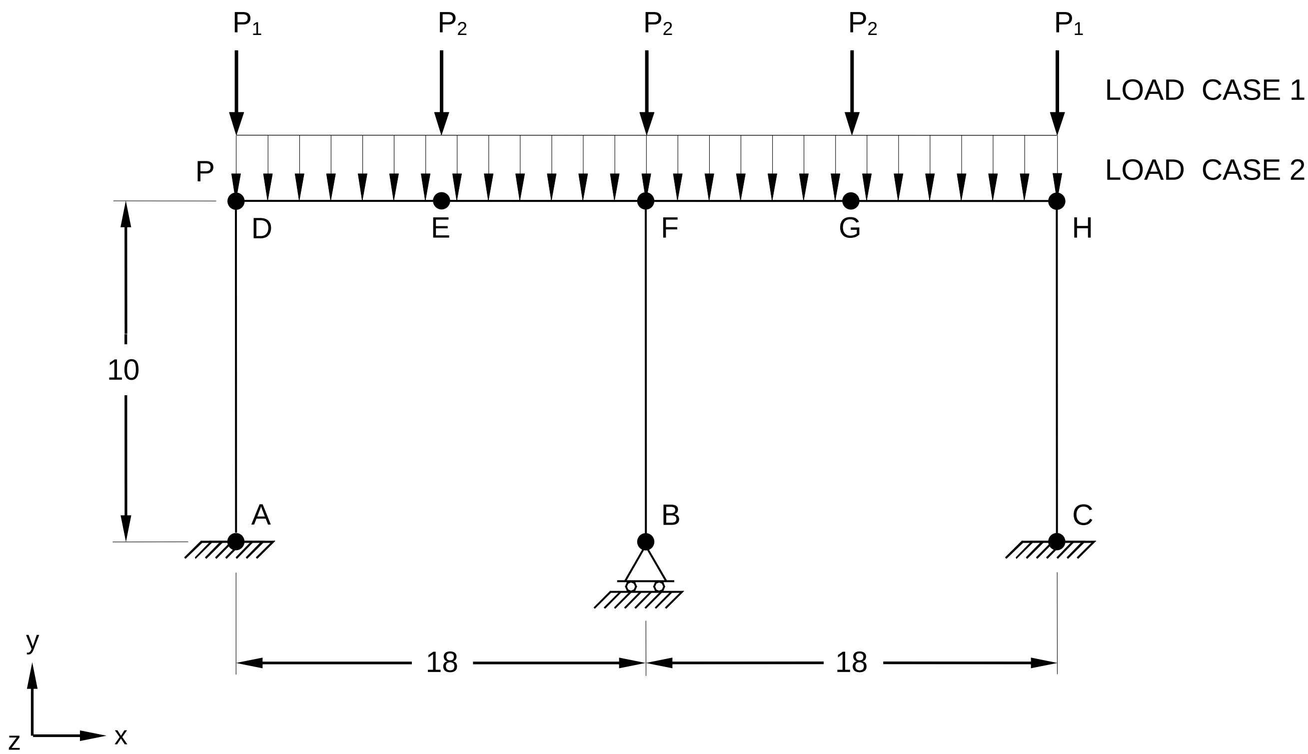


Figure 1. Plane Frame Model with Beam Span Loads

Autodesk Inventor Nastran Analysis Model Filename

- vm2_14.nas

Model Data

Finite Element Modeling

- 8 nodes, 7 beam elements

Units

inch/Kip/second

Model Geometry

Length: L = 216 in (18 ft)
Height: h = 120 in (10 ft)

Cross Sectional Properties

Area: A = 1.0E+7 in^2 for both columns and beams
Rectangular Cross Section = (12 in x 24 in) for columns
Rectangular Cross Section = (12 in x 30 in) for beams
Moment of Inertia: I_z = 13824 in^4 for columns
Moment of Inertia: I_z = 27000 in^4 for beams

Material Properties

Young's Modulus: E = 30.0 E+3 ksi
Shear Modulus of Elasticity: G = 15.0 E+3 ksi

Boundary Conditions

The model is constrained at ends A and C in all translations and all rotations. The middle column is pinned at the base, node B (all translations retrained). In addition, the beams at the connection to the left and right columns have the strong moment released. For the first load case, a load P1 of 50 Kip is applied at nodes D and H, and another load P2 of 100 Kip is applied at nodes E, F, and G (negative Y-direction). The second load case is a uniformly distributed load of 10 Kip/ft on both beams, in the negative Y-direction.

Solution Type

Static

Comparison of Results

Tabular results are given in Tables 1 and 2.

Table 1. Test Case 1 Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Moment (k-in)	3375	3375	0.0
Shear (k)	68.75	68.75	0.0

Table 2. Test Case 2 Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Moment (k-in)	2430	2430	0.0
Shear (k)	22.50	22.50	0.2

References

1. *Manual of Steel Construction-Allowable Stress Design*, AISC. Chicago, Illinois, 1989.

2.15 Thermal Gradient Loads on a Beam

Problem Description

Figure 1 shows the cantilever beam. Static analysis is performed on the beam. The bending moment at support and maximum displacement are determined. All dimensions are in inches.

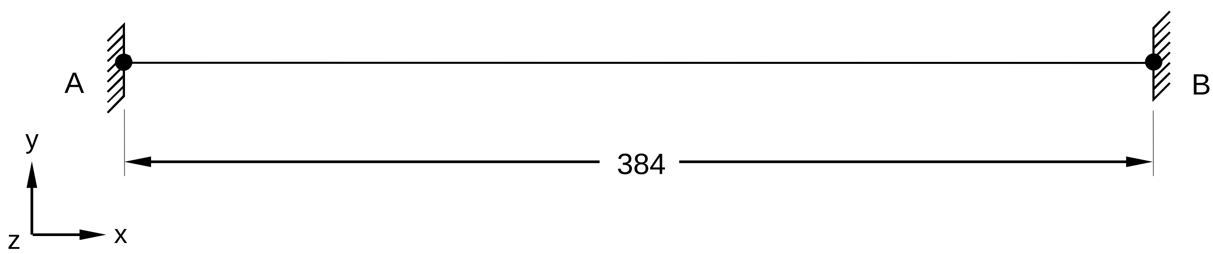


Figure 1. Beam Model with Thermal Gradient Loads

Autodesk Inventor Nastran Analysis Model Filename

- vm2_15.nas

Model Data

Finite Element Modeling

- 7 nodes, 6 bar elements

Units

inch/Kip/second

Model Geometry

Length: L = 384 in

Cross Sectional Properties

Area: A = 144 in^2
Square Cross Section = (12 in x 12 in)
Moment of Inertia: I = 1728 in^4

Material Properties

Young's Modulus: E = 3150.0 ksi
Poisson's Ratio: ν = 0.17
Thermal Expansion Coefficient: α = 5.5 E-06 in/in/deg F

Boundary Conditions

The model is constrained at both ends in all translations and all rotations. A thermal gradient of 120º F is defined across the depth of the beam for all beams.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Maximum Translation (in)	0.0	0.0	0.0
Restraint Moment (Kip-in)	299.4	299.4	0.0

References

1. Roark, R. and Young, W., *Formulas for Stress and Strain*, 6th Edition. New York: McGraw-Hill Book Co., 1989. p. 112.

2.16 Statically Indeterminate Reaction Force Analysis

Problem Description

Figure 1 shows the beam model. Static analysis is performed on the beam. The reactions at nodes 1 and 4 (A and D) are determined. All dimensions are in inches.

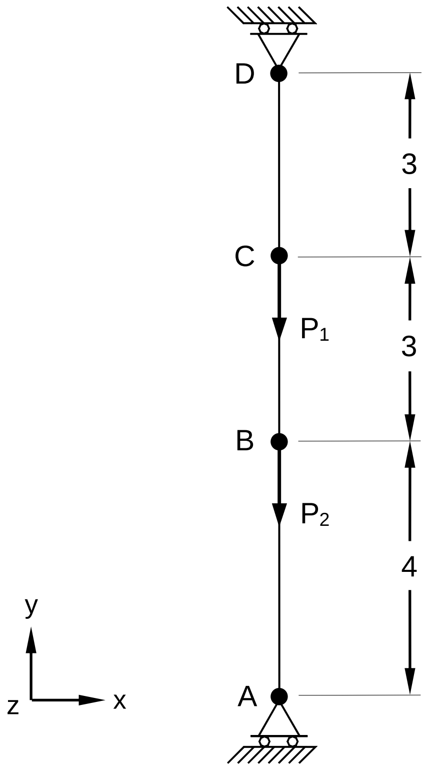


Figure 1. Beam Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_16.nas

Model Data

Finite Element Modeling

- 4 nodes, 3 bar elements

Units

inch/pound/second

Model Geometry

Length: L = 10 in

Cross Sectional Properties

Area: A = 1 in^2
Square Cross Section = (1 in x 1 in)

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Shear Modulus of Elasticity: G = 15.0 E+6 psi

Boundary Conditions

The model is constrained at both ends (A and D) in all translations and rotations in X and Y-directions. A load P_1 = 1,000 lb is applied at node C, and another load P_2 = 500 lb is applied at node B in the negative Y-direction.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Reaction R1 (lb)	600	600	0.0
Reaction R2 (lb)	900	900	0.0

References

1. Timoshenko, S., *Strength of Materials, Part 1, Elementary Theory and Problems*, 3rd Edition. New York: D. VanNostrand Co., Inc., 1955.

2.17 Beam Stresses and Deflection

Problem Description

Figure 1 shows the beam model. Static analysis is performed on the beam. The maximum stress in the middle of the beam and maximum deflection at center of the beam are determined. All dimensions are in inches.

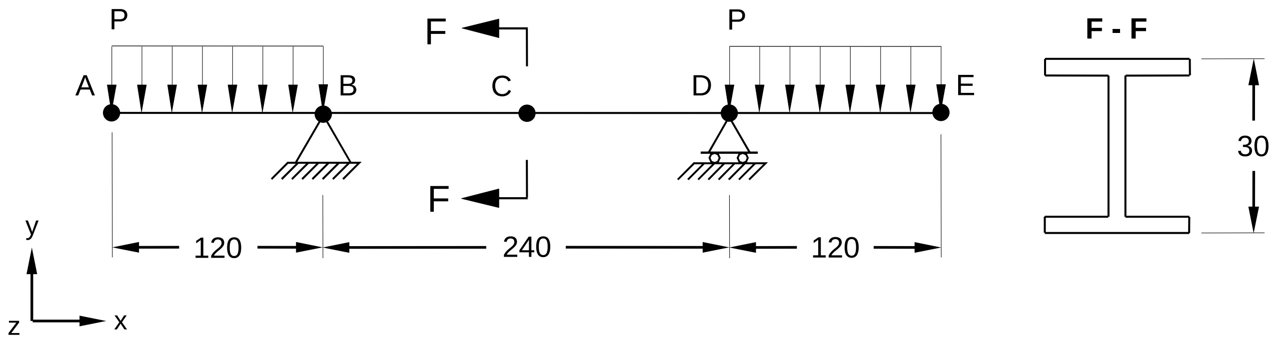


Figure 1. Beam Stresses and Deflection Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_17.nas

Model Data

Finite Element Modeling

- 5 nodes, 4 beam elements

Units

inch/pound/second

Model Geometry

Length: L = 480 in
Height: h = 30 in

Cross Sectional Properties

Area: A = 50.65 in^2
Moment of Inertia: I_z = 7892 in^4

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Shear Modulus of Elasticity: G = 15.0 E+6 psi

Boundary Conditions

One joint is pinned (at point B) and the other one is a roller (at point D). Points A, C and E are constrained in the Z-translation, and X and Y-rotations. A uniform distributed load P = 833.33 lb/in is applied on each cantilever in the negative Y-direction.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Maximum Bending Stress (psi)	11,404	11,404	0.0
Maximum Deflection (in)	0.182	0.182	0.0

References

1. Timoshenko, S., *Strength of Materials, Part 1, Elementary Theory and Problems*, 3rd Edition. New York: D. VanNostrand Co., Inc., 1955.

2.18 Laterally Loaded Tapered Support Structure Case 1

Problem Description

Figure 1 shows the beam model. Static analysis is performed on the beam. The maximum bending stress at mid span (point B) is determined. All dimensions are in inches.

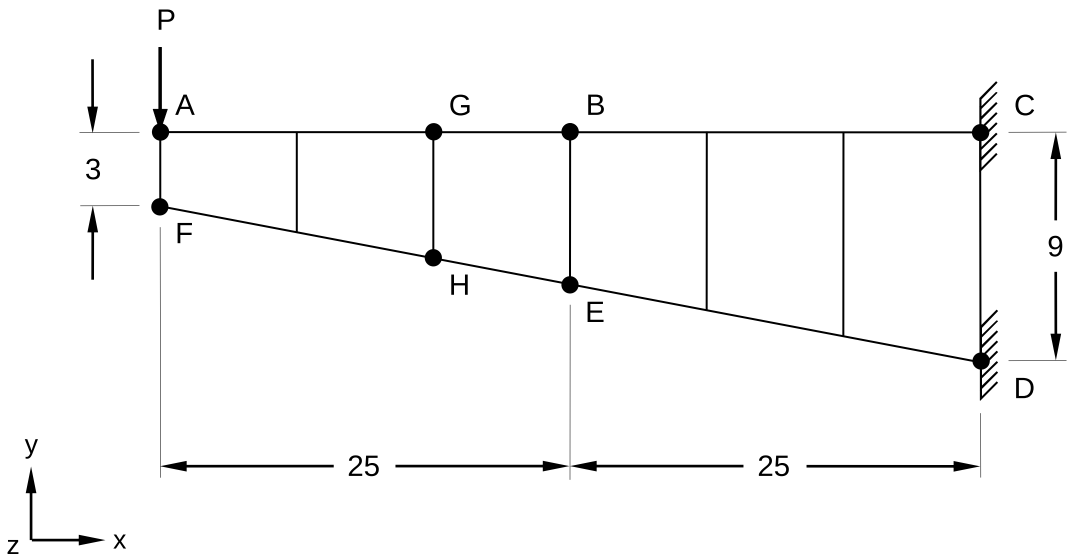


Figure 1. Tapered Support Structure Model Case 1

Autodesk Inventor Nastran Analysis Model Filename

- vm2_18.nas

Model Data

Finite Element Modeling

- 14 nodes, 6 6-DOF/node quadrilateral plate elements, and 2 beam elements along edges BG and EH (used to recover the stress at mid-span)

Units

inch/pound/second

Model Geometry

Length: L = 50 in
Width: w = 2 in
Height: h_1 = 3 in and h_2 = 9 in

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Poisson's Ratio: ν = 0.2

Boundary Conditions

The deep end (CD) is fixed in all translations and all rotations. All the other nodes are constrained in the Z-translation and X and Y-rotations. A load P = 4,000 lb is applied at the free end of the cantilever in the negative Y-direction.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Maximum Bending Stress at Mid-Span (psi)	8,333	8,234	1.2

References

1. Crandall, S. H. and Dahl, N. C., *An Introduction to the Mechanics of Solids*, 3rd Edition. New York: McGraw-Hill, Inc., 1959.

2.19 Laterally Loaded Tapered Support Structure Case 2

Problem Description

Figure 1 shows the beam model. Static analysis is performed on the beam. The maximum bending stress at mid span (point B) is determined. All dimensions are in inches.

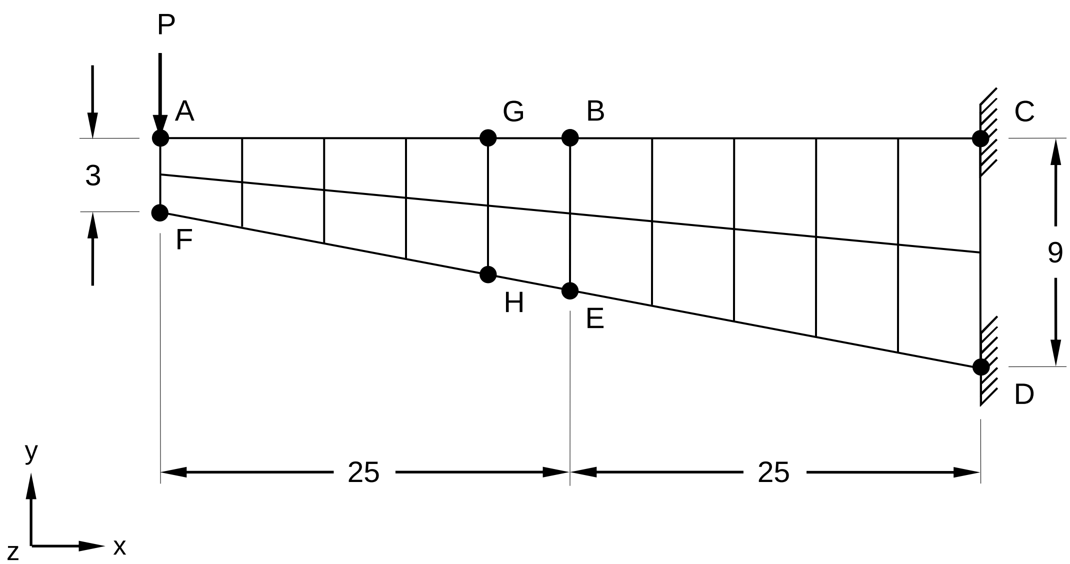


Figure 1. Tapered Support Structure Model Case 2

Autodesk Inventor Nastran Analysis Model Filename

- vm2_19.nas

Model Data

Finite Element Modeling

- 33 nodes, 20 6-DOF/node quadrilateral plate elements, and 2 beam elements along edges BG and EH (used to recover the stress at mid-span)

Units

inch/pound/second

Model Geometry

Length: L = 50 in
Width: w = 2 in
Height: h_1 = 3 in and h_2 = 9 in

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Poisson's Ratio: ν = 0.2

Boundary Conditions

The deep end (CD) is fixed in all translations and all rotations. All the other nodes are constrained in the Z-translation and X and Y-rotations. A load P = 4,000 lb is applied at the free end of the cantilever in the negative Y-direction.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Maximum Bending Stress at Mid-Span (psi)	8,333	8,330	0.0

References

1. Crandall, S. H. and Dahl, N. C., *An Introduction to the Mechanics of Solids*, 3rd Edition. New York: McGraw-Hill, Inc., 1959.

2.20 Bending of a Tee-Shaped Beam

Problem Description

Figure 1 shows the short tee-shaped beam model and a cross-section of the beam. Static analysis is performed on the beam. The maximum tensile and compressive stresses are determined. All dimensions are in inches.

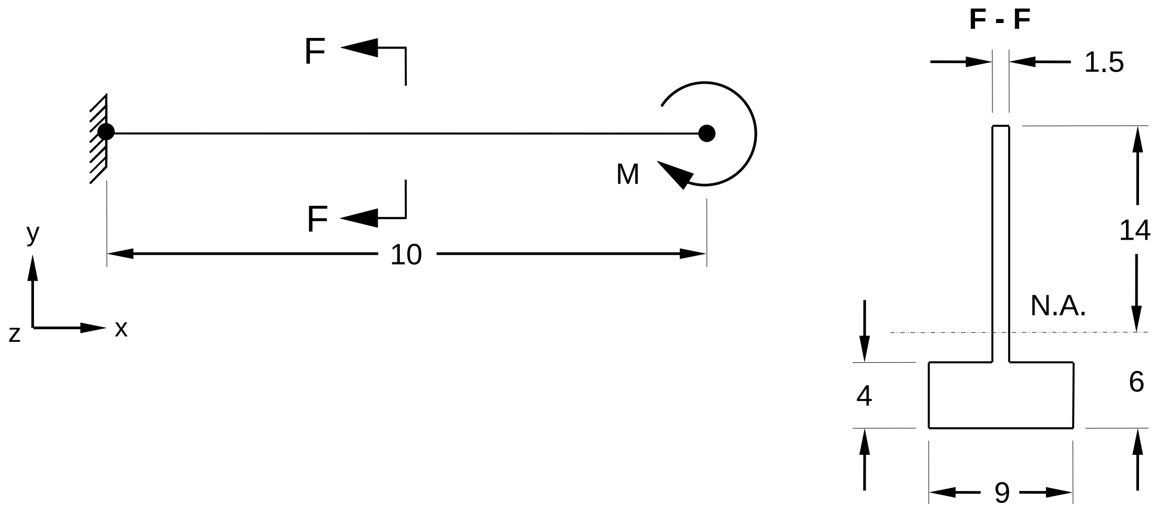


Figure 1. Tee-Shaped Beam Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_20.nas

Model Data

Finite Element Modeling

- 2 nodes, 1 beam element

Units

inch/pound/second

Model Geometry

Length: L = 10 in
Width: w = 9 in
Height to Neutral Axis: h_1 = 6 in
Height: h_2 = 20 in
Thickness Flange: t_f = 4 in
Thickness Web: t_w = 1.5 in

Cross Sectional Properties

Area: A = 60 in^2
Moment of Inertia: I_z = 2000 in^4

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Shear Modulus of Elasticity: G = 15.0 E+6 psi

Boundary Conditions

The left end is fixed in all translations and all rotations. The right end is constrained in the Z-translation and X and Y-rotations. A moment M = 100,000 lb-in is applied at the free end of the cantilever in the positive Z-direction.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Maximum Tensile Stress (psi)	300	300	0.0
Maximum Compressive Stress (psi)	-700	-700	0.0

References

1. Crandall, S. H. and Dahl, N. C., *An Introduction to the Mechanics of Solids*, 3rd Edition. New York: McGraw-Hill, Inc., 1959

.

2.21 Bending of a Circular Plate

Problem Description

Figure 1 shows the plate model with pressure loading. Static analysis is performed on the plate. The maximum deflection in the middle of the plate and the maximum stress in the plate are determined. All dimensions are in inches.

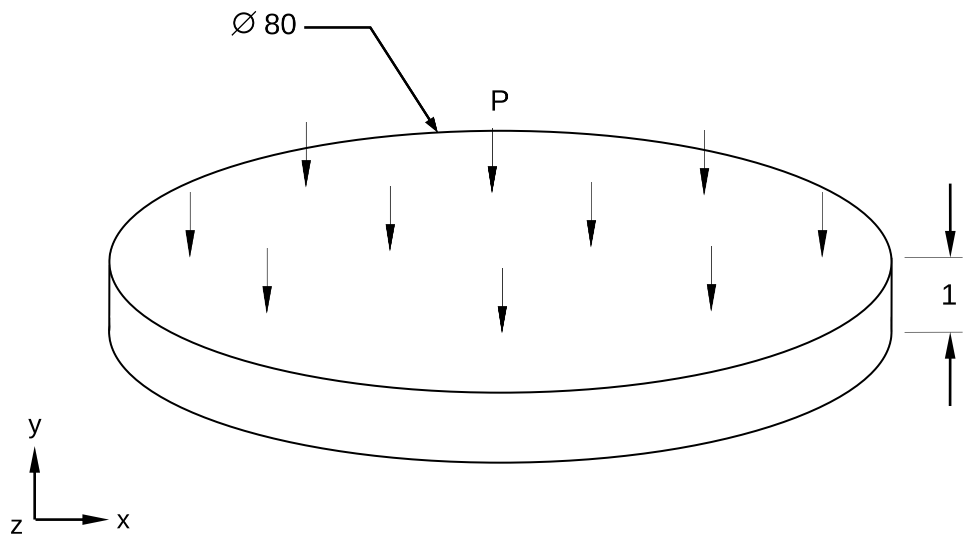


Figure 1. Circular Plate Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_21.nas

Model Data

Finite Element Modeling

- 641 nodes, 640 5-DOF/node quadrilateral plate elements

Units

inch/pound/second

Model Geometry

Radius: R = 40 in
Thickness: t = 1 in

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Poisson's Ratio: ν = 0.3

Boundary Conditions

The outside edge is fixed in all translations and all rotations. A uniform pressure P = 6 psi is applied to the entire surface of the plate.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Maximum Displacement (in)	-8.736E-2	-8.603E-2	1.5
Maximum Stress (psi)	7200	7164	0.5

References

1. Timoshenko, S., *Strength of Materials, Part 2, Elementary Theory and Problems*, 3rd Edition. New York: D. VanNostrand Co., Inc., 1956.

2.22 Beam on Elastic Foundation

Problem Description

Figure 1 shows the beam model. Static analysis is performed on the beam. The maximum deflection and bending stress at the center of the beam are determined. All dimensions are in inches.

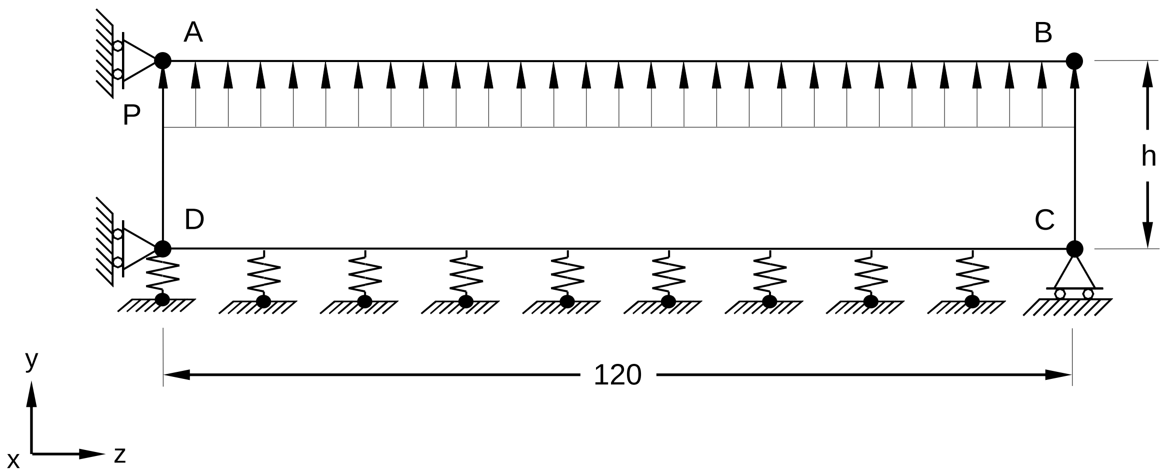


Figure 1. Beam Model on Elastic Foundation

Autodesk Inventor Nastran Analysis Model Filename

- vm2_22.nas

Model Data

Finite Element Modeling

- A ½ model is created with 62 nodes, 20 5-DOF/node quadrilateral plate elements, 20 rod elements for the springs, and 20 beam elements placed along the top of the plates (used to recover the stress at mid-span).

Units

inch/pound/second

Model Geometry

Length: L = 240 in
Height: H = 7.113786 in
Thickness: t = 1 in

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Shear Modulus of Elasticity: G = 15.0 E+6 psi
Spring Constant: k = 156.25 lb/in (k = 78.125 lb/in at mid point due to symmetry)

Boundary Conditions

The end, node C, is constrained in the X and Y-translations, and in the Y and Z-rotations (roller). Springs are attached at each node along the bottom edge DC, except at node C. At mid symmetry, top and bottom nodes (A and D) are restrained against movement in the horizontal translation (Z-direction) in addition to one half of the spring constant in the vertical direction. A uniform load P = 43.4 lbs/in is applied to the entire top part of the beam in the positive Y-direction.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Maximum Displacement (in)	1.0453	1.0455	0.0
Maximum Bending Stress (psi)	18052	18058	0.0

References

1. Peterson, F. E., *EASE2, Elastic Analysis for Structural Engineering, Example Problem Manual*, Engineering Analysis Corporation. Berkeley, California, 1981.

2.23 Thick Walled Cylinder Plain Strain

Problem Description

Figure 1 shows the thick walled cylinder model. Static analysis is performed on the beam. The radial displacements, radial stress, tangential and longitudinal stresses at inner surface of the thick wall cylinder are determined. All dimensions are in inches.

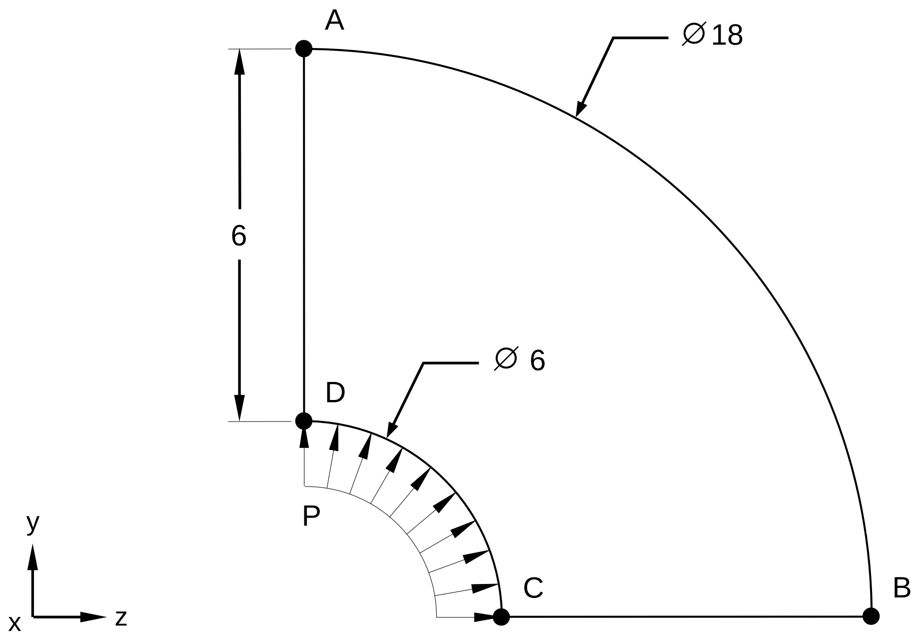


Figure 1. Thick Walled Cylinder Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_23.nas

Model Data

Finite Element Modeling

- A ¼ symmetry model is created using joint restraints with 589 nodes, 540 6-DOF/node quadrilateral plate elements, and 18 beam elements placed along the inner circle (used for load distribution).

Units

inch/pound/second

Model Geometry

Inner Radius: R_i = 3 in
Outer Radius: R_o = 9 in
Thickness: t = 1 in

Material Properties

Young's Modulus: E = 1.0 E+6 psi
Poisson's Ratio: ν = 0.3

Boundary Conditions

All nodes are constrained against movement in the direction perpendicular to the surface (translation in the Z-direction and rotations in the X and Y-directions). Symmetry boundary conditions are applied to the AD and BC edges. A unit pressure load P = 1 psi is applied on the inner radius on the beams.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Radial Displacement x1E-5 (in)	0.4582	0.4649	1.5
Radial Stress (psi)	-1.0	-0.98	2.0
Tangential Stress (psi)	1.25	1.25	0.0

References

1. McNeal, R. H. and Harder, R. C., *A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Element in Analysis and Design 1*. North Holland, 1985.

2.24 Scordelis-Lo Roof

Problem Description

Figure 1 shows the roof model. Static analysis is performed on the roof. The vertical displacement at the center of the free edge, top and bottom stresses at the centerline section at vertical angle, and top and bottom stresses at the centerline section at the free edge are determined. All dimensions are in feet.

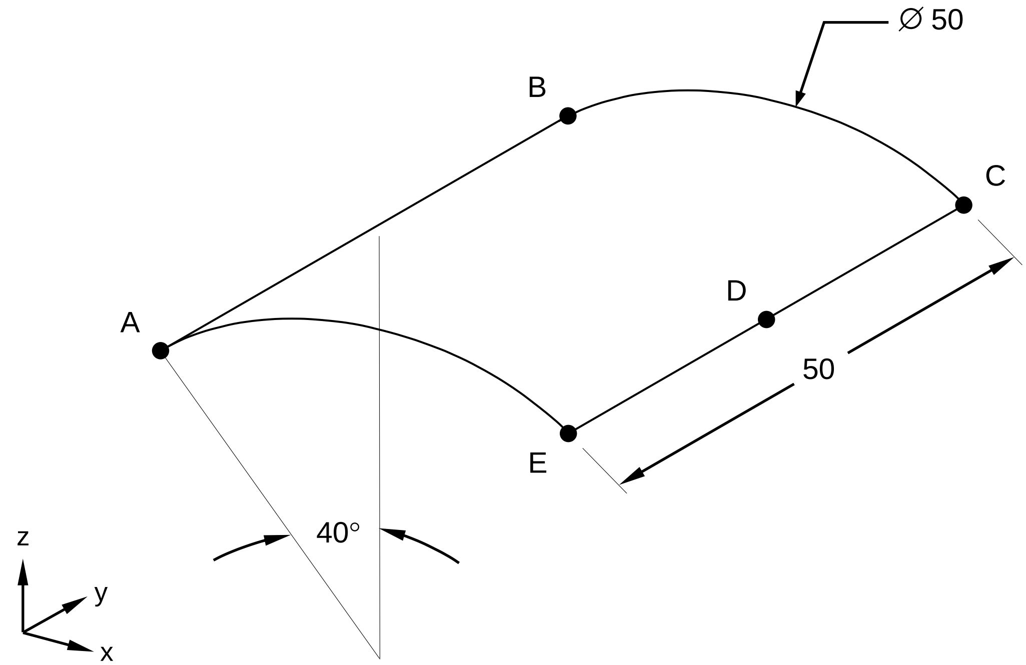


Figure 1. Scordelis-Lo Roof Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_24.nas

Model Data

Finite Element Modeling

- A ½ symmetry model is created using 1387 nodes and 1296 6-DOF/node quadrilateral plate elements.

Units

foot/pound/second

Model Geometry

Length: L = 50 feet
Radius: R = 25 feet
Thickness: t = 0.25 feet

Material Properties

Young's Modulus: E = 4.32 E+8 psf
Poisson's Ratio: ν = 0.17

Boundary Conditions

All nodes on the curved edges are simply supported (constrained translation in the X and Z-directions and rotation in the Y-direction). Symmetry boundary conditions are applied to the AB edge. The node D at the centerline section on the free edge (CE) is constrained in the Y-translation to stabilize the model. A gravity load P = 90 psf is applied uniformly on the surface.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Displacement at Center of Free Edge (ft) – Vertical	-0.3019	-0.3152	4.4
Displacement at Center of Free Edge (ft) – Horizontal	-0.1593	-0.1673	5.0
Stress at Centerline Section at Vertical Angle x 1E+3 (ksf) – Top	191.23	195.53	2.2
Stress at Centerline Section at Vertical Angle x 1E+3 (ksf) – Bottom	-218.74	-222.64	1.8
Stress at Centerline Section at Free End x 1E+3 (ksf) – Top	215.57	242.19	12.3
Stress at Centerline Section at Free End x 1E+3 (ksf) – Bottom	340.70	374.19	9.8

References

1. Scordelis, A. C. and Lo, K. S., *Computer Analysis of Cylindrical Shells*, Journal of the American Concrete Institute, Vol. 61, May 1964.

2.25 Out-of-Plane Bending of a Curved Bar

Problem Description

Figure 1 shows the curved bar model (90 degrees arc). Static analysis is performed on the bar. The maximum displacement at the free end, and maximum bending stress at support are determined. All dimensions are in inches.

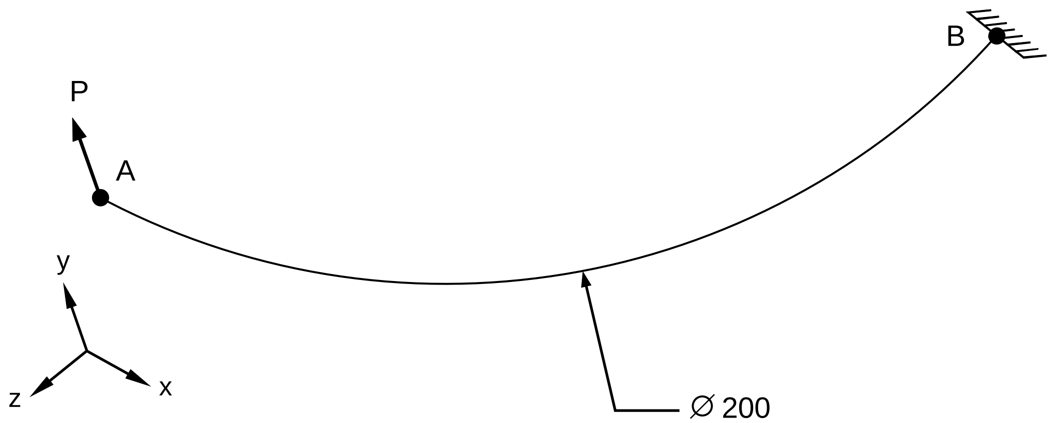


Figure 1. Curved Bar Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_25.nas

Model Data

Finite Element Modeling

- 19 nodes, 18 beam elements

Units

inch/pound/second

Model Geometry

Arc: 90 degrees
Arc Radius: R = 100 in
Bar Radius: R_{bar} = 1 in

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Poisson's Ratio: ν = 0.3

Boundary Conditions

One end (point B) is fixed in all translations and rotations. A load P = 50 lb is applied at the free end (point A), out of plane to the curved beam.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Displacement at Free End (in)	-2.648	-2.647	0.0
Maximum Bending Stress at Support (psi)	6366	6638	4.3

References

1. Timoshenko, S., *Strength of Materials, Part 1, Elementary Theory and Problems*, 3rd Edition. New York: D. VanNostrand Co., Inc., 1955.

2.26 Deflection of Hinged Support

Problem Description

Figure 1 shows the hinged beam model. Static analysis is performed on the beams. The maximum displacement at point C (node 2) and stress in each beam are determined. All dimensions are in inches.

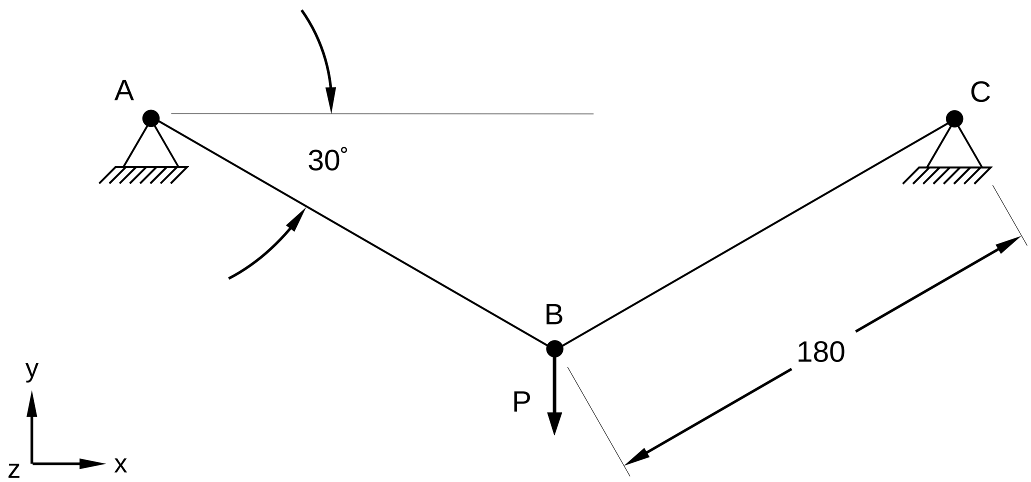


Figure 1. Hinged Support Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_26.nas

Model Data

Finite Element Modeling

- 3 nodes, 2 beam elements

Units

inch/pound/second

Model Geometry

Length: L = 180 in
Angle: α = 30 degrees
Beam Radius: R_{beam} = 1 in

Cross Sectional Properties

Area: A = 0.50 in^2

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Poisson's Ratio: ν = 0.3

Boundary Conditions

All translations and X and Y-rotations are restrained for points A and B. A point load P = 5000 lb is applied at point C (node 2) in the negative Y-direction.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Displacement at Node 2 (in)	-0.12	-0.12	0.0
Maximum Stress (psi)	10,000	10,000	0.0

References

1. Timoshenko, S., *Strength of Materials, Part 1, Elementary Theory and Problems*, 3rd Edition. New York: D. VanNostrand Co., Inc., 1955.

2.27 Thermal Stresses in a Plate

Problem Description

Figure 1 shows the plate model. Static analysis is performed on the plate. The variation in temperature from the inside to the outside follows a linear law. The bending moment per unit length of the clamped edge to prevent the plate from bending and the maximum bending stress are determined. All dimensions are in inches.

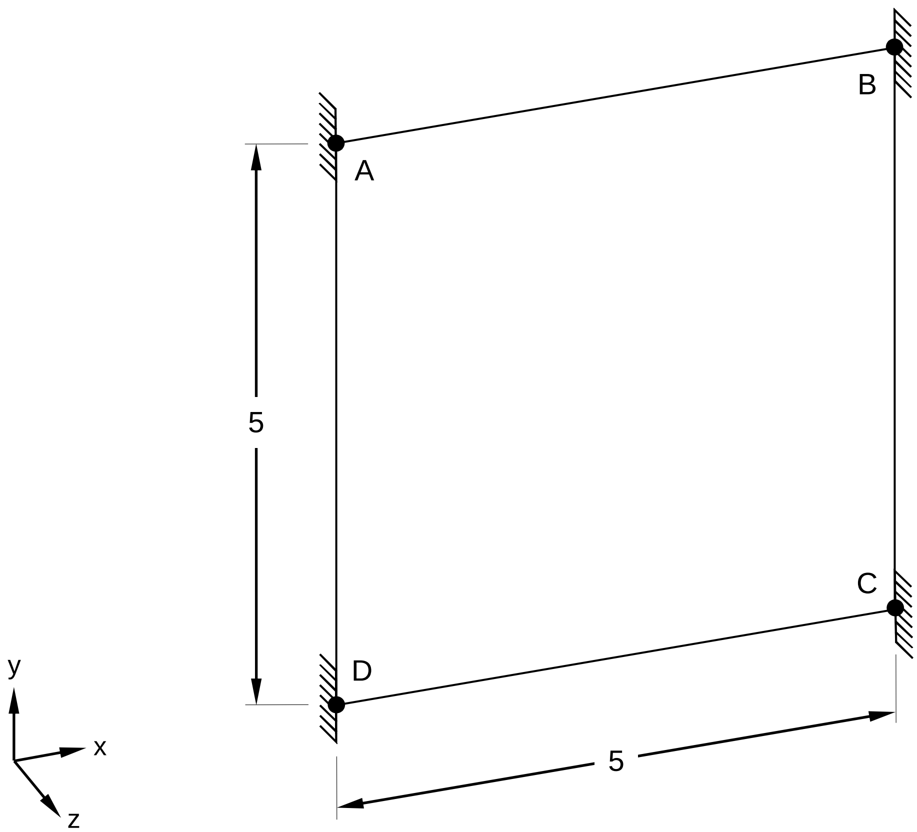


Figure 1. Plate Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_27.nas

Model Data

Finite Element Modeling

- 4 nodes, 1 5-DOF/node quadrilateral element

Units

inch/pound/second

Model Geometry

Length: L = 5 in
Thickness: t = 0.5 in

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Poisson's Ratio: ν = 0.3
Thermal Expansion Coefficient: α = 7.0 E-6 in/in- °F
Temperature one Face: T_o = 0 °F
Temperature the other Face: T_1 = 100 °F

Boundary Conditions

All translations and rotations are restrained for points A, B, C and D (clamped edges). A temperature load from one face to the other of 100° F is applied to the plate. The temperature varies linearly from one face to another.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Bending Moment (in-lb/in)	-625	-625	0.0
Maximum Stress (psi)	-15,000	-15,000	0.0

References

1. Timoshenko, S., *Strength of Materials, Part 2, Elementary Theory and Problems*, 3rd Edition. New York: D. VanNostrand Co., Inc., 1956.

2.28 Bending of a Tapered Plate

Problem Description

Figure 1 shows the tapered plate model. Static analysis is performed on the plate. The maximum deflection and maximum principal stress in the plate are determined. All dimensions are in inches.

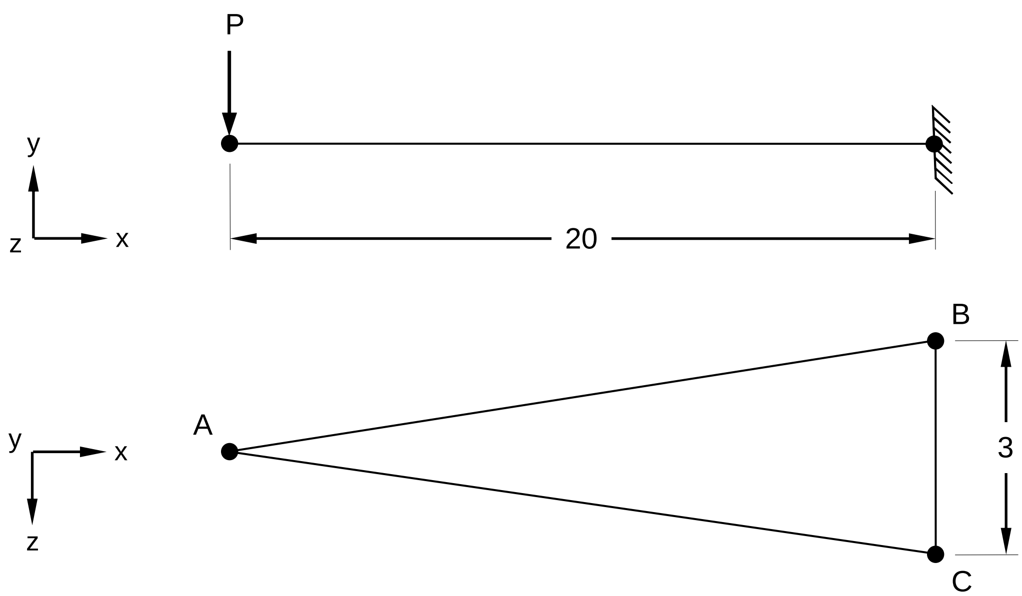


Figure 1. Tapered Plate Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_28.nas

Model Data

Finite Element Modeling

- 41 nodes, 38 5-DOF/node triangle elements

Units

inch/pound/second

Model Geometry

Length: L = 20 in
Width: w = 3 in
Thickness: t = 0.5 in

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Poisson's Ratio: ν = 0.3

Boundary Conditions

All translations and rotations are restrained for points B and C. A vertical point load P = 10 lb is applied to the plate model at point A (see Figure 1).

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Displacement (in)	-4.2667E-2	-4.2556E-2	0.3
Maximum Principal Stress (psi)	1,600	1,600	0.0

References

1. Harris, C. O., *Introduction to Stress Analysis.* New York: The Macmillan Co., 1959.

2.29 Bending of a Tapered Beam

Problem Description

Figure 1 shows the tapered beam model. Static analysis is performed on the beam. The maximum deflection and maximum principal stress in the beam are determined. All dimensions are in inches.

This problem is used to compare the results to that of a finite element plate model (see problem 2.28). The results compare favorably.

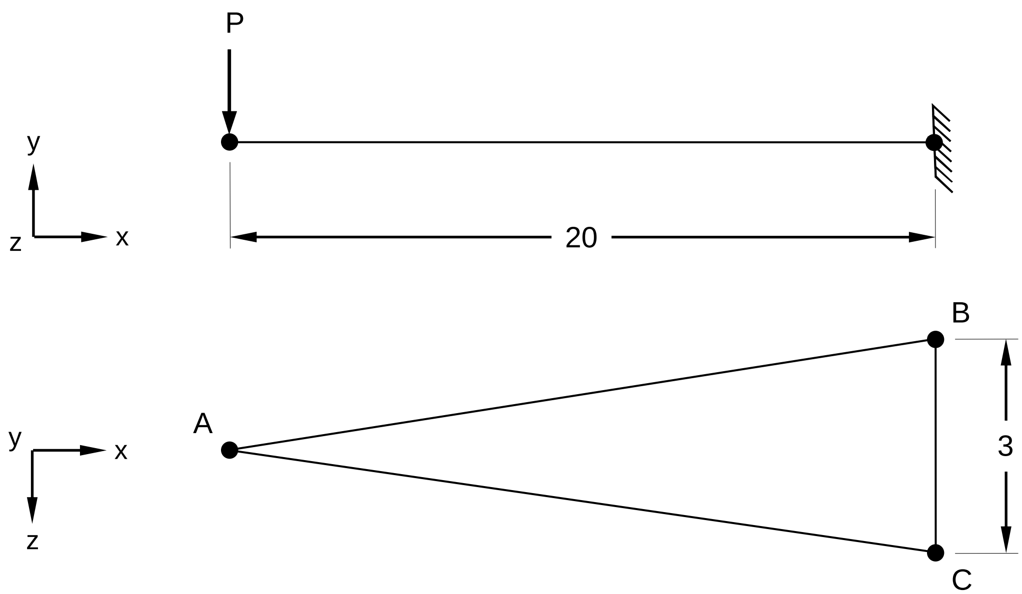


Figure 1. Tapered Beam Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_29.nas

Model Data

Finite Element Modeling

- 11 nodes, 10 beam elements

Units

inch/pound/second

Model Geometry

Length: L = 20 in
Width: w = 3 in
Thickness: t = 0.5 in

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Poisson's Ratio: ν = 0.3

Boundary Conditions

All translations and rotations are restrained for points B and C. A vertical point load P = 10 lb is applied to the beam model at point A (see Figure 1).

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Displacement (in)	-4.2667E-2	-4.2667E-2	0.0
Maximum Principal Stress (psi)	1.600E+3	1.613E+3	0.8

References

1. Harris, C. O., *Introduction to Stress Analysis.* New York: The Macmillan Co., 1959.

2.30 Bending of a Curved Thick Beam

Problem Description

Figure 1 shows the curved thick beam model. Static analysis is performed on the beam. The maximum tensile stress and maximum compressive stress in the beam are determined. All dimensions are in inches.

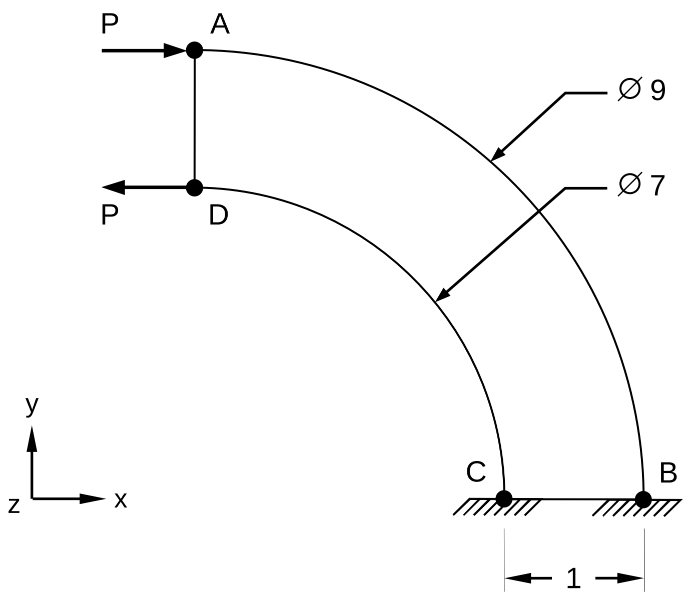


Figure 1. Curved Thick Beam Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_30.nas

Model Data

Finite Element Modeling

- A ¼ model (90 degrees arc) model is created using 105 nodes, 80 5-DOF/node quadrilateral elements and 4 beam elements on the edge AD. Symmetry boundary conditions are applied.

Units

inch/pound/second

Model Geometry

Inner Radius: R_i = 3.5 in
Outer Radius: R_o = 4.5 in
Width: w = 1 in

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Poisson's Ratio: ν = 0.3

Boundary Conditions

All translations and rotations are restrained for all nodes between points B and C. A force couple P = 100 lb is applied at the free end of the beam to simulate a 100 in-lb moment (see Figure 1).

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Tensile Stress (psi)	655.0	655.5	0.1
Compressive Stress (psi)	-555.0	-552.1	0.5

References

1. Timoshenko, S., *Strength of Materials, Part 1, Elementary Theory and Problems*, 3rd Edition. New York: D. VanNostrand Co., Inc., 1955. p.98.

2.31 Truss Reaction One

Problem Description

Figure 1 shows the truss model. Static analysis is performed on the truss. The vertical reaction at the supports of the truss are determined. All dimensions are in inches.

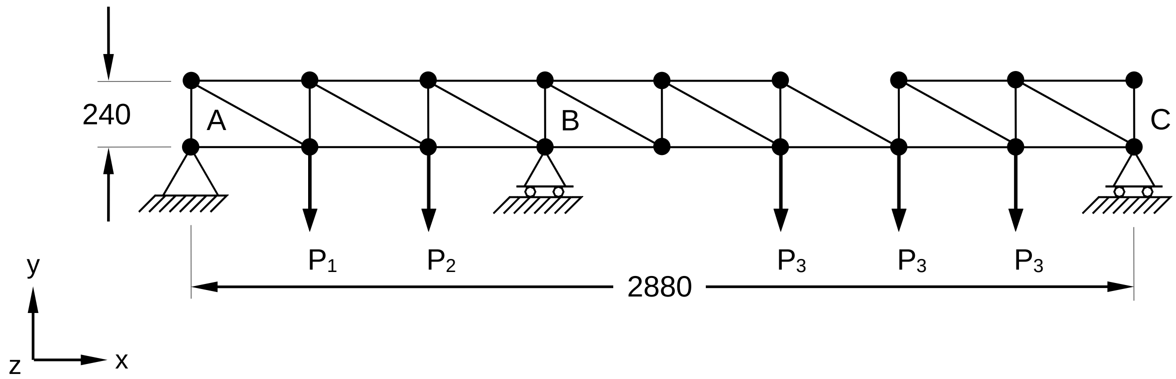


Figure 1. Truss Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_31.nas

Model Data

Finite Element Modeling

- 18 nodes, 32 beam elements

Units

inch/Kip/second

Model Geometry

Length: L = 2880 in (8x360 in)
Height: h = 240 in

Cross Sectional Properties

Area: A = 100 in^2 (for input only)

Material Properties

Young's Modulus: E = 30.0 E+3 ksi
Poisson's Ratio: ν = 0.3

Boundary Conditions

All translations are restrained at point A (pinned support). At points B and C, translations in the Y and Z-directions, and rotations in the X and Y-directions are restrained (rollers). All nodes are constrained in the Z-translation, X and Y-rotations. Several point loads are applied as shown in Figure 1: P_1 = 40 Kip, P_2 = 80 Kip, and P_3 = 60 Kip.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Reaction at Point A (Kip)	-76.7	-76.7	0.0
Reaction at Point B (Kip)	346.7	346.7	0.0
Reaction at Point C (Kip)	30.0	30.0	0.0

References

1. McCormac, J. C., *Structural Analysis*, 3rd Edition. New York: In Text Educational Publishers, 1975.

2.32 Truss Reaction Two

Problem Description

Figure 1 shows the truss model. Static analysis is performed on the truss. The vertical reaction at the supports of the truss are determined. All dimensions are in inches.

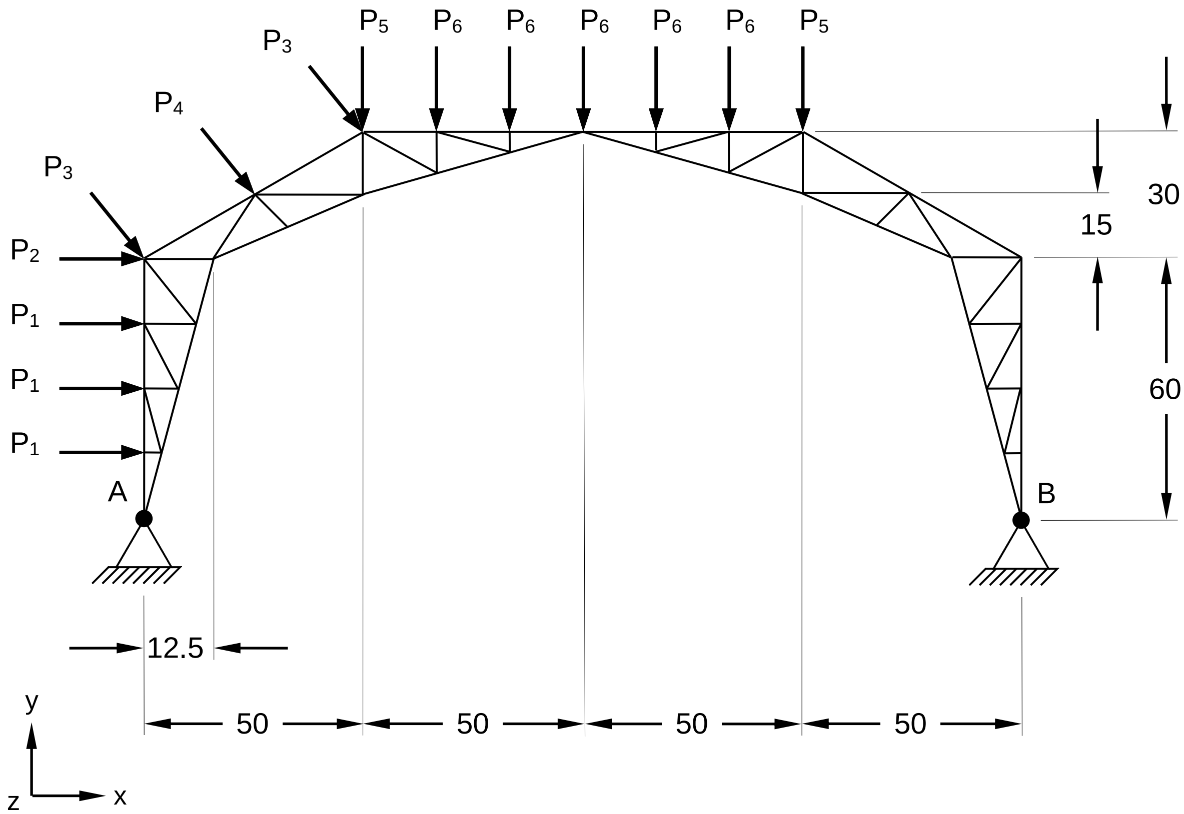


Figure 1. Truss Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_32.nas

Model Data

Finite Element Modeling

- 35 nodes, 66 beam elements

Units

inch/Kip/second

Model Geometry

Length: $L = 200$ in
Height: $h = 90$ in

Cross Sectional Properties

Area: $A = 100\ in^2$ (for input only)

Material Properties

Young's Modulus: $E = 30.0$ E+3 ksi
Poisson's Ratio: $\nu = 0.3$

Boundary Conditions

All translations are restrained at points A and B (pinned support). All nodes are constrained in the Z-translation, X and Y-rotations. Several point loads are applied as shown in Figure 1: $P_1 = 4$ Kip, $P_2 = 2$ Kip, $P_3 = 3$ Kip, $P_4 = 6$ Kip, $P_5 = 5$ Kip, and $P_6 = 10$ Kip.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Vertical Reaction at Point A (Kip)	34.3	34.3	0.0
Horizontal Reaction at Point A (Kip)	11.5	11.5	0.0
Vertical Reaction at Point B (Kip)	36.0	36.0	0.0
Horizontal Reaction at Point B (Kip)	-31.7	-31.7	0.0

References

1. McCormac, J. C., *Structural Analysis*, 3rd Edition. New York: In Text Educational Publishers, 1975.

2.33 Fixed Ended Beam Un-symmetric Tapered Member

Problem Description

Figure 1 shows the beam model. Static analysis is performed on the beam. The moments at each of the supports are determined. All dimensions are in inches.

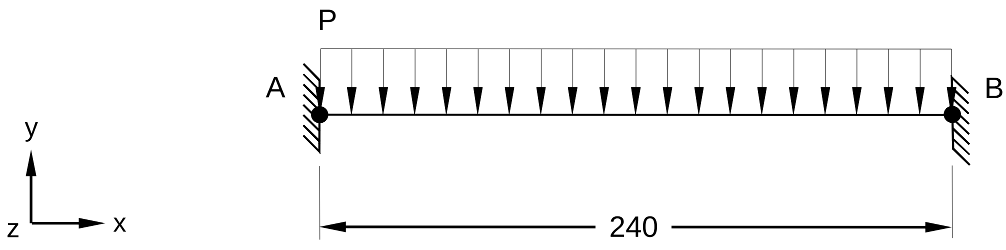


Figure 1. Beam Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_33.nas

Model Data

Finite Element Modeling

- 3 nodes, 2 beam elements

Units

inch/Kip/second

Model Geometry

Length: L = 240 in
Height: h_1 = 10 in (constant section) and h_2 = 20 in (the deep end of the variable section)
Width: w = 1 in

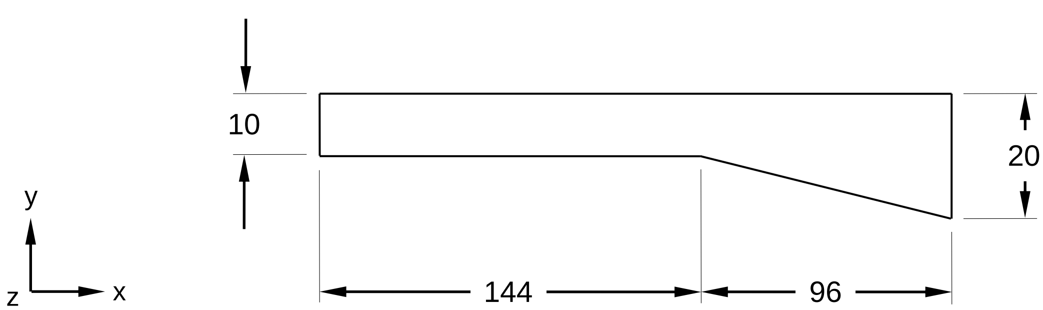


Figure 2. Un-symmetric Tapered Beam

Material Properties

Young's Modulus: E = 30.0 E+3 ksi
Poisson's Ratio: ν = 0.3

Boundary Conditions

All translations and rotations are restrained at points A and B (see Figure 1). A uniform load P = 4 Kip/ft (or 0.3333 Kip/inch) is applied to the entire beam.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Moment at Point A (Kip)	-98.2	-95.9	2.3
Moment at Point B (Kip)	-217.2	-223.5	2.9

References

1. McCormac, J. C., *Structural Analysis*, 3rd Edition. New York: In Text Educational Publishers, 1975.

2.34 Straight Cantilever Beam Using Solids

Problem Description

Figure 1 shows the straight beam model. Static analysis is performed on the cantilever beam. The displacements in the direction of the loads are determined. All dimensions are in inches.

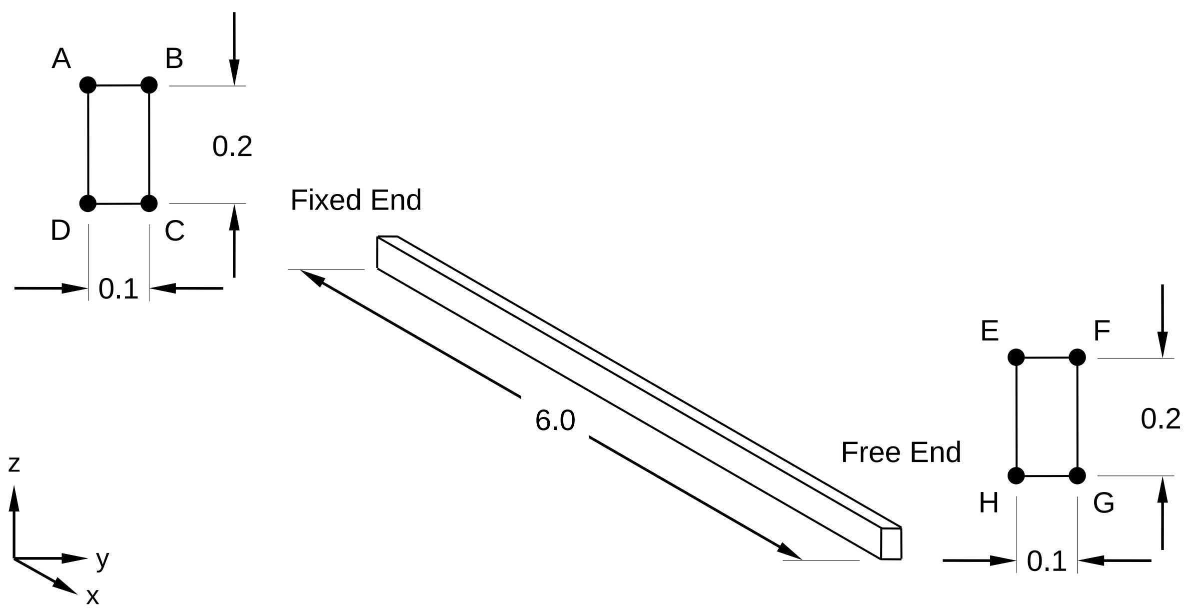


Figure 1. Beam Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_34.nas

Model Data

Finite Element Modeling

- 28 nodes, 6 solid elements

Units

inch/Kip/second

Model Geometry

Length: L = 6.0 in
Height: h = 0.2 in
Width: w = 0.1 in

Material Properties

Young's Modulus: E = 1.0 E+7 ksi
Poisson's Ratio: ν = 0.3

Boundary Conditions

At points A and B the nodes are restrained in the X-translation, while at points C and D the nodes are restrained in all translations (pinned). A unit axial load P in the X-direction, a unit load at tip along Z-direction, a unit load at tip along Y-direction, and a unit couple at tip about the X-axis via force couple P = 5 Kip along edge HG and edge EF, respectively, are applied to the beam.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Translation in X-direction – Axial (in)	3.0E-5	3.0E-5	0.0
Translation in Z-direction – In-Plane Shear (in)	1.081E-1	1.072E-1	0.8
Translation in Y-direction – Out-of-Plane Shear (in)	4.321E-1	4.233E-1	2.0
Translation in Y-direction – Twist (in)	3.21E-3	2.86E-3	10.9

References

1. McNeal, R. H. and Harder, R. C., *A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Element in Analysis and Design 1*. North Holland, 1985.

2.35 Curved Beam Using Solids

Problem Description

Figure 1 shows the curved beam model. Static analysis is performed on the beam. The displacement at the free end in the direction of the loads are determined. All dimensions are in inches.

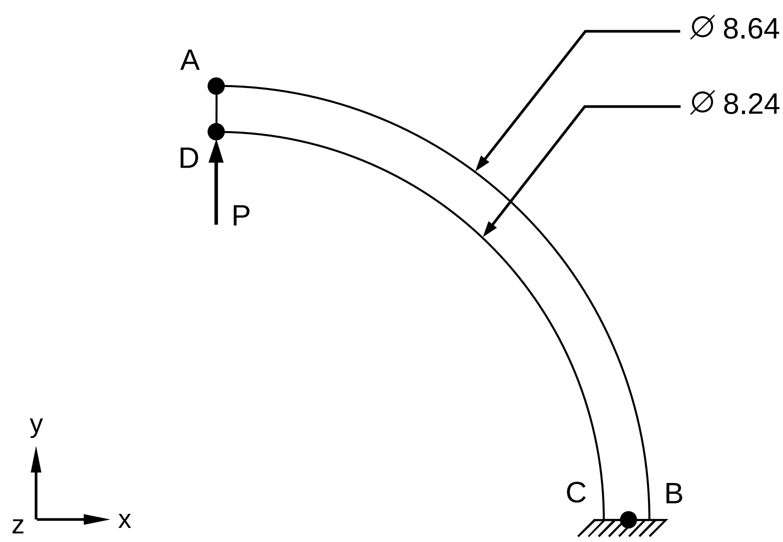


Figure 1. Beam Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_35.nas

Model Data

Finite Element Modeling

- 84 nodes, 20 solid elements

Units

inch/Kip/second

Model Geometry

Inner Radius: R_i = 4.12 in
Outer Radius: R_o = 4.32 in
Width: w = 0.1 in
Height: h = 0.2 in

Material Properties

Young's Modulus: E = 1.0 E+7 ksi
Poisson's Ratio: ν = 0.25

Boundary Conditions

At clamped end (edge BC) the nodes are retrained in all translations (pinned). This will produce the restraint couples to simulate a clamped condition. A unit load P in the Y-direction is applied at the tip (0.25 Kip/node). Note that the total force is 1 Kip at the tip.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Translation in Y-direction (in)	8.854E-2	8.816E-2	3.8

References

1. Young, W. C., *Roark's Formulas for Stress and Strain*, 6^{th} Edition. New York: McGraw-Hill Co., 1989.

2.36 Cantilever Modeled with Variable Thickness Shells and Membranes

Problem Description

Figure 1 shows the tapered plate model. The thickness varies linearly from 3 inches at the fixed edge to 1 inch at the free end. Static analysis is performed on the plate. The tip displacement in the Z-direction and the tip rotation in the X-rotation due to the uniform moment, together with the tip displacement in the Y-direction due to the tension force are determined. All dimensions are in inches.

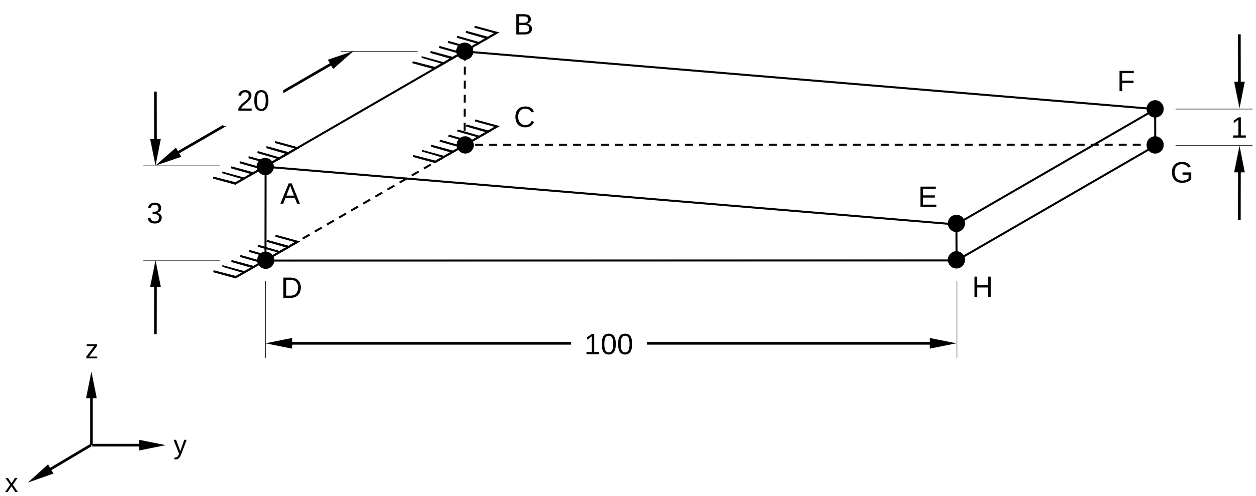


Figure 1. Tapered Plate Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_36.nas

Model Data

Finite Element Modeling

- 12 nodes, 5 5-DOF/node quadrilateral elements with varying thickness

Units

inch/Kip/second

Model Geometry

Length: L = 100.0 in
Width: w = 20.0 in
Height: h = 3.0 in at fixed end, and 1.0 in at the free end

Material Properties

Young's Modulus: E = 1.0 E+3 ksi
Poisson's Ratio: ν = 0.3

Boundary Conditions

At fixed end the nodes are restrained in all translations and rotations. A uniform moment M = 3 in-Kip/in about the X-direction at the free end is applied as 2 moments of 30 in-Kip at each node. A tensile force in the Y-direction P = 500 Kips per node (uniform edge load of 50 Kips/in) is applied at the free end as a second load case.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Translation in Z-direction due to Uniform Moment (in)	20.0	20.1	0.5
Rotation in X-direction due to Uniform Moment (rad)	0.80	0.76	5.0
Translation in Y-direction due to Tension Force (in)	2.7465	2.7225	0.9

References

1. Young, W. C., *Roark's Formulas for Stress and Strain*, 6th Edition. New York: McGraw-Hill Co., 1989.

2.37 Cantilever Modeled with Variable Thickness Solids

Problem Description

Figure 1 shows the solid model. This is the same problem as problem 2.36, using solids rather than shells. The thickness varies linearly from 3 inches at the fixed edge to 1 inch at the free end. Static analysis is performed on the solid. The tip displacement in the Z-direction due to the uniform moment is determined. All dimensions are in inches.

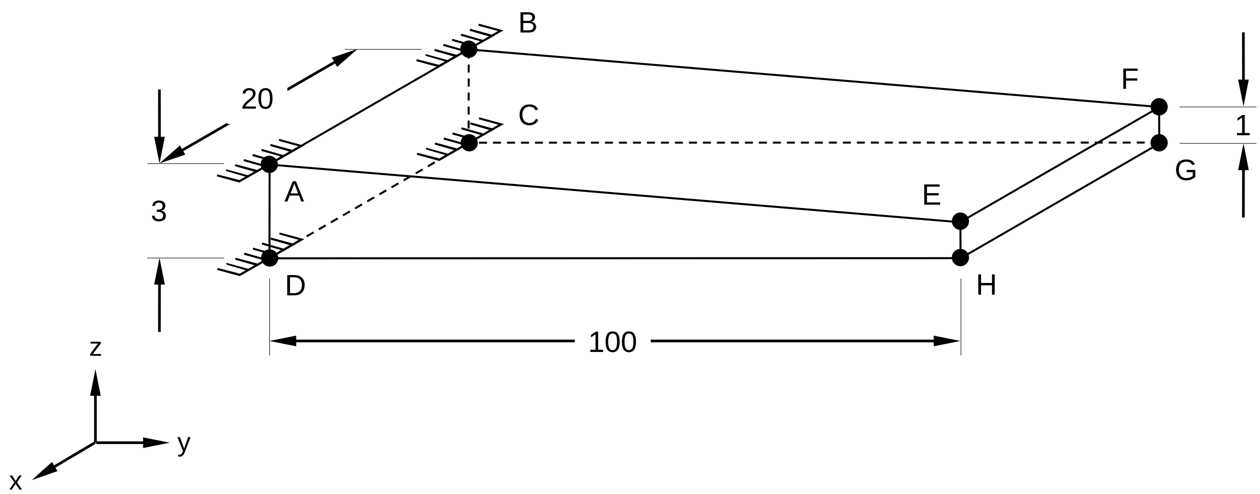


Figure 1. Tapered Solid Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_37.nas

Model Data

Finite Element Modeling

- 210 nodes, 80 solid elements with varying thickness

Units

inch/Kip/second

Model Geometry

Length: L = 100.0 in
Width: w = 20.0 in
Height: h = 3.0 in at fixed end, and 1.0 in at the free end

Material Properties

Young's Modulus: E = 1.0 E+3 ksi
Poisson's Ratio: ν = 0.3

Boundary Conditions

At fixed end the nodes are restrained in all translations and rotations. A uniform moment M = 3 in-Kip/in about the X-direction at the free end is applied as moments of 7.5 and 15 in-Kip simulated by a force couple at each external end nodes and internal edge nodes, respectively.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Translation in Z-direction due to Uniform Moment (in)	20.0	19.7	1.5

References

1. Young, W. C., *Roark's Formulas for Stress and Strain*, 6th Edition. New York: McGraw-Hill Co., 1989.

2.38 Elongation of a Solid Bar

Problem Description

Figure 1 shows the tapered solid bar model suspended from the ceiling. The thickness varies linearly from a 2 x 2 inch square at the fixed edge to a 1 x 1 inch square at the free end. Static analysis is performed on the solid. The maximum tip displacement in the Y-direction and the axial stress at mid point (E) are determined. All dimensions are in inches.

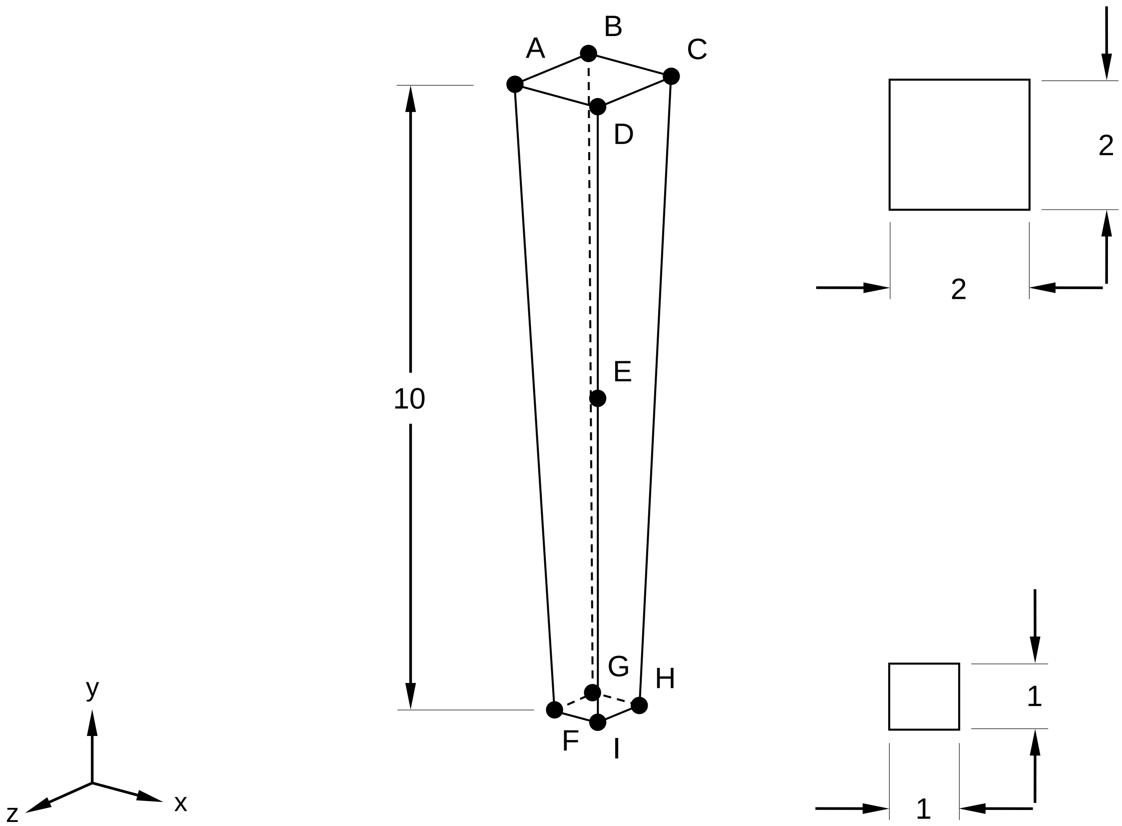


Figure 1. Tapered Solid Bar Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_38.nas

Model Data

Finite Element Modeling

- 32 nodes, 7 solid elements with varying thickness

Units

inch/pound/second

Model Geometry

Length: L = 10.0 in
Width: w = 5.0 in
Height: h = 0.1 in

Material Properties

Young's Modulus: E = 10.4 E+3 ksi
Poisson's Ratio: ν = 0.3

Boundary Conditions

At fixed end the nodes are restrained in all translations and rotations (points A, B, C and D). A uniform force P = 10,000 lbs in the negative Y-direction (2,500 lbs/node) is applied at the free end.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Translation in Y-direction (in)	-4.8077E-3	-4.757E-3	1.1
Axial Stress at Mid Point C (psi)	4,444	4,441	0.1

References

1. Harris, C. O., *Introduction to Stress Analysis.* New York: The Macmillan Co., 1959.

2.39 Thin Shell Beam in Pure Bending

Problem Description

Figure 1 shows the thin flat plate fixed to a wall. Static analysis is performed on the shell beam. The maximum major stress, maximum displacement and strain energy are determined. All dimensions are in inches.

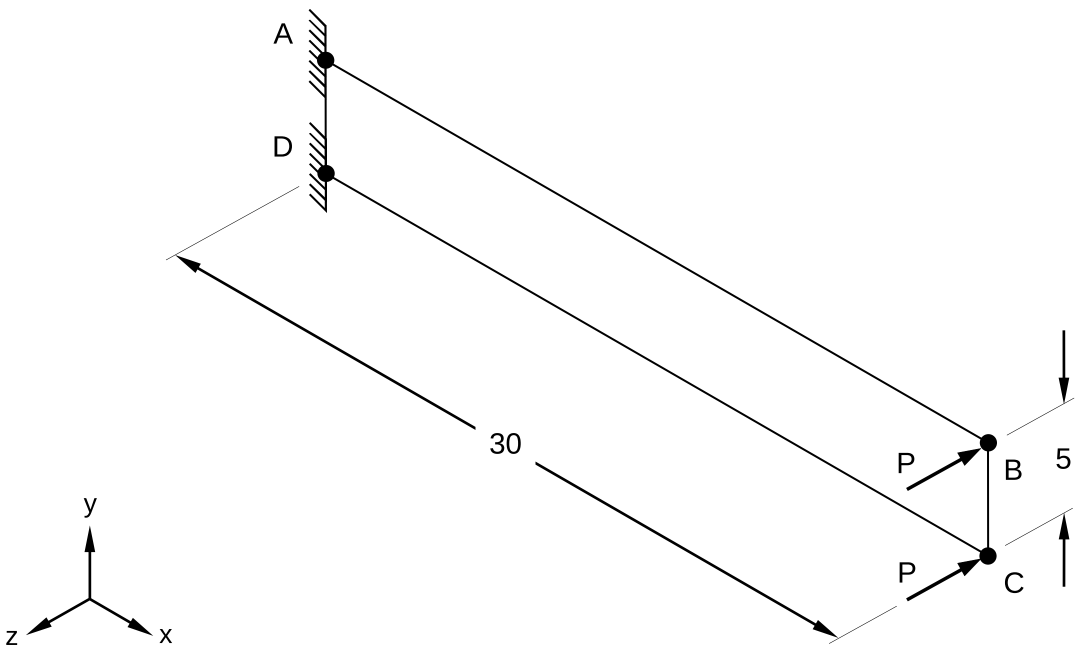


Figure 1. Shell Beam Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_39.nas

Model Data

Finite Element Modeling

- 14 nodes, 6 5-DOF/node quadrilateral plate elements

Units

inch/pound/second

Model Geometry

Length: L = 30.0 in
Width : w = 0.1 in
Height: h = 5.0 in

Material Properties

Young's Modulus: E = 30.0 E+3 ksi
Poisson's Ratio: ν = 0.3

Boundary Conditions

At fixed end (edge AD) the nodes are restrained in all translations and rotations. A nodal force P = 3 lbs at each node (edge BD) is applied in the negative Z-direction (total force is 6 lbs).

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Bottom Major Stress at Node 7 (psi)	21,600	20,535	4.9
Translation in Z-direction at Node 1(in)	4.320	4.255	1.5
Total Strain Energy (lb-in)	12.96	12.77	1.5

References

1. Shigley, J. and Mitchel, L., *Mechanical Engineering Design*, 4th Edition. New York: McGraw-Hill Co., 1983.

2.40 Static Analysis of Thermal Loading

Problem Description

Figure 1 shows the pipe line with two right angles subjected to a large temperature change. Static analysis is performed on the pipe model. The support reactions are determined (axial force, shear and moment reaction). All dimensions are in inches.

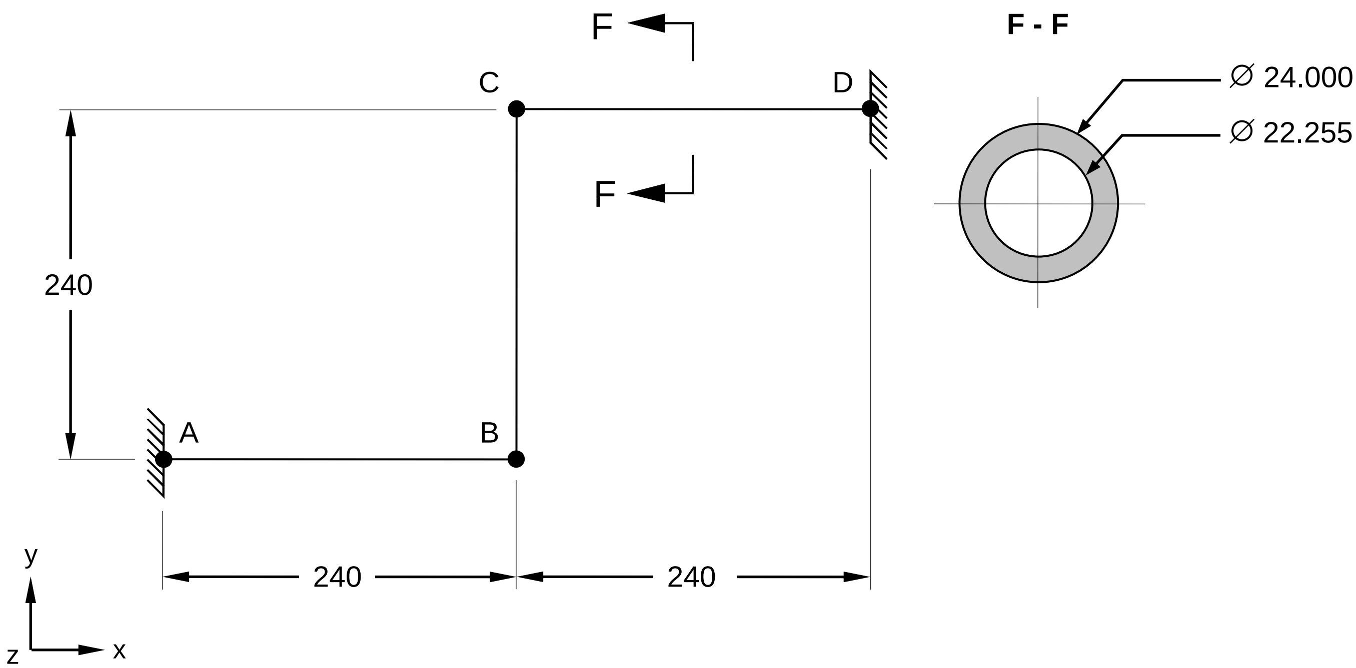


Figure 1. Pipe Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_40.nas

Model Data

Finite Element Modeling

- 4 nodes, 3 beam elements

Units

inch/pound/second

Model Geometry

Length: L = 240 in (each pipe)
Outer Radius : R_o = 12 in
Thickness: t = 0.8725 in

Cross Sectional Properties

Area: A_x = 30.5 in^2
Torsional Constant: J = 950.0 in^4
Moment of Inertia: I = 475.0 in^4
Thermal Expansion Coefficient: α = 7.26744 E-6 in/in/deg F

Material Properties

Young's Modulus: E = 26.4 E+6 psi
Poisson's Ratio: ν = 0.3

Boundary Conditions

The model is fully restrained in all translations and rotations at each end of the pipe line (nodes A and D). The other 2 intermediary nodes (B and C) are constrained in the Z-translation, X and Y-rotations. A temperature increase of 430 degrees F to each element is applied. Note that reactions at both supports should be the same.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Axial Force (lb)	8,980.47	8,949.17	0.3
Shear Reaction (lb)	7,755.86	7,729.19	0.4
Moment Reaction (in-lb)	783,750.00	781,105.00	0.3

References

1. Seely, F. B. and Smith, J. O., *Advanced Mechanics of Materials*, 2nd Edition. John Wiley and Sons, 1955.

2.41 Static Analysis of Pressure Loading

Problem Description

Figure 1 shows the square plate model. Static analysis is performed on the model with different mesh sizes. The stress at point A is determined for all models. All dimensions are in inches.

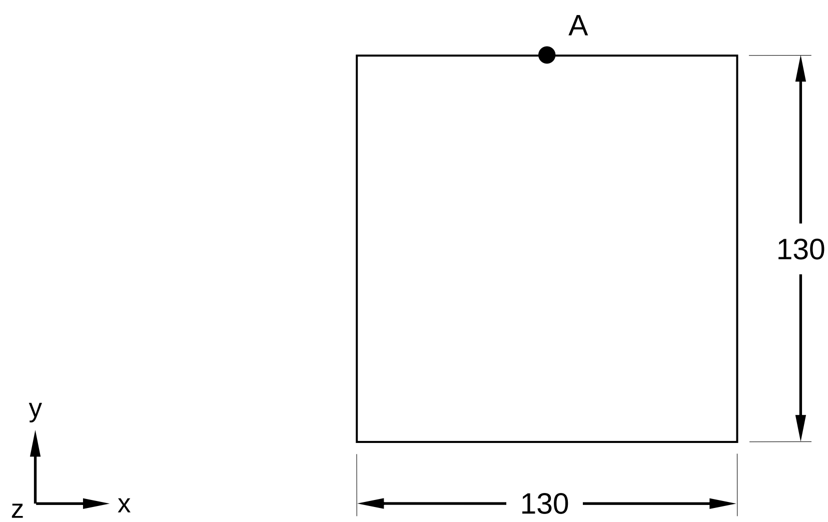


Figure 1. Plate Model

Autodesk Inventor Nastran Analysis Model Filename

- vm2_41a4.nas – 5-DOF/node, CQUAD4 elements
- vm2_41b4.nas – 5-DOF/node, CQUAD4 elements
- vm2_41c4.nas – 5-DOF/node, CQUAD4 elements

Model Data

Finite Element Modeling

- Test vm2_41a4: 16x16 mesh, 289 nodes, 256 5-DOF/node quadrilateral plate elements
- Test vm2_41b4: 32x32 mesh, 1089 nodes, 1024 5-DOF/node quadrilateral plate elements
- Test vm2_41c4: 64x64 mesh, 4225 nodes, 4096 5-DOF/node quadrilateral plate elements

Units

inch/pound/second

Model Geometry

Length: L = 130 in
Thickness: t = 1 in

Material Properties

Young's Modulus: E = 2.9 E+7 psi
Poisson's Ratio: ν = 0.3

Boundary Conditions

The model is fully restrained in all translations and rotations around the perimeter. A pressure load of 10 psi is applied on the plate.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

		Bench Value	**Autodesk Inventor Nastran**	
Mesh Size	**Element ID**	**Stress** (psi)	**Stress** (psi)	**Error** (%)
16x16	300	52,020	48,635	6.5
32x32	1196	52,020	50,417	3.1
64x64	4780	52,020	51,178	1.6

Note: The same stress values are obtained with CQUADR elements.

References

1. Hsu, T. H., *Stress and Strain Data Handbook*. Houston, London, Paris, Tokyo: Gulf Publishing Company, Book Division.

3. Linear Statics Verification Using Standard NAFEMS Benchmarks

The purpose of these linear static test cases is to verify the functionality of Autodesk Inventor Nastran using standard benchmarks published by NAFEMS (National Agency for Finite Element Methods and Standards, National Engineering Laboratory, Glasgow, U.K.).

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

3.1 Warped Element Test Cases

The following linear static verification problems using standard NAFEMS Benchmarks are performed using warped plate elements

3.1.1 Cylindrical Shell Patch

Problem Description

Figure 1 shows the model of the cylindrical shell patch. A static analysis is performed on the model. The plate top major stress at the center node is determined. All dimensions are in meters.

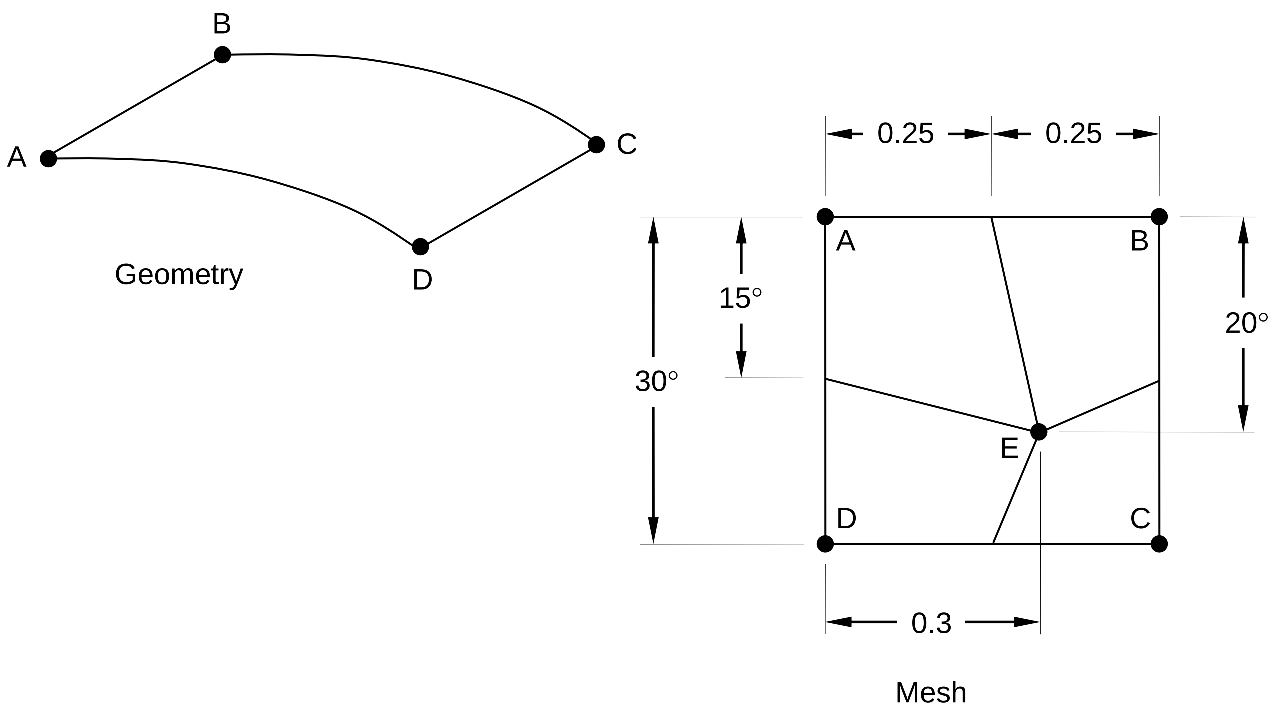


Figure 1. Cylindrical Shell Patch

Autodesk Inventor Nastran Analysis Model Filename

- vm3_1_1a4.nas – Test 1, 5-DOF/node, CQUAD4
- vm3_1_1aR.nas – Test 1, 6-DOF/node, CQUADR
- vm3_1_1b4.nas – Test 2, 5-DOF/node, CQUAD4
- vm3_1_1bR.nas – Test 2, 6-DOF/node, CQUADR

Model Data

Finite Element Modeling

- 9 nodes, 4 5-DOF/node warped quadrilateral plate elements

Units

meter/Newton/second

Model Geometry

Length: *L* = 0.5 m
Height: *h* = 0.5 m
Thickness: *t* = 0.01 m

Material Properties

Young's Modulus: *E* = 210.0 E+3 MPa
Poisons ratio: ν = 0.3

Boundary Conditions

Loads Test 1 (vm3_1_1a4 and vm3_1_1aR): Edge DC is loaded with nodal moments of -125 N-m at point D, -250 N-m at bottom center node, and -125 N-m at point C about the Z-axis.

Loads Test 2 (vm3_1_1b4 and vm3_1_1bR): Nodal forces of 75,000 N are applied at points D and C, and a force of 150,000 N is applied to the bottom edge middle node in the direction away from and tangential to the element. An elemental pressure load is applied to all elements of 600,000 Pa in the Y-direction.

Constraints Test 1 & 2: The nodes along edge AB are fully constrained in all translations and rotations. The nodes along edges AD and BC are constrained in the Z-translation and X and Y-rotations.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Test 1 results

Description	**Element**	**NAFEMS**	**Autodesk Inventor Nastran**	**Error** (%)
Plate Top Major Stress at Point E	CQUAD4	60.0	47.7	20.5
	CQUADR	60.0	50.3	16.2

Table 2. Test 2 results

Description	**Element**	**NAFEMS**	**Autodesk Inventor Nastran**	**Error** (%)
Plate Top Major Stress at Point E	CQUAD4	60.0	61.58	2.6
	CQUADR	60.0	62.4	4.0

References

1. NAFEMS Finite Element Methods & Standards, *The Standard NAFEMS Benchmarks*, Rev. 3. Glasgow: NAFEMS, 1990. Test No. LE2.

2. Davies, G. A. O., Fenner, R. T., and Lewis, R. W., *Background to Benchmarks*. Glasgow: NAFMES, 1993.

3.1.2 Hemisphere-Point Loads

Problem Description

Figure 1 shows the model of the hemisphere with an acting point load at point A. Static analysis is performed on the model. The T1 translation at point A is determined. All dimensions are in meters.

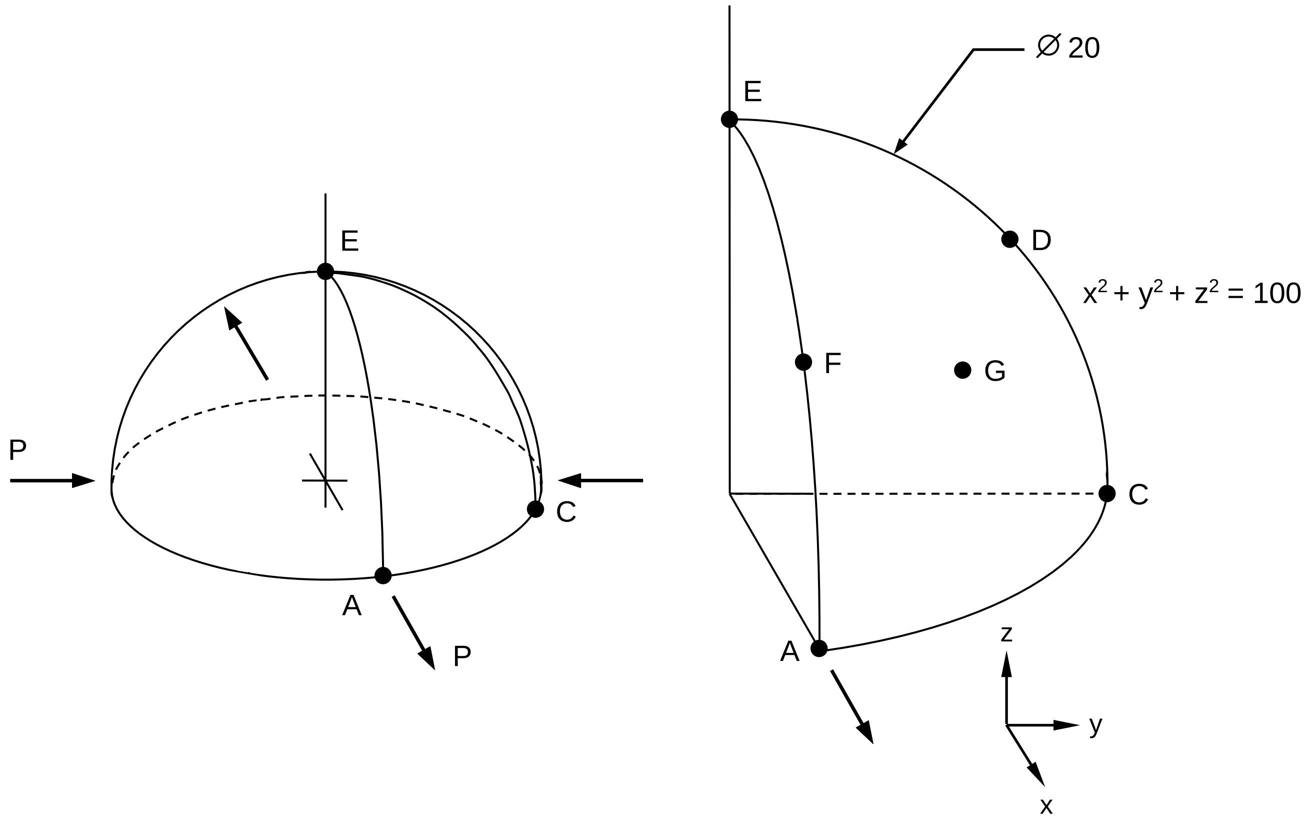


Figure 1. Hemisphere with Point Load

Autodesk Inventor Nastran Analysis

- vm3_1_2b4.nas – 5-DOF/node, CQUAD4
- vm3_1_2bR.nas – 6-DOF/node, CQUADR

Model Data

Finite Element Modeling

- 61 nodes, 48 warped quadrilateral plate elements. Point G is at $X = Y = Z = \frac{10}{\sqrt{3}}$

Units

meter/Newton/second

Model Geometry

Thickness: t = 0.04 m
Radius: R = 10 m

Material Properties

Young's Modulus: E = 68.25 E+3 MPa
Poisons Ratio: ν = 0.3

Boundary Conditions

Point E is fully constrained in all translations and rotations. Symmetry boundary conditions are applied to edges AE and CE. An outward radial load P = 2 kN is applied to point A and an inward radial load P = 2 kN is applied to point C.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Element**	**NAFEMS**	**Autodesk Inventor Nastran**	**Error** (%)
T1 Translation at Point A (m)	CQUAD4	0.185	0.179	3.2
	CQUADR	0.185	0.170	8.1

References

1. NAFEMS Finite Element Methods & Standards, *The Standard NAFEMS Benchmarks*, Rev. 3. Glasgow: NAFEMS, 1990. Test No. LE3.
2. Davies, G. A. O., Fenner, R. T., and Lewis, R. W., *Background to Benchmarks*. Glasgow: NAFMES, 1993.

3.2 Laminate Plate Element Test Cases

The following linear static verification problems using standard NAFEMS Benchmarks are performed using laminate plate elements.

3.2.1 Laminated Strip

Problem Description

Figure 1 shows the model of the laminated strip. A static analysis is performed on the model. The interlaminar shear stress at D and the Z-deflection at E are determined. All dimensions are in millimeters.

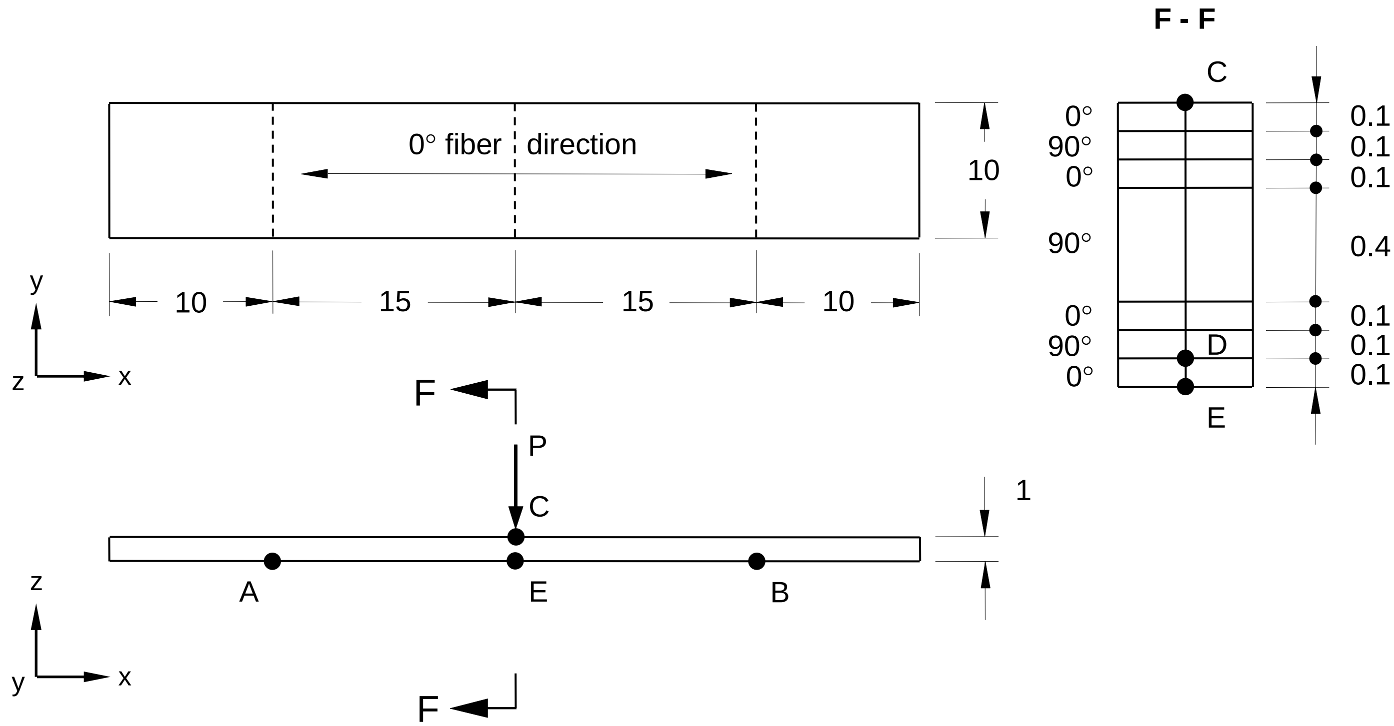


Figure 1. Laminated Strip

Autodesk Inventor Nastran Analysis Model Filename

- vm3_2_1.nas

Model Data

Finite Element Modeling

- 33 nodes, 20 5-DOF/node laminate plate elements (2x10 elements, on the smaller and larger sides of the plate, respectively).

Units

meter/Newton/second

Model Geometry

Length: L = 0.050 m
Width: w = 0.010 m
Thickness: t = 0.001 mm

Material Properties

Young's Modulus: E_1 = 1.0 E+5 MPa
Young's Modulus: E_2 = 5.0 E+3 MPa
Poisons Ratio: ν_{12} = 0.4
Poisons Ratio: ν_{23} = 0.3
Shear Modulus of Elasticity: G_{12} = 3.0 E+3 MPa
Shear Modulus of Elasticity: G_{13} = G_{23} = 2.0 E+3 MPa

Boundary Conditions

The model is simply supported at points A and B. A distributed load P = 10N/mm is applied at point C (X=25, Z=1).

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**NAFEMS**	**Autodesk Inventor Nastran**	**Error** (%)
Interlaminar Shear Stress at D (MPa)	-4.1	-4.1	0.0
Deflection in Z-direction at E (mm)	-1.06	-1.06	0.0

References

1. NAFEMS Report R0031, *Laminated Strip*. Test No. R0031/1, date issued 17/12/98/1.

3.2.2 Wrapped Thick Cylinder

Problem Description

Figure 1 shows the geometry of the cylinder. A static analysis is performed on the model (an 8^{th} part of the cylinder). The hoop stresses in the inner and outer cylinders are determined. All dimensions are in millimeters.

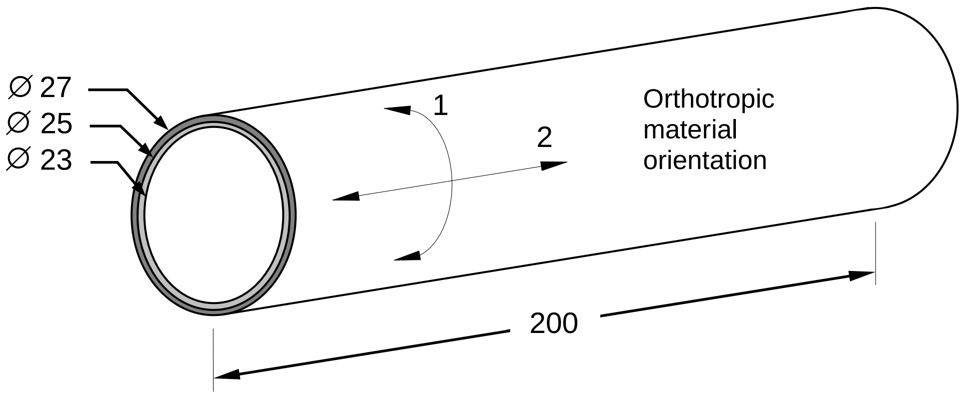


Figure 1. Wrapped Thick Cylinder

Autodesk Inventor Nastran Analysis Model Filename

- vm3_2_2.nas

Model Data

Finite Element Modeling

- 33 nodes, 100 5-DOF/node laminate plate elements (10x10 elements, on an 8^{th} part of the cylinder model, using symmetry)

Units

meter/Newton/second

Model Geometry

Length: L = 0.200 m
Radius: R_1 = 0.023 m
Radius: R_2 = 0.025 m
Radius: R_3 = 0.027 m

Material Properties

Inner Cylinder Isotropic
Young's Modulus: E = 2.1 E+5 MPa
Poisons Ratio: ν = 0.3

Outer Cylinder Circumferentially Wound

Young's Modulus: E_1 = 1.3 E+5 MPa
Young's Modulus: E_2 = 5.0 E+3 MPa
Poisons Ratio: ν = 0.25
Shear Modulus of Elasticity: G_{12} = 1.0 E+4 MPa
Shear Modulus of Elasticity: $G_{13} = G_{23}$ = 5.0 E+3 MPa

Boundary Conditions

Symmetry boundary conditions are applied to the 8th model cylinder.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**NAFEMS**	**MSC.Nastran**	**Autodesk Inventor Nastran**	**Error** (%)
Hoop Stress in Inner Cylinder	1,497.5	1,518.1	1,518.2	1.4
Hoop Stress in Outer Cylinder	816.9	975.2	975.2	19.4

References

1. NAFEMS Report R0031, *Wrapped Thick Cylinder*. Test No. R0031/2, date issued 17/12/98/1.

3.3 Shell Element Test Cases

The following linear static verification problems using standard NAFEMS Benchmarks are performed using shell plate elements.

3.3.1 Elliptic Membrane

Problem Description

Figure 1 shows the membrane model. Static analysis is performed on the elliptical shell model. The normal stress at point C is determined. All dimensions are in meters.

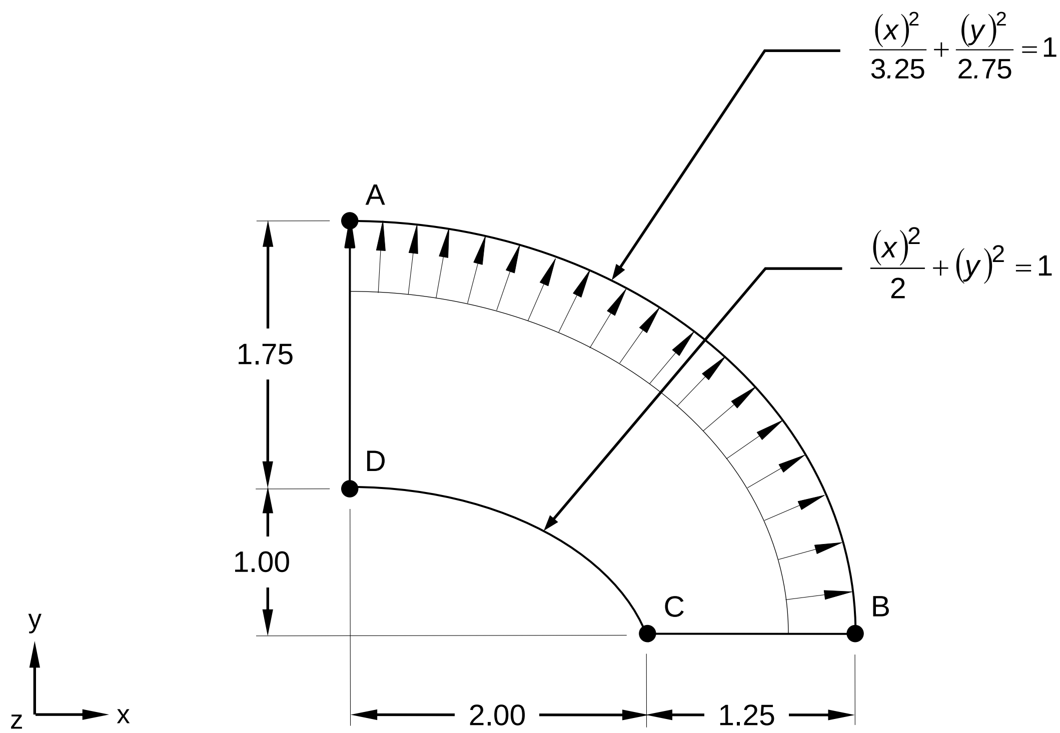


Figure 1. Elliptic Membrane

Autodesk Inventor Nastran Analysis Model Filename

- vm3_3_1.nas

Model Data

Finite Element Modeling

- A ¼ model (90 degrees sector of two ellipses) is created using 231 nodes, 200 5-DOF/node quadrilateral elements and 20 bar elements on outer edge AB (used to transfer load to the outer edge nodes in the proper direction).

Units

meter/Newton/second

Model Geometry

Length: L_1 = 1.75 m and L_2 = 1.25 m
Thickness: t = 0.1 m

Material Properties

Young's Modulus: E = 210.0 E+3 MPa
Poisson's Ratio: ν = 0.3

Boundary Conditions

All nodes along edge AD are restrained in the X and Z-translations, and X and Y-rotations. All nodes along edge BC are restrained in the in the Y and Z-translations, and X and Y-rotations. A uniform outward pressure P =10 MPa at outer edge AB is applied to the model.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Tangential (Normal) Edge Stress at Point C (MPa)	92.7	91.2	1.6

References

1. NAFEMS Finite Element Methods & Standards, *The Standard NAFEMS Benchmarks*, Rev. 3. Glasgow: NAFEMS, 1990. Test No. LE1.

3.3.2 Z-Section Cantilever

Problem Description

Figure 1 shows the Z-Section cantilever model. Static analysis is performed on the shell model. The plate minimal principal stress at point A is determined. All dimensions are in meters.

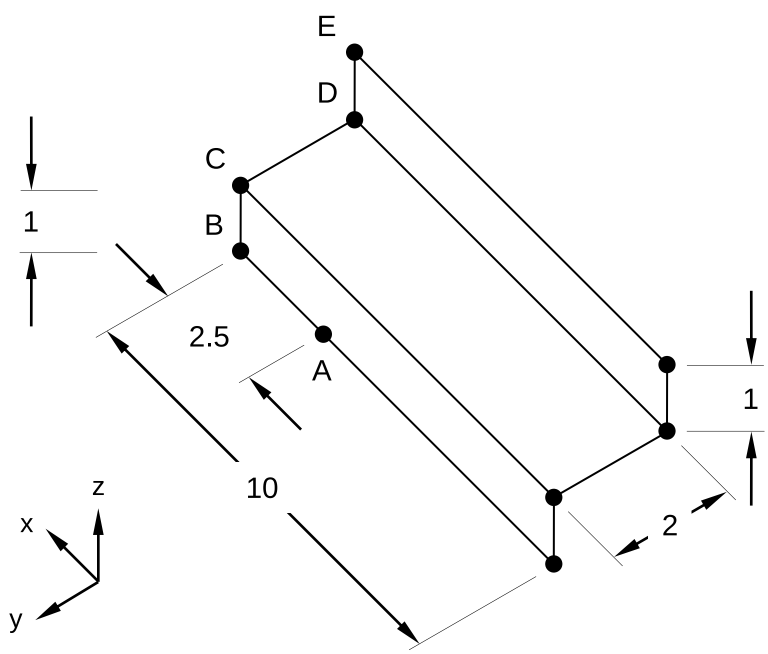


Figure 1. Z-Section Cantilever

Autodesk Inventor Nastran Analysis Model Filename

- vm3_3_2.nas

Model Data

Finite Element Modeling

- 36 nodes, 24 5-DOF/node quadrilateral elements

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Width: w = 2.0 m
Height: h = 1.0 m
Thickness: t = 0.1 m

Material Properties

Young's Modulus: E = 210.0 E+3 MPa
Poisson's Ratio: ν = 0.3

Boundary Conditions

The nodes at B, C, D, and E are restrained in all translations and rotations. A torque of 1.20 MN-m is applied at the free end by way of two uniformly distributed edge shears P = 0.60 MN at each flange.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Shell Min Principal Stress at Point A (MPa)	-108	-108	0.0

References

1. NAFEMS Finite Element Methods & Standards, *The Standard NAFEMS Benchmarks*, Rev. 3. Glasgow: NAFEMS, 1990. Test No. LE5.

3.3.3 Skew Plate Under Normal Pressure

Problem Description

Figure 1 shows the skew plate model. Static analysis is performed on the shell model. The plate bottom major stress at center of plate is determined. All dimensions are in meters.

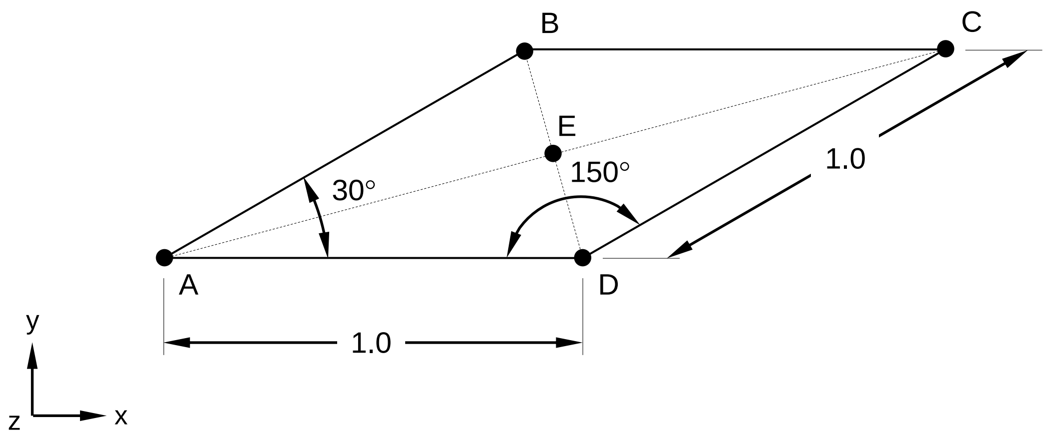


Figure 1. Skew Plate

Autodesk Inventor Nastran Analysis Model Filename

- vm3_3_3.nas

Model Data

Finite Element Modeling

- 81 nodes, 64 6-DOF/node quadrilateral elements

Units

meter/Newton/second

Model Geometry

Length: L = 1 m
Thickness: t = 0.01 m
Internal Skew Angles: θ = 30° and 150°

Material Properties

Young's Modulus: E = 210.0 E+3 MPa
Poisson's Ratio: ν = 0.3

Boundary Conditions

All nodes along all edges are restrained in the Z-translation. Point A has X and Y-translations restrained, while point D has Y-translation restrained. A uniform pressure P = 700 Pa is applied in the negative Z-direction.

Solution Type

Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Bottom Major Stress at Center of Plate (MPa)	0.802	0.769	4.1

References

1. NAFEMS Finite Element Methods & Standards, *The Standard NAFEMS Benchmarks*, Rev. 3. Glasgow: NAFEMS, 1990. Test No. LE6.

4. Normal Modes/Eigenvalue Verification Using Theoretical Solutions

The purpose of these normal mode dynamic test cases is to verify the functionality of Autodesk Inventor Nastran using theoretical solutions of well-known engineering normal mode dynamic problems. The test cases are basic in form and most of them have closed-form theoretical solutions.

The theoretical solutions given in these examples are from reputable engineering texts. For each case, a specific reference is cited. All theoretical reference texts are listed in Appendix A.

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

For most cases, discrepancies between Autodesk Inventor Nastran computed and theoretical results are minor and can be considered negligible. To produce exact results, for most cases, a larger number of elements would need to be used. Element quantity is chosen to achieve reasonable engineering accuracy in a reasonable amount of time.

4.1 Two Degree of Freedom Undamped Free Vibration – Principle Modes

Problem Description

Figure 1 shows the model of the two degree of freedom system. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the system are determined.

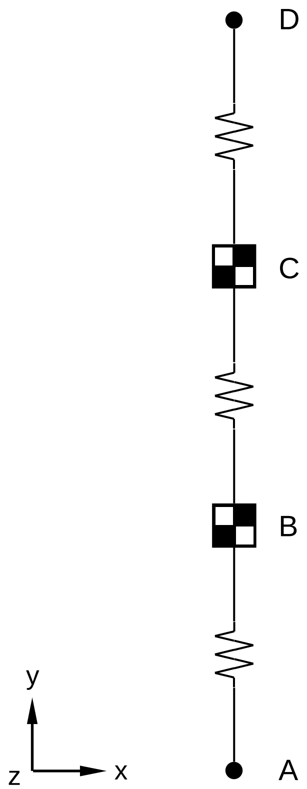


Figure 1. Two Degree of Freedom System

Autodesk Inventor Nastran Analysis Model Filenames

- vm4_1d.nas – Diagonal mass formulation
- vm4_1c.nas – Coupled mass formulation

Model Data

Finite Element Modeling

- Four nodes are created on the Y-axis (A, B, C, D). Three DOF springs are created with stiffness of 1 N/m and a stiffness reference coordinate system being uniaxial. Mass elements with a mass of 1 kg are created. DOF springs and mass elements are used.

Units

meter/Newton/second

Physical Properties

Mass: m = 1 kg
Spring Constant: k = 1 N/m

Boundary Conditions

The end nodes (A and D) are constrained in all DOF. The other nodes (B and C) are constrained in all DOF except the Y-translation.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

	Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	0.1592	0.1592	0.0	0.1592	0.0
2	0.2757	0.2757	0.0	0.2757	0.0

References

1. Tse, F., Morse, I., and Hinkle, R., *Mechanical Vibrations*, 2nd Edition. Boston: Allyn and Bacon, Inc., 1978. pp. 145-149.

4.2 Three Degree of Freedom Torsional System

Problem Description

Figure 1 shows the torsional system. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the system are determined.

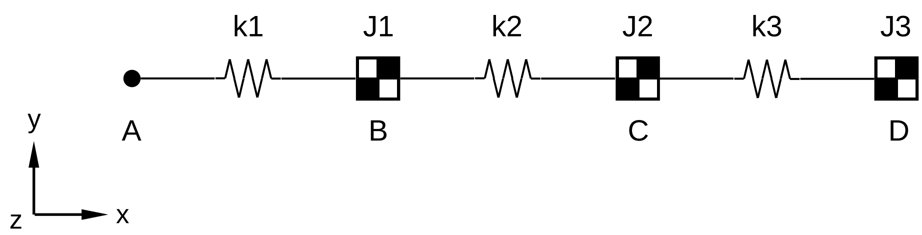


Figure 1. Three Degree of Freedom Torsional System

Autodesk Inventor Nastran Analysis Model Filenames

- vm4_2d.nas – Diagonal mass formulation
- vm4_2c.nas – Coupled mass formulation

Model Data

Finite Element Modeling

- Four nodes are created on the X-axis (A, B, C, D). Three DOF springs are created with stiffness of 1 N/m and a stiffness reference coordinate system being uniaxial. Three mass elements are created with a mass coordinate system = 1 and mass inertia system of: 0.1, 0.0, 0.0, 0.0, 0.0, 0.0. DOF springs and mass elements are used.

Units

meter/Newton/second

Physical Properties

Mass: $J = J1 = J2 = J3 = 0.1$ (mass)
Spring Constant: $k = k1 = k2 = k3 = 1$ N-m (stiffness)

Boundary Conditions

One end node (node A), is constrained in all translations and rotations. All other nodes are constrained in all translations and rotations except the X-rotation.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

	Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	0.2240	0.2240	0.0	0.2240	0.0
2	0.6276	0.6276	0.0	0.6276	0.0
3	0.9069	0.9069	0.0	0.9069	0.0

References

1. Tse, F., Morse, I., and Hinkle, R., *Mechanical Vibrations*, 2nd Edition. Boston: Allyn and Bacon, Inc., 1978. pp. 153-155.

4.3 Cantilever Plate Eigenvalue Problem

Problem Description

Figure 1 shows the cantilever plate. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The first two modes of vibration of the plate are determined. All dimensions are in inches.

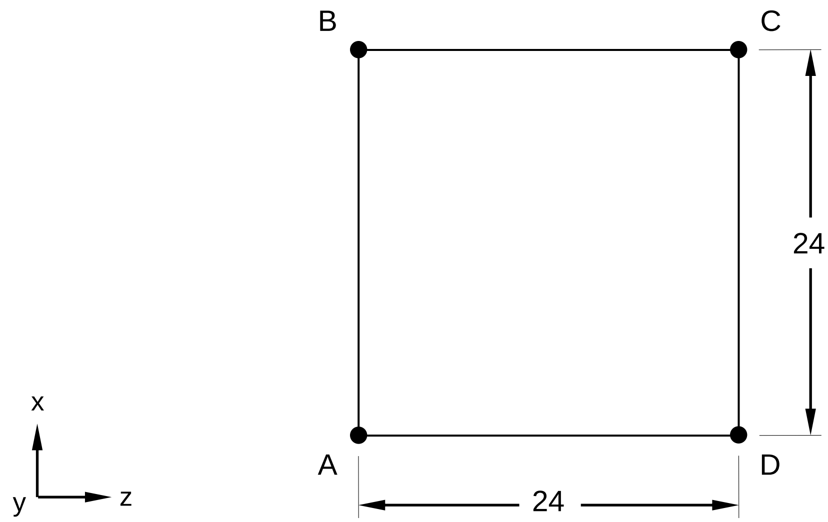


Figure 1. Cantilever Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm4_3d4.nas – 5-DOF/node, CQUAD4 elements, diagonal mass formulation
- vm4_3c4.nas – 5-DOF/node, CQUAD4 elements, coupled mass formulation
- vm4_3dR.nas – 6-DOF/node, CQUADR elements, diagonal mass formulation
- vm4_3cR.nas – 6-DOF/node, CQUADR elements, coupled mass formulation

Model Data

Finite Element Modeling

- 400 nodes, 361 quadrilateral plate elements

Units

inch/pound/second

Model Geometry

Length: L = 24 in
Thickness: t = 1 in

Material Properties

Young's Modulus: E = 29.5 E+3 ksi
Poisson's Ratio: ν = 0.2
Mass Density: ρ = 0.28356 lbs/in^3

Boundary Conditions

All the nodes on edge AB are restrained in all translations and rotations. The load is mass distribution based on density and given geometry (as nodal forces in the X, Y and Z-directions).

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

		Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Element**	**Natural Frequency** (s)	**Natural Frequency** (s)	**Error** (%)	**Natural Frequency** (s)	**Error** (%)
1	CQUAD4	0.01790	0.01757	1.8	0.01755	2.0
	CQUADR	0.01790	0.01757	1.8	0.01755	2.0
2	CQUAD4	0.00732	0.00696	4.9	0.00695	5.1
	CQUADR	0.00732	0.00696	4.9	0.00695	5.1

References

1. Harris, C. M., and Crede, C. E., *Shock and Vibration Handbook*. New York: McGraw-Hill, Inc., 1976.

4.4 Bathe and Wilson Frame Eigenvalue Problem

Problem Description

Figure 1 shows the frame model. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The first three eigenvalues are determined. All dimensions are in feet.

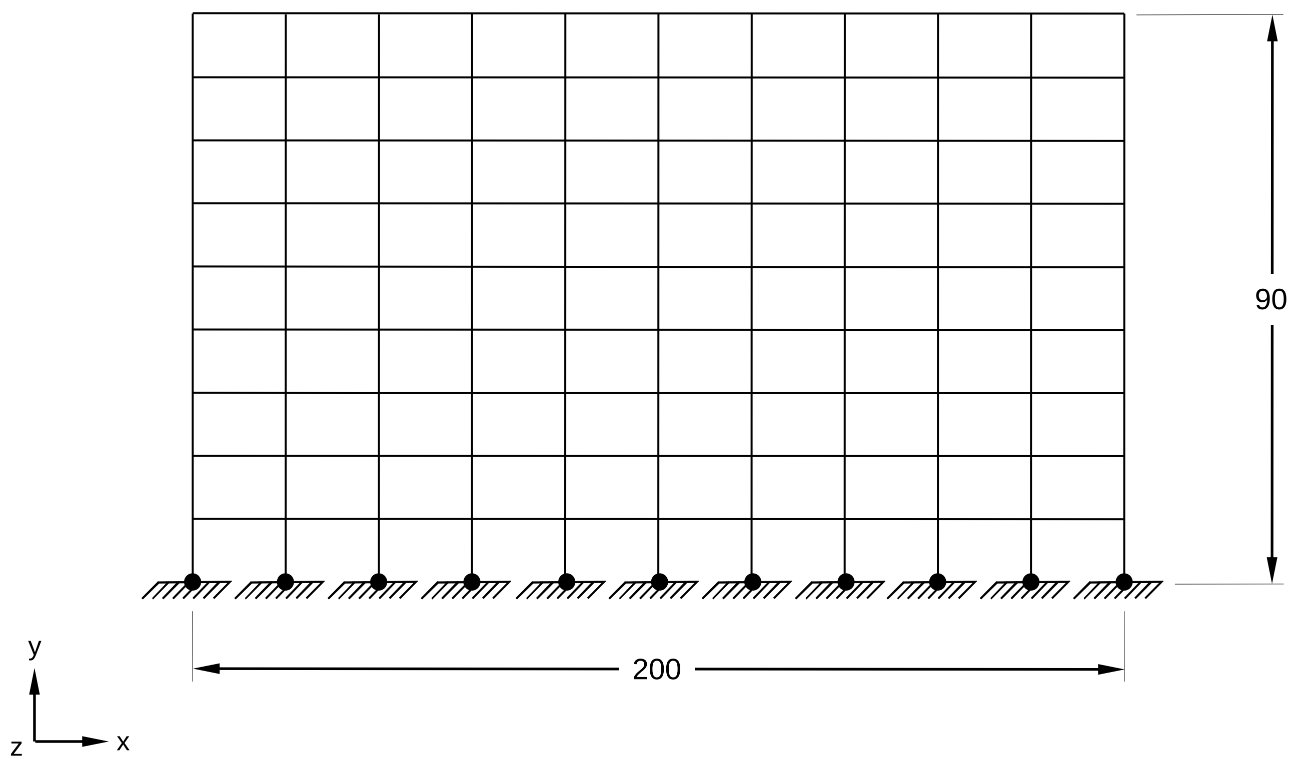


Figure 1. Bathe and Wilson Frame

Autodesk Inventor Nastran Analysis Model Filenames

- vm4_4d.nas – Diagonal mass formulation
- vm4_4c.nas – Coupled mass formulation

Model Data

Finite Element Modeling

- 110 nodes, 189 beam elements

Units

feet/Kip/second

Model Geometry

Length: L = 200 feet (10 @ 20)
Height: h = 90 feet (9 @ 10)

Cross Sectional Properties

Area: A = 3 ft^2
Moment of Inertia: I_z = 1 ft^4 (for each beam and column)

Material Properties

Young's Modulus: E = 4.32 E+5 k/ft^2
Poisson's Ratio: ν = 0.3
Mass per Unit Length: m = 3 kip-sec^2 /ft /ft

Boundary Conditions

The supports are restrained in all translations and rotations. All the other nodes are restrained in the Z-translation, X and Y-rotations. The load is mass distribution based on density and given geometry.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

	Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Eigenvalue** (rad/s)	**Eigenvalue** (rad/s)	**Error** (%)	**Eigenvalue** (rad/s)	**Error** (%)
1	0.589541	0.588858	0.1	0.589827	0.0
2	5.52695	5.47130	1.0	5.55036	0.4
3	16.5878	16.1568	2.6	16.7757	1.1

References

1. Bathe, K. J., and Wilson, E. L., *Large Eigenvalue Problems in Dynamic Analysis*. Journal of the Engineering Mechanics Division, ASCE, Vol. 98, No. EM6, Paper 9433, 1972.

4.5 Natural Frequency of a Cantilevered Mass

Problem Description

Figure 1 shows the cantilevered mass model. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequency of vibration of the mass attached to the end of the light cantilever beam is determined. All dimensions are in inches.

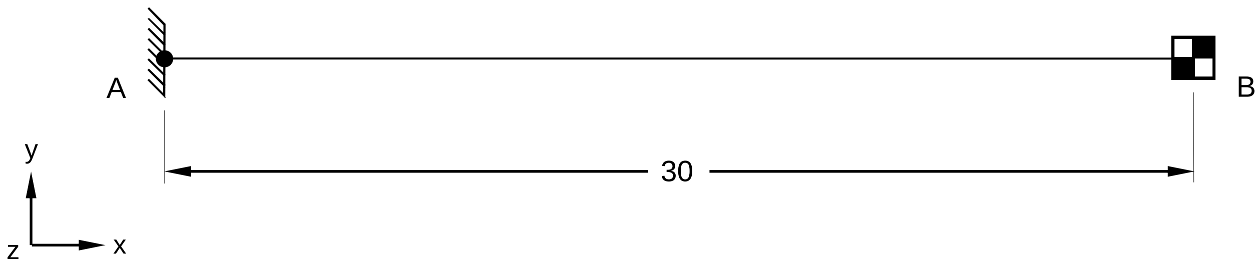


Figure 1. Cantilevered Mass Model

Autodesk Inventor Nastran Analysis Model Filenames

- vm4_5d.nas – Diagonal mass formulation
- vm4_5c.nas – Coupled mass formulation

Model Data

Finite Element Modeling

- 2 nodes, 1 beam element, and 1 mass element

Units

inch/pound/second

Model Geometry

Length: L = 30 in

Cross Sectional Properties

Moment of Inertia: I = 1.33333 in^4

Material Properties

Young's Modulus: E = 30.0 E+3 ksi
Poisson's Ratio: ν = 0.3
Mass: m = 0.1 lb-sec^2/in

Boundary Conditions

The beam is fixed at point A (all translations and rotations are restrained). The other end of the beam (point B) is restrained in the Z-translation and X-rotation. The load is a lamped mass at free end of the beam.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

	Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	33.553	33.553	0.0	33.553	0.0

References

1. Thompson, W. T., *Vibration Theory and Applications,* 2nd Edition. New Jersey, Englewood Cliff: Prentice-Hall, Inc., 1965.

4.6 Fundamental Frequency of a Simply Supported Beam

Problem Description

Figure 1 shows the simply supported beam model. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The fundamental frequency of the simply supported beam of uniform cross-section is determined. All dimensions are in inches.

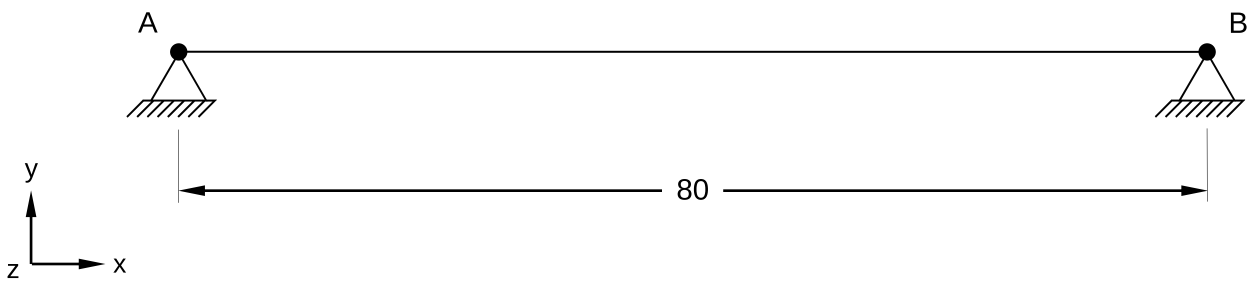


Figure 1. Simply Supported Beam

Autodesk Inventor Nastran Analysis Model Filenames

- vm4_6d.nas – Diagonal mass formulation
- vm4_6c.nas – Coupled mass formulation

Model Data

Finite Element Modeling

- 5 nodes, 4 beam elements

Units

inch/pound/second

Model Geometry

Length: L = 80 in
Height: h = 2 in

Cross Sectional Properties

Area: A = 4 in^2
Moment of Inertia: I = 1.33333 in^4
Square Cross Section = (2 in x 2 in)

Material Properties

Young's Modulus: E = 30.0 E+3 ksi
Poisson's Ratio: ν = 0.3

Boundary Conditions

The beam is pinned at both ends (all translations are restrained). The load is a uniform mass along the beam w = 1.124 lb/in (m = 1.124/386.4 = 0.002909 lb-sec^2 /in^2).

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

	Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	28.766	28.520	0.9	28.782	0.1

References

1. Thompson, W. T., *Vibration Theory and Applications,* 2nd Edition. New Jersey, Englewood Cliff: Prentice-Hall, Inc., 1965.

4.7 Natural Frequencies of a Cantilever Beam

Problem Description

Figure 1 shows the cantilever beam model. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The first three natural frequencies of the cantilever beam of uniform cross-section are determined. All dimensions are in inches.

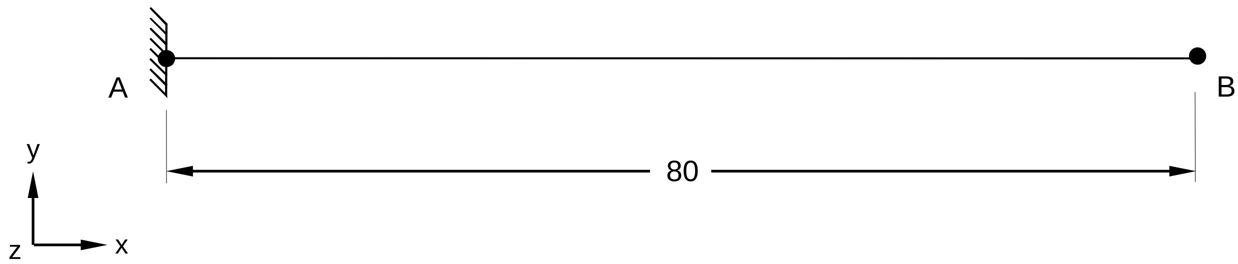


Figure 1. Cantilever Beam

Autodesk Inventor Nastran Analysis Model Filenames

- vm4_7d.nas – Diagonal mass formulation
- vm4_7c.nas – Coupled mass formulation

Model Data

Finite Element Modeling

- 21 nodes, 20 beam elements

Units

inch/pound/second

Model Geometry

Length: L = 80 in
Height: h = 2 in

Cross Sectional Properties

Area: A = 4 in^2
Moment of Inertia: I = 1.33333 in^4
Square Cross Section = (2 in x 2 in)

Material Properties

Young's Modulus: E = 30.0 E+3 ksi
Poisson's Ratio: ν = 0.3

Boundary Conditions

The beam is fixed at end A (all translations and rotations are restrained). All other nodes are restrained in the Y-translation, X and Z-rotations. The load is a uniform mass along the beam w = 1.124 lb/in (m = 1.124/386.4 = 0.002909 lb-sec^2 /in^2).

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

	Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	10.247	10.239	0.1	10.252	0.0
2	64.221	63.907	0.5	64.202	0.0
3	179.82	178.12	0.9	179.56	0.1

References

1. Thompson, W. T., *Vibration Theory and Applications,* 2nd Edition. New Jersey, Englewood Cliff: Prentice-Hall, Inc., 1965.

4.8 Vibration of a String Under Tension

Problem Description

Figure 1 shows the string (beam) model. A linear prestress modal analysis is performed using the subspace iterative method. The stress, force in the string (under initial strain and fixed ends), and the first three natural frequencies of lateral vibration of the stretched string are determined. All dimensions are in inches.

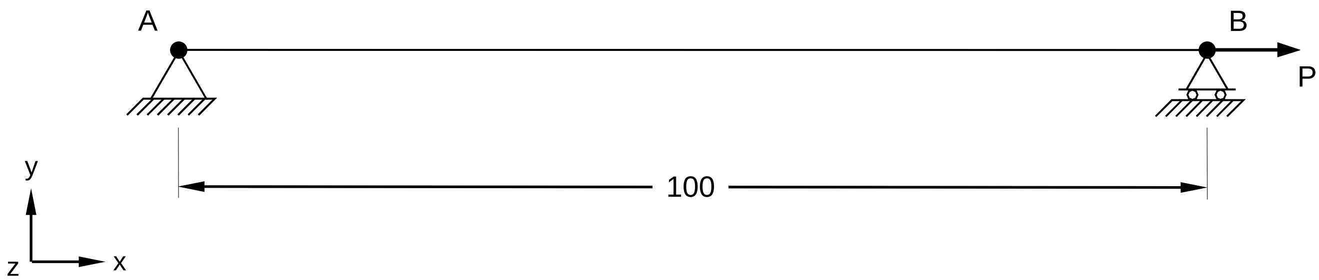


Figure 1. String Under Tension

Autodesk Inventor Nastran Analysis Model Filenames

- vm4_8d.nas – Diagonal mass formulation
- vm4_8c.nas – Coupled mass formulation

Model Data

Finite Element Modeling

- 14 nodes, 13 beam elements

Units

inch/pound/second

Model Geometry

Length: L = 100 in

Cross Sectional Properties

Area: A = 3.06796 E-3 in^2

Material Properties

Young's Modulus: E = 30.0 E+3 ksi
Poisson's Ratio: ν = 0.3
Mass Density: ρ = 7.3 E-4 lb-sec^2 /in^4

Boundary Conditions

The beam is restrained in all translations at end A, while end B has the Y and Z-translations restrained. All other nodes are restrained in the Z-translation, X and Y-rotations. The load is an initial strain e_0 of 0.00543248 and a uniform mass along the string based on the mass density. This translates into a load P = stress x area = E x e_0 x A = 500 lb.

Solution Type

Linear Prestress – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1, Table 2 and Table 3.

Table 1. Force Results

Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Force (lb)	**Force** (lb)	**Error** (%)	**Force** (lb)	**Error** (%)
500	500	0.0	500	0.0

Table 2. Stress Results

Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Stress (psi)	**Stress** (psi)	**Error** (%)	**Stress** (psi)	**Error** (%)
162,974	162,973	0.0	162,973	0.0

Table 3. Frequency Results

	Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	74.708	74.645	0.1	74.708	0.0
2	149.42	148.88	0.4	149.42	0.0
3	224.12	222.18	0.9	224.13	0.0

References

1. Thompson, W. T., *Vibration Theory and Applications,* 2nd Edition. New Jersey, Englewood Cliff: Prentice-Hall, Inc., 1965.

4.9 Vibration of a Wedge

Problem Description

Figure 1 shows the wedge plate model. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The fundamental frequency of lateral vibration of the wedge plate is determined. All dimensions are in inches.

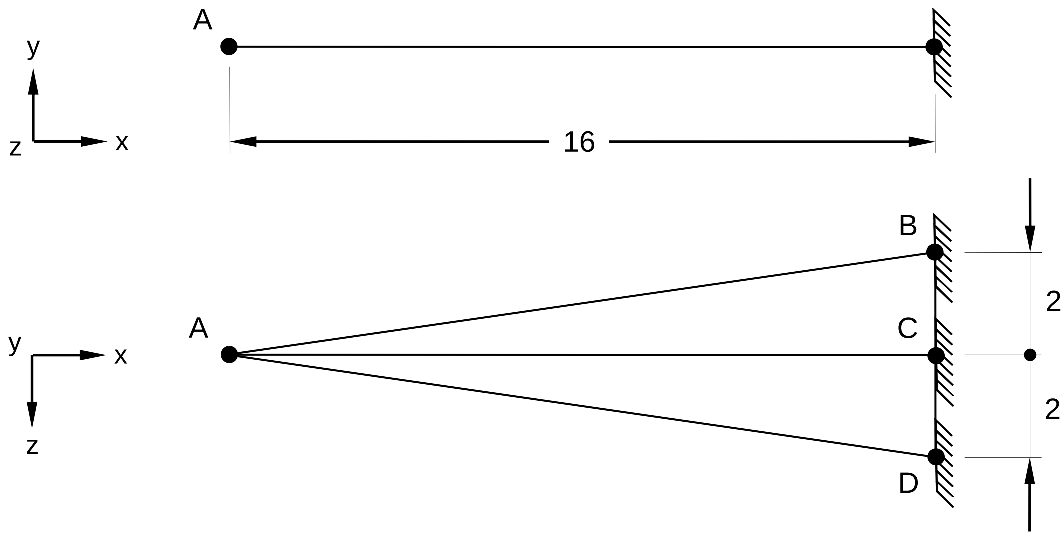


Figure 1. Wedge Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm4_9d.nas – Diagonal mass formulation
- vm4_9c.nas – Coupled mass formulation

Model Data

Finite Element Modeling

- 31 nodes, 38 5-DOF/node triangle elements

Units

inch/pound/second

Model Geometry

Length: L = 16 in
Width: w = 2 in
Thickness: t = 1 in

Material Properties

Young's Modulus: E = 30.0 E+3 ksi
Shear Modulus of Elasticity: G = 1.5 E+3 ksi
Mass Density: ρ = 7.28 E-4 lb-sec^2 /in^4

Boundary Conditions

At fixed end (edge BCD) the nodes are restrained in all translations and rotations. The load is a uniform mass over surface area based on mass density and material thickness.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

	Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	259.16	257.34	0.7	259.60	0.2

References

1. Timoshenko, S. and Young, D. H., *Vibration Problems in Engineering*, 3rd Edition. New York: D. VanNostrand Co., Inc., 1955.

5. Normal Modes/Eigenvalue Verification Using Standard NAFEMS Benchmarks

The purpose of these normal mode dynamic test cases is to verify the functionality of Autodesk Inventor Nastran using standard benchmarks published by NAFEMS (National Agency for Finite Element Methods and Standards, National Engineering Laboratory, Glasgow, U.K.).

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

5.1 Bar Element Test Cases

The following normal mode/eigenvalue verification problems using standard NAFEMS Benchmarks are performed using bar elements.

5.1.1 Pin-ended Cross – In-plane Vibration

Problem Description

Figure 1 shows the pin-ended cross. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the pin-ended cross are determined. All dimensions are in meters.

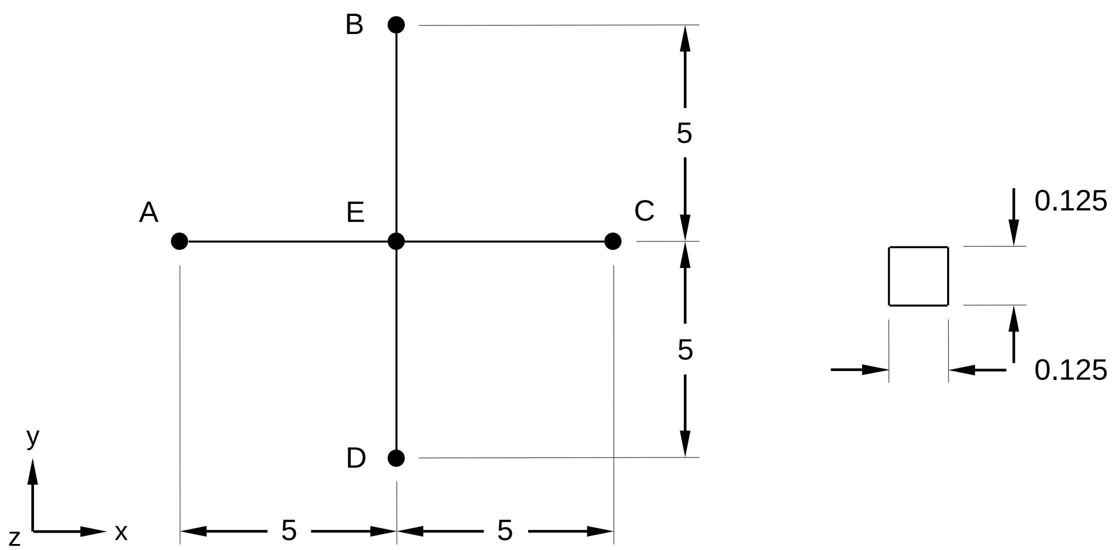


Figure 1. Pin-ended Cross

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_1_1c.nas – 5-DOF/node, coupled mass formulation
- vm5_1_1d.nas – 5-DOF/node, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Coupling between flexural and extensional behavior. Repeated and closed eigenvalues.
- 17 nodes, 16 bar elements; four elements per arm (AE, BE, DE, CE)

Units

meter/Newton/second

Model Geometry

Length: L = 10 m

Cross Sectional Properties

Square Cross Section = (0.125 m x 0.125 m)
Area: A = 0.015625 m^2
Shear Ratio:
Y = 0
Z = 0

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Shear Modulus of Elasticity: G = 8.01 E+10 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.29

Boundary Conditions

Points A, B, C, D (nodes 2, 3, 4, 5) are constrained in all directions except for the Z-rotation. Point E (node 1, at coordinates 0,0,0) is constrained in the Z-translation and X-rotation. All other nodes (6–17) are constrained in the Z-translation and X and Y-rotations.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

	NAFEMS	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	11.34	11.33	0.1	11.34	0.0
2, 3	17.69	17.66	0.2	17.69	0.0
4	17.72	17.69	0.2	17.72	0.0
5	45.48	45.02	1.0	45.48	0.0
6, 7	57.36	56.07	2.2	57.37	0.0
8	57.68	56.35	2.3	57.69	0.0

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 1.

5.1.2 Free Square Frame – In-plane Vibration

Problem Description

Figure 1 shows the free square frame. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the square are determined. All dimensions are in meters.

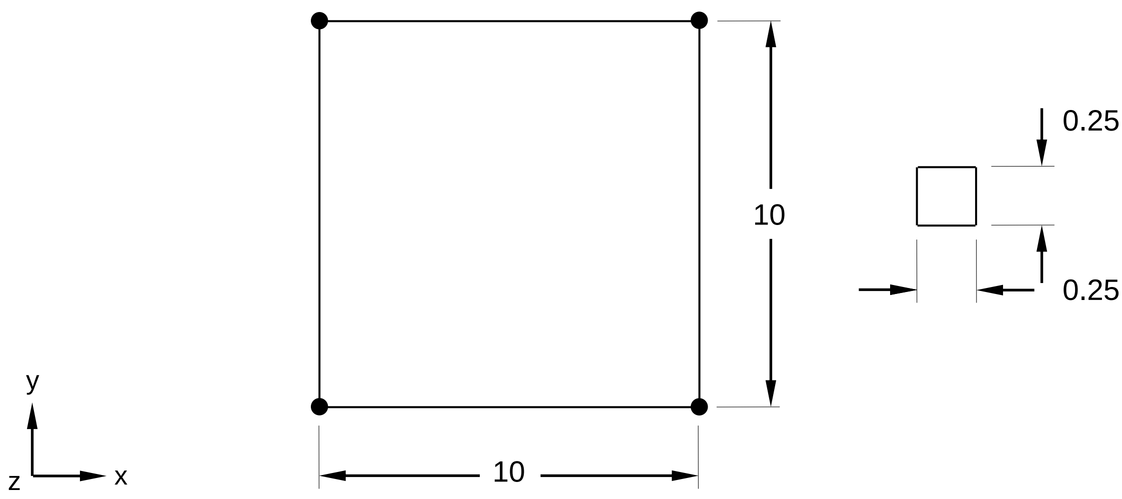


Figure 1. Free Square Frame

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_1_2c.nas – 5-DOF/node, coupled mass formulation
- vm5_1_2d.nas – 5-DOF/node, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Coupling between flexural and extensional behavior. Rigid body modes (3 modes). Repeated and close eigenvalues.
- 16 nodes, 16 bar elements; four elements per arm.

Units

meter/Newton/second

Model Geometry

Length: L = 10 m

Cross Sectional Properties

Square Cross Section = (0.25 m x 0.25 m)
Shear Ratio:
$Y = 1.0$
$Z = 1.0$

Material Properties

Young's Modulus: $E = 200.0$ E+9 N/m^2
Mass Density: $\rho = 8000$ kg/m^3

Boundary Conditions

Constraint Set 1: All the nodes are constrained in the Z-translation and X and Y-rotations.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

	NAFEMS	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
4	3.262	3.259	0.1	3.259	0.1
5	5.665	5.660	0.1	5.662	0.1
6, 7	11.14	10.89	2.3	11.13	0.1
8	12.83	12.74	0.8	12.79	0.3
9	24.66	23.53	4.6	24.61	0.2
10, 11	28.81	28.13	2.3	28.70	0.4

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 3.

5.1.3 Cantilever with Off-center Point Masses

Problem Description

Figure 1 shows the cantilever with off-center point masses. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the cantilever are determined. All dimensions are in meters.

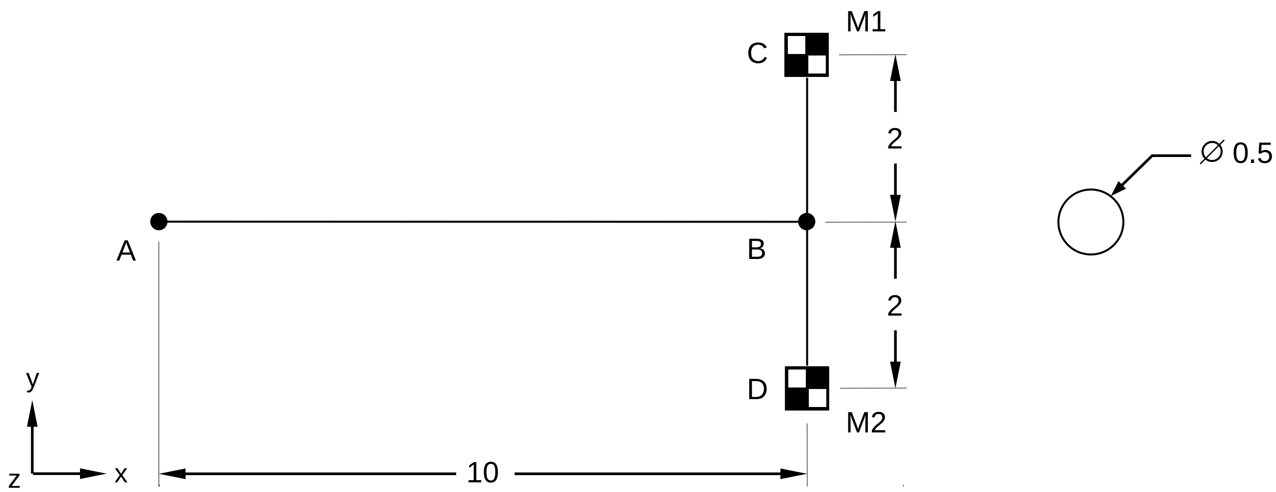


Figure 1. Cantilever with Off-center Point Mass

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_1_3c.nas – 5-DOF/node, coupled mass formulation
- vm5_1_3d.nas – 5-DOF/node, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Coupling between torsional and flexural behavior. Internal axis non-coincident with flexibility axis. Discrete mass, rigid links. Close eigenvalues.
- 8 nodes, 9 elements; five bar elements along cantilever (from point A to point B, as shown in Figure 1), two mass elements (at point C, M1 = 10,000 kg, and point D, M2 = 1,000 kg), and two rigid elements (between points C and B, and points B and D, respectively).

Units

meter/Newton/second

Model Geometry

Length: L_1 = 10 m and L_2 = 4 m

Cross Sectional Properties

Circular Cross Section = 0.5 m
Shear Ratio:
Y = 1.128
Z = 1.128

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.3

Boundary Conditions

Point A (node 1) is fully constrained in all translations and rotations (see Figure 1).

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

	NAFEMS	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	1.72	1.71	0.6	1.72	0.0
2	1.73	1.72	0.6	1.73	0.0
3	7.41	7.56	2.0	7.55	1.9
4	9.97	9.95	0.2	9.95	0.2
5	18.16	17.68	2.6	18.07	0.5
6	26.97	26.78	0.7	26.72	0.9

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 4.

5.1.4 Deep Simply-Supported Beam

Problem Description

Figure 1 shows the deep simply supported beam. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the simply supported beam are determined. All dimensions are in meters.

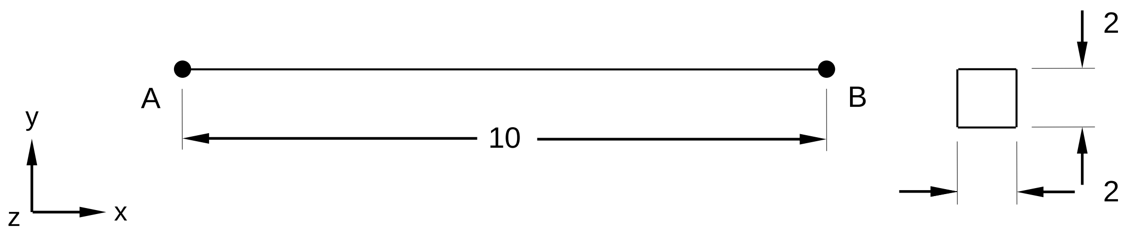


Figure 1. Deep Simply – Supported Beam

Model Input and FEMAP Neutral Base Filenames

- vm5_1_4c.nas – 5-DOF/node, coupled mass formulation
- vm5_1_4d.nas – 5-DOF/node, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Shear deformation and rotary inertial (Timoshenko beam). Possibility of missing extensional modes when using iteration solution methods. Repeated eigenvalues.
- 6 nodes, 5 bar elements (from point A to point B, as shown in Figure 1).

Units

meter/Newton/second

Model Geometry

Length: L = 10 m

Cross Sectional Properties

Square Cross Section = (2 m x 2 m)
Shear Ratio:
Y = 1.176923
Z = 1.176923

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.3
Boundary Conditions

Point A (node 1) is constrained in the X, Y, and Z-translations and X-rotation. Point B (node 6) is constrained in the Y and Z-translations.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

	NAFEMS	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1, 2	42.57	42.58	0.0	42.71	0.3
3	77.84	77.19	0.8	77.83	0.0
4	125.51	124.45	0.8	125.51	0.0
5, 6	145.46	145.43	0.0	150.75	3.7
7	241.24	224.02	7.1	241.20	0.0
8, 9	267.01	263.73	1.2	301.07	12.8

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 5.

5.1.5 Pin-ended Double Cross – In-plane Vibration

Problem Description

Figure 1 shows the pin-ended double cross. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The first 16 natural frequencies of vibration of the pin-ended double cross are determined. All dimensions are in meters.

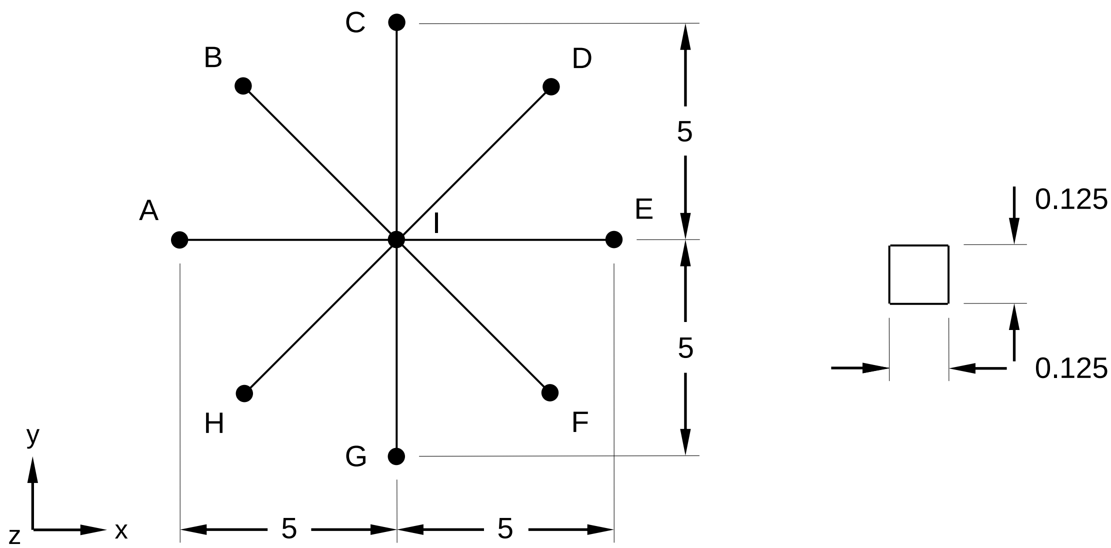


Figure 1. Pin-ended Double Cross

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_1_5c.nas – 5-DOF/node, coupled mass formulation
- vm5_1_5d.nas – 5-DOF/node, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Coupling between flexural and extensional behavior. Repeated and closed eigenvalues.
- 33 nodes, 32 bar elements; four elements per arm.

Units

meter/Newton/second

Model Geometry

Length: L = 10 m

Cross Sectional Properties

Square Cross Section = (0.125 m x 0.125 m)
Area: A = 0.015625 m^2
Moment of Inertia: I = 2.034E-5 m^4

Material Properties

Young's Modulus: E = 20.0 E+10 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.29

Boundary Conditions

Points A, B, C, D, E, F, G, H are constrained in all directions except for the Z-rotation. All other nodes are constrained in the Z-translation, X and Y-rotations.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

	NAFEMS	Autodesk Inventor Nastran Diagonal Mass Formulation		Autodesk Inventor Nastran Coupled Mass Formulation	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	11.336	11.334	0.0	11.338	0.0
2, 3	17.709	17.664	0.3	17.689	0.1
4,5,6,7,8	17.709	17.693	0.1	17.717	0.0
9	45.345	45.021	0.7	45.483	0.3
10,11	57.390	56.065	2.3	57.371	0.0
12,13,14,15,16	57.390	56.351	1.8	57.690	0.5

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 1.

5.1.6 Circular Ring – In-plane and Out-of-plane Vibration

Problem Description

Figure 1 shows the circular ring. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The first 16 natural frequencies of vibration of the circular ring are determined. All dimensions are in meters.

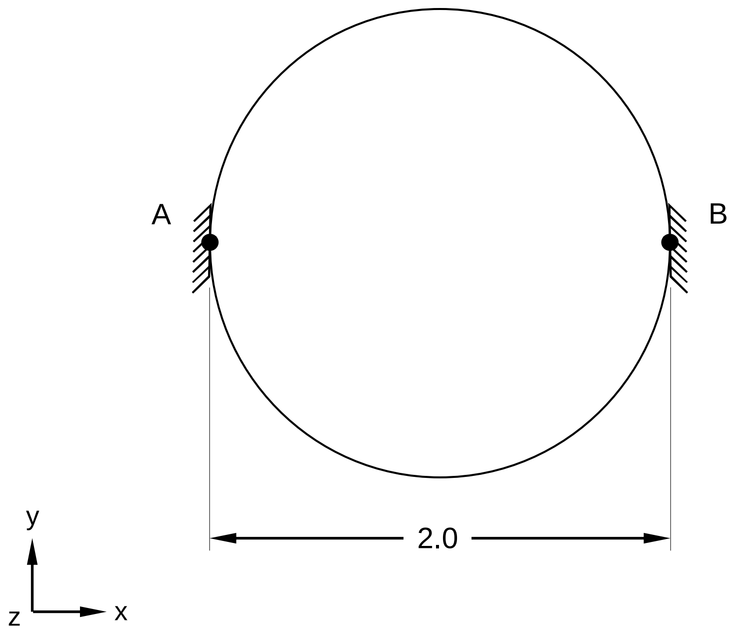


Figure 1. Circular Ring

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_1_6c.nas – 5-DOF/node, coupled mass formulation
- vm5_1_6d.nas – 5-DOF/node, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Rigid Body Modes. Repeated and closed eigenvalues.
- 20 nodes, 20 bar elements

Units

meter/Newton/second

Model Geometry

Radius: $R = 1$ m

Cross Sectional Properties

Area: A = 0.00785398 m^2 ; $A_y = A_z$ = 0.00696148 m^2
Moment of Inertia: I = 4.90874E-6 m^4
Torsional Constant: J = 9.81748E-6 m^4

Material Properties

Young's Modulus: E = 20.0 E+10 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.3

Boundary Conditions

Points A and B are constrained in all translations and rotations.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

	NAFEMS	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
7,8 (out of plane)	51.849	51.617	0.4	52.210	0.7
9,10 (in plane)	53.382	54.054	1.3	53.776	0.7
11,12 (out of plane)	148.77	146.94	1.2	148.92	0.1
13,14 (in plane)	150.99	152.21	0.8	151.26	0.2
15 (out of plane)	286.98	280.42	2.3	285.33	0.6
16 (out of plane)	289.51	280.42	2.3	285.33	0.6

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 6.

5.2 Plate Element Test Cases

The following normal mode/eigenvalue verification problems using standard NAFEMS Benchmarks are performed using plate elements.

5.2.1 Thin Square Cantilevered Plate – Symmetric Modes

Problem Description

Figure 1 shows the thin square cantilevered plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the square plate are determined. All dimensions are in meters.

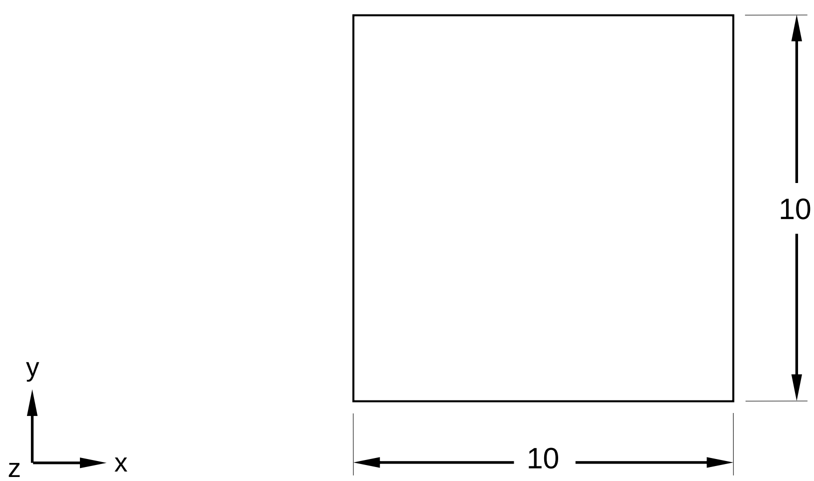


Figure 1. Thin Square Cantilevered Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_2_1c4.nas – 5-DOF/node, CQUAD4, coupled mass formulation
- vm5_2_1d4.nas – 5-DOF/node, CQUAD4, diagonal mass formulation
- vm5_2_1cR.nas – 6-DOF/node, CQUADR, coupled mass formulation
- vm5_2_1dR.nas – 6-DOF/node, CQUADR, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Symmetric modes, symmetric boundary conditions along the cutting plane.
- Test 1 and 2 (vm5_2_1c4 and vm5_2_1d4): 45 nodes, 32 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5_2_1cR and vm5_2_1dR): 45 nodes, 32 6-DOF/node quadrilateral plate elements.
- Only the bottom half of the plate is meshed (10 m x 5 m).
- There are 4 elements vertically, on the 5 m portion of the plate and 8 elements horizontally, on the 10 m portion of the plate.

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Thickness: t = 0.05 m

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.3

Boundary Conditions

Constraints (all tests): The model is constrained using symmetry boundary conditions. The nodes on the left hand side edge are fully constrained in all translations and rotations. All other nodes are constrained in the X and Y-translations and the Z-rotation.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

		Ref. Value	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Element**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	CQUAD4	0.421	0.416	1.2	0.419	0.5
	CQUADR	0.421	0.416	1.2	0.419	0.5
2	CQUAD4	2.582	2.496	3.3	2.611	1.1
	CQUADR	2.582	2.496	3.3	2.611	1.1
3	CQUAD4	3.306	3.152	4.6	3.354	1.4
	CQUADR	3.306	3.152	4.6	3.354	1.4
4	CQUAD4	6.555	6.267	4.4	6.894	5.2
	CQUADR	6.555	6.267	4.4	6.894	5.2
5	CQUAD4	7.381	7.156	3.0	7.882	6.3
	CQUADR	7.381	7.156	3.0	7.882	6.3
6	CQUAD4	11.402	11.253	1.3	13.164	15.4
	CQUADR	11.402	11.253	1.3	13.164	15.4

Note: Reference value (Ref. Value) refers to the accepted solution to this problem.

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 11a.

5.2.2 Thin Square Cantilevered Plate – Anti-symmetric Modes

Problem Description

Figure 1 shows the thin square cantilevered plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the square plate are determined. All dimensions are in meters.

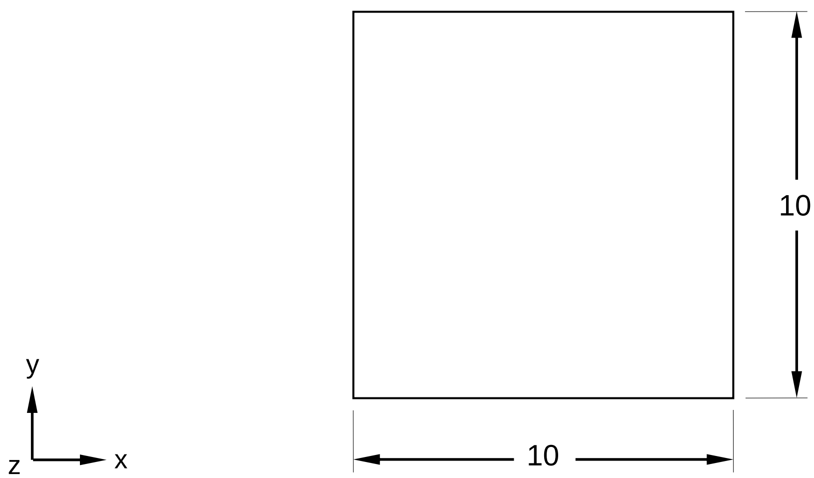


Figure 1. Thin Square Cantilevered Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_2_2c4.nas – 5-DOF/node, CQUAD4, coupled mass formulation
- vm5_2_2d4.nas – 5-DOF/node, CQUAD4, diagonal mass formulation
- vm5_2_2cR.nas – 6-DOF/node, CQUADR, coupled mass formulation
- vm5_2_2dR.nas – 6-DOF/node, CQUADR, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Anti-symmetric modes.
- Test 1 and 2 (vm5_2_2c4 and vm5_2_2d4): 45 nodes, 32 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5_2_2cR and vm5_2_2dR): 45 nodes, 32 6-DOF/node quadrilateral plate elements.
- Only the bottom half of the plate is meshed (10 m x 5 m).
- There are 4 elements vertically, on the 5 m portion of the plate, and 8 elements horizontally, on the 10 m portion of the plate.

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Thickness: t = 0.05 m

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.3

Boundary Conditions

Constraints (all tests): Nodes 1-5 are fully constrained in all translations and rotations. Nodes 6, 11, 16, 21, 26, 31, 36, and 41 are constrained in the X, Y, and Z-translations and Z-rotation. All other nodes are constrained in the X and Y-translations and Z-rotation.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

		NAFEMS	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Element**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	CQUAD4	1.019	1.012	0.7	1.032	1.3
	CQUADR	1.019	1.012	0.7	1.032	1.3
2	CQUAD4	3.839	3.641	5.1	3.843	0.1
	CQUADR	3.839	3.641	5.1	3.843	0.1
3	CQUAD4	8.313	7.199	13.4	8.248	0.8
	CQUADR	8.313	7.199	13.4	8.248	0.8
4	CQUAD4	9.424	8.303	11.9	9.321	1.1
	CQUADR	9.424	8.303	11.9	9.321	1.1
5	CQUAD4	11.728	10.411	11.2	12.275	4.7
	CQUADR	11.728	10.411	11.2	12.275	4.7
6	CQUAD4	17.818	15.097	15.2	18.366	3.1
	CQUADR	17.818	15.097	15.2	18.366	3.1

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 11b.

5.2.3 Free Thin Square Plate

Problem Description

Figure 1 shows the thin square plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the square plate are determined. All dimensions are in meters.

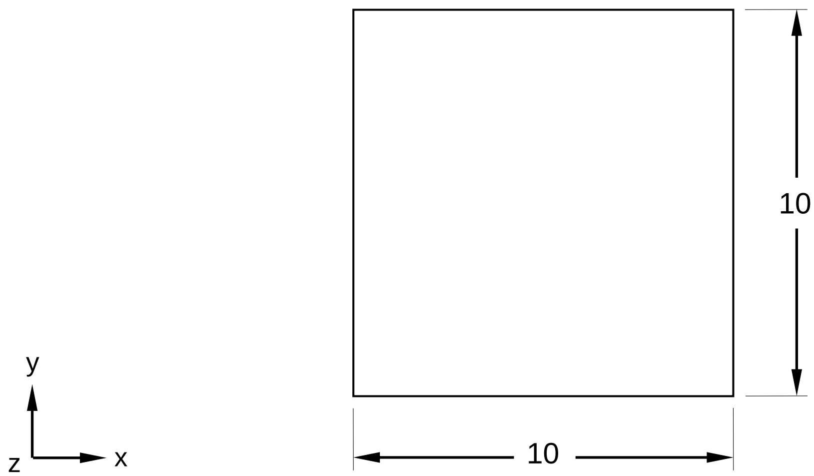


Figure 1. Free Thin Square Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_2_3c4.nas – 5-DOF/node, CQUAD4, coupled mass
- vm5_2_3d4.nas – 5-DOF/node, CQUAD4, diagonal mass
- vm5_2_3cR.nas – 6-DOF/node, CQUADR, coupled mass
- vm5_2_3dR.nas – 6-DOF/node, CQUADR, diagonal mass

Model Data

Finite Element Modeling

- Attributes: Rigid body modes (three modes). Repeated eigenvalues. Use of kinematic DOF for the rigid body mode calculation with the subspace iterative eigensolver.
- Test 1 and 2 (vm5_2_3c4 and vm5_2_3d4): 81 nodes, 64 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5_2_3cR and vm5_2_3dR): 81 nodes, 64 6-DOF/node quadrilateral plate elements.
- The whole plate is modeled with 8 elements on each side.

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Thickness: t = 0.05 m

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.3

Boundary Conditions

Constraint set 1: All nodes are constrained in the X and Y-translations and Z-rotation.

Constraint set 2: Nodes 1 and 3 (lower left and upper right) are fully constrained in all translations and rotations.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

		Ref. Value	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Element**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
4	CQUAD4	1.622	1.587	2.1	1.633	0.8
	CQUADR	1.622	1.587	2.1	1.633	0.8
5	CQUAD4	2.360	2.242	5.0	2.390	1.3
	CQUADR	2.360	2.242	5.0	2.390	1.3
6	CQUAD4	2.922	2.804	4.0	2.978	1.9
	CQUADR	2.922	2.804	4.0	2.978	1.9
7, 8	CQUAD4	4.233	3.980	6.0	4.293	1.4
	CQUADR	4.233	3.980	6.0	4.293	1.4
9	CQUAD4	7.416	6.844	7.7	7.816	5.4
	CQUADR	7.416	6.844	7.7	7.816	5.4
10	CQUAD4	N/A	6.844	N/A	7.816	N/A
	CQUADR	N/A	6.844	N/A	7.816	N/A

Note: Reference value (Ref. Value) refers to the accepted solution to this problem.

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 12.

5.2.4 Simply – Supported Thin Square Plate

Problem Description

Figure 1 shows the simply – supported thin square plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the square plate are determined. All dimensions are in meters.

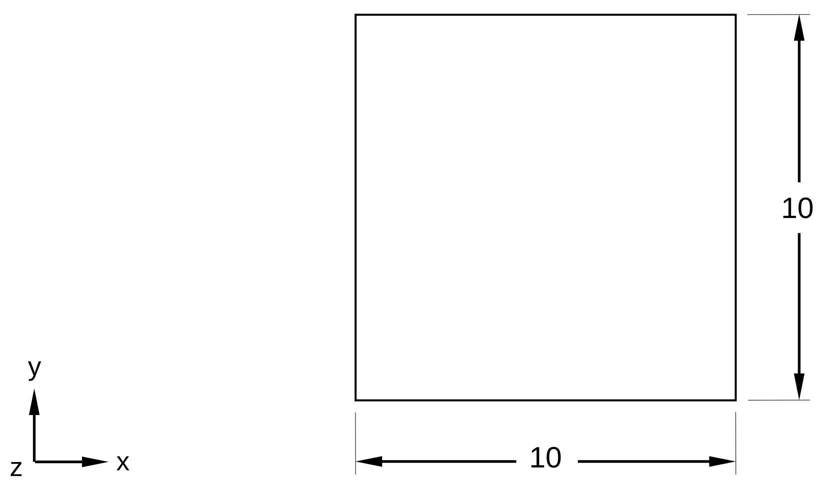


Figure 1. Simply – Supported Thin Square Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_2_4c4.nas – 5-DOF/node, CQUAD4, coupled mass formulation
- vm5_2_4d4.nas – 5-DOF/node, CQUAD4, diagonal mass formulation
- vm5_2_4cR.nas – 6-DOF/node, CQUADR, coupled mass formulation
- vm5_2_4dR.nas – 6-DOF/node, CQUADR, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Well established, repeated eigenvalues.
- Test 1 and 2 (vm5_2_4c4 and vm5_2_4d4): 81 nodes, 64 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5_2_4cR and vm5_2_4dR): 81 nodes, 64 6-DOF/node quadrilateral plate elements.
- The whole plate is modeled with 8 elements on each edge.

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Thickness: t = 0.05 m

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.3

Boundary Conditions

All nodes are constrained in the X and Y-translations, and Z-rotation. All nodes along edges X = 0 and X = 10 m are constrained in the Z-translation and X-rotation, while the nodes along edges Y = 0 and Y = 10 m are constrained in the Z-translation and Y-rotation. A constraint set is created to fully constrain the four corner nodes (9, 13, 41, 68) in all translations and rotations.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

		Ref. Value	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Element**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	CQUAD4	2.377	2.377	0.0	2.377	0.0
	CQUADR	2.377	2.377	0.0	2.377	0.0
2, 3	CQUAD4	5.942	6.001	1.0	6.001	1.0
	CQUADR	5.942	6.001	1.0	6.001	1.0
4	CQUAD4	9.507	9.526	0.2	9.527	0.2
	CQUADR	9.507	9.526	0.2	9.527	0.2
5, 6	CQUAD4	11.884	12.325	3.7	12.327	3.7
	CQUADR	11.884	12.325	3.7	12.327	3.7
7, 8	CQUAD4	15.449	15.756	2.0	15.758	2.0
	CQUADR	15.449	15.756	2.0	15.758	2.0

Note: Reference value (Ref. Value) refers to the accepted solution to this problem.

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 13.

5.2.5 Simply – Supported Thin Annular Plate

Problem Description

Figure 1 shows the simply – supported thin annular plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the annular plate are determined. All dimensions are in meters.

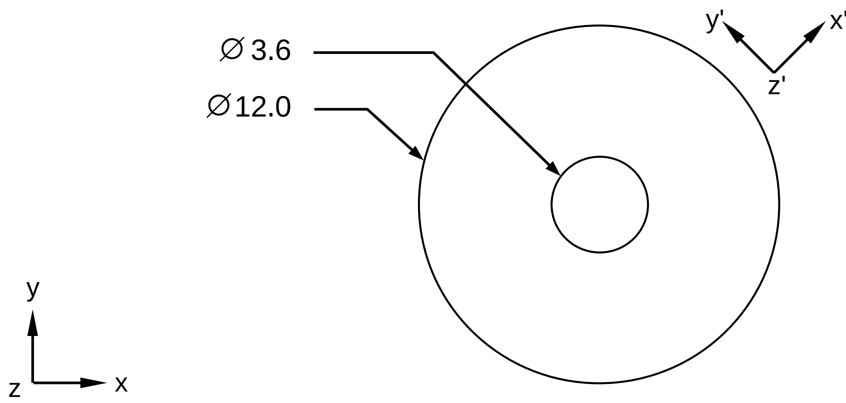


Figure 1. Simply – Supported Thin Annular Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_2_5c4.nas – 5-DOF/node, CQUAD4, coupled mass formulation
- vm5_2_5d4.nas – 5-DOF/node, CQUAD4, diagonal mass formulation
- vm5_2_5cR.nas – 6-DOF/node, CQUADR, coupled mass formulation
- vm5_2_5dR.nas – 6-DOF/node, CQUADR, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Curved boundary (Skewed coordinate system). Repeated eigenvalues.
- Test 1 and 2 (vm5_2_5c4 and vm5_2_5d4): 192 nodes, 160 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5_2_5cR and vm5_2_5dR): 192 nodes, 160 6-DOF/node quadrilateral plate elements.
- The model has a mapped mesh with 32 elements on the inner and outer circles of the plate, respectively.

Units

meter/Newton/second

Model Geometry

Outer Radius: R_o = 6.0 m
Inner Radius: R_i = 1.8 m
Thickness: t = 0.05 m

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.3

Boundary Conditions

Constraint set 1 (All Tests): All nodes are constrained in the R and T-translations, and Z-rotation. All nodes around the model's circumference are additionally constrained in the Z-translation and R-rotation.

Constraint set 2 (All Tests): Nodes 258 and 290 on the outer circumference are constrained in the R and T-translations.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

		Ref. Value	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Element**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	CQUAD4	1.870	1.863	0.4	1.882	0.6
	CQUADR	1.870	1.863	0.4	1.882	0.6
2, 3	CQUAD4	5.137	5.164	0.5	5.238	2.0
	CQUADR	5.137	5.164	0.5	5.238	2.0
4, 5	CQUAD4	9.673	9.796	1.3	10.093	4.3
	CQUADR	9.673	9.796	1.3	10.093	4.3
6	CQUAD4	14.850	14.239	4.1	15.469	4.2
	CQUADR	14.850	14.239	4.1	15.469	4.2
7, 8	CQUAD4	15.573	15.743	1.1	16.606	6.6
	CQUADR	15.573	15.743	1.1	16.606	6.6
9	CQUAD4	18.382	17.765	3.4	19.275	4.9
	CQUADR	18.382	17.765	3.4	19.275	4.9

Note: Reference value (Ref. Value) refers to the accepted solution to this problem.

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 14.

5.2.6 Clamped Thin Rhombic Plate

Problem Description

Figure 1 shows the thin rhombic plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the rhombic plate are determined. All dimensions are in meters.

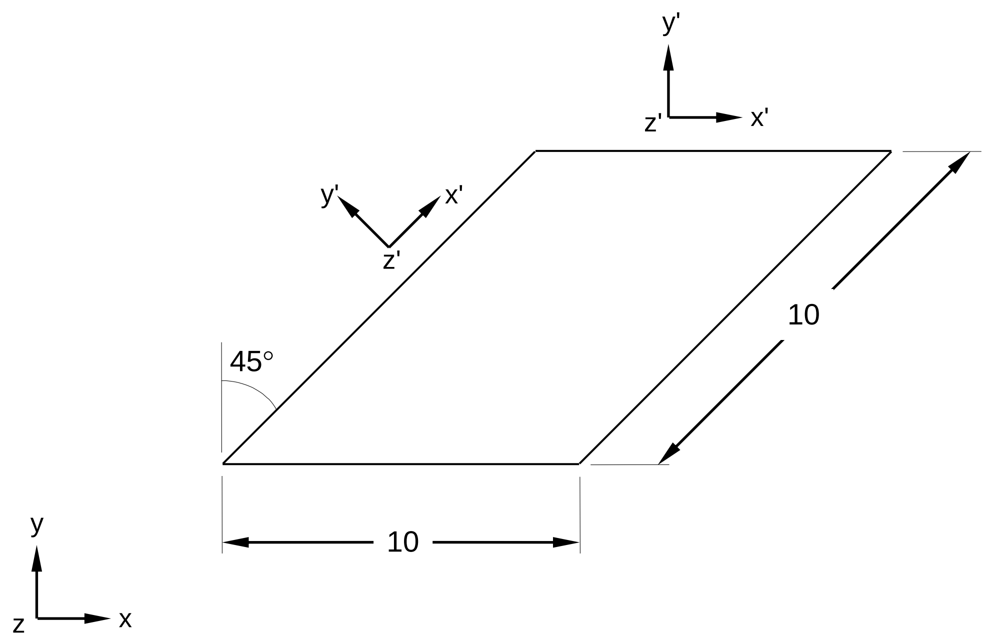


Figure 1. Clamped Thin Rhombic Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_2_6c4.nas – 5-DOF/node, CQUD4, coupled mass formulation
- vm5_2_6d4.nas – 5-DOF/node, CQUD4, diagonal mass formulation
- vm5_2_6cR.nas – 6-DOF/node, CQUDR, coupled mass formulation
- vm5_2_6dR.nas – 6-DOF/node, CQUDR, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Distorted elements.
- Test 1 and 2 (vm5_2_6c4 and vm5_2_6d4): 169 nodes, 144 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5_2_6cR and vm5_2_6dR): 169 nodes, 144 6-DOF/node quadrilateral plate elements.
- The plate is modeled with 12 elements on each edge.

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Thickness: t = 0.05 m

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.3

Boundary Conditions

The nodes along all four edges are fully constrained in all translations and rotations. All other nodes are constrained in the X and Y-translations, and Z-rotation.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

		NAFEMS	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Element**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	CQUAD4	8.142	7.988	1.9	8.130	0.1
	CQUADR	8.142	7.988	1.9	8.130	0.1
2	CQUAD4	13.891	13.088	5.8	13.666	1.6
	CQUADR	13.891	13.088	5.8	13.666	1.6
3	CQUAD4	20.036	18.443	8.0	19.776	1.3
	CQUADR	20.036	18.443	8.0	19.776	1.3
4	CQUAD4	20.165	19.340	4.1	20.071	0.5
	CQUADR	20.165	19.340	4.1	20.071	0.5
5	CQUAD4	27.704	24.901	10.1	27.582	0.4
	CQUADR	27.704	24.901	10.1	27.582	0.4
6	CQUAD4	32.046	28.777	10.2	31.039	3.1
	CQUADR	32.046	28.777	10.2	31.039	3.1

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 15.

5.2.7 Simply – Supported Thick Square Plate, Test A

Problem Description

Figure 1 shows the thick square plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the square plate are determined. All dimensions are in meters.

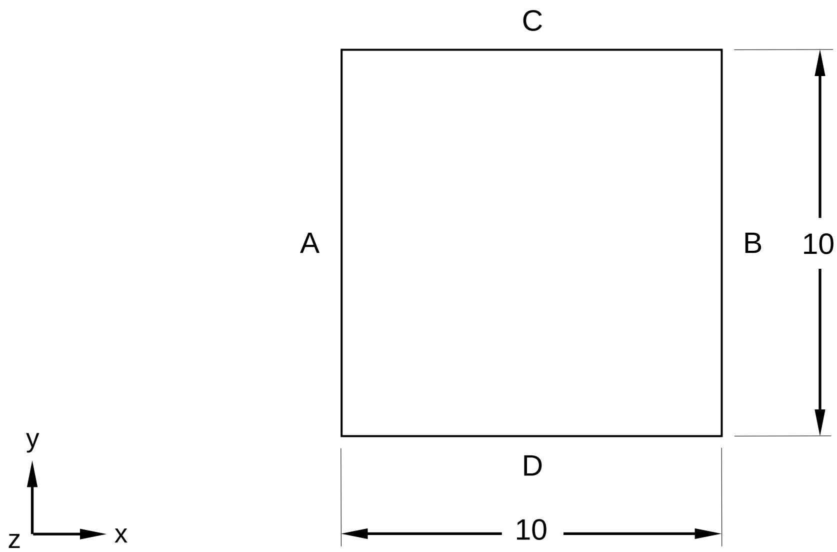


Figure 1. Simply – Supported Thick Square Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_2_7c4.nas – 5-DOF/node, CQUAD4, coupled mass formulation
- vm5_2_7d4.nas – 5-DOF/node, CQUAD4, diagonal mass formulation
- vm5_2_7cR.nas – 6-DOF/node, CQUADR, coupled mass formulation
- vm5_2_7dR.nas – 6-DOF/node, CQUADR, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Well established. Repeated eigenvalues. Effect of secondary restraints.
- Test 1 and 2 (vm5_2_7c4 and vm5_2_7d4): 81 nodes, 64 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5_2_7cR and vm5_2_7dR): 81 nodes, 64 6-DOF/node quadrilateral plate elements.
- The plate is modeled with 8 elements on each edge.

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Thickness: t = 0.05 m

Material Properties

Young's Modulus: $E = 200.0\text{ E+9 N/m}^2$
Mass Density: $\rho = 8000\text{ kg/m}^3$
Poisson's Ratio: $\nu = 0.3$

Boundary Conditions

The corner nodes are fully constrained in all translations and rotations. The nodes along edges A and B (see Figure 1) are constrained in all translations and rotations, except the Y-rotation. The nodes along edges C and D are constrained in all translations and rotations, except the X-rotation. All other nodes are constrained in the X and Y-translations, and Z-rotation.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

		NAFEMS	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Element**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	CQUAD4	46.66	46.21	1.0	47.06	0.8
	CQUADR	46.66	46.21	1.0	47.06	0.8
2, 3	CQUAD4	115.84	111.00	4.2	116.47	0.6
	CQUADR	115.84	111.00	4.2	116.47	0.6
4	CQUAD4	177.53	169.48	4.5	183.63	3.4
	CQUADR	177.53	169.48	4.5	183.63	3.4
5, 6	CQUAD4	233.40	208.10	10.8	230.76	1.1
	CQUADR	233.40	208.10	10.8	230.76	1.1
7, 8	CQUAD4	283.60	256.70	9.5	294.72	3.9
	CQUADR	283.60	256.70	9.5	294.72	3.9
9, 10	CQUAD4	371.11	322.76	13.0	389.14	4.8
	CQUADR	371.11	322.76	13.0	389.14	4.8

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 21a.

5.2.8 Simply – Supported Thick Square Plate, Test B

Problem Description

Figure 1 shows the thick square plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the square plate are determined. All dimensions are in meters.

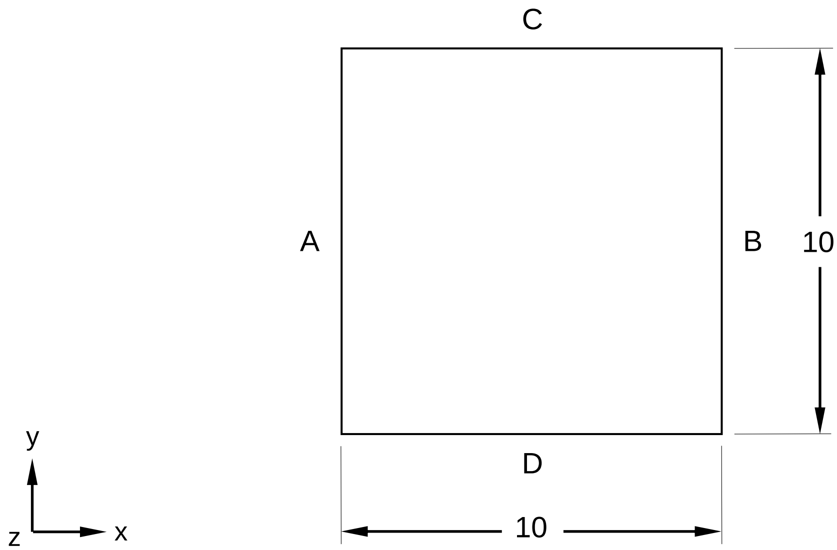


Figure 1. Simply – Supported Thick Square Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_2_8c4.nas – 5-DOF/node, CQUAD4, coupled mass formulation
- vm5_2_8d4.nas – 5-DOF/node, CQUAD4, diagonal mass formulation
- vm5_2_8cR.nas – 6-DOF/node, CQUADR, coupled mass formulation
- vm5_2_8dR.nas – 6-DOF/node, CQUADR, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Well established. Repeated eigenvalues. Effect of secondary restraints.
- Test 1 and 2 (vm5_2_8c4 and vm5_2_8d4): 81 nodes, 64 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5_2_8cR and vm5_2_8dR): 81 nodes, 64 6-DOF/node quadrilateral plate elements.
- The plate is modeled with 8 elements on each edge.

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Thickness: t = 0.05 m

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.3

Boundary Conditions

The corner nodes are fully constrained in all translations and rotations. The nodes along edges A and B are constrained in all translations and rotations, except the Y-rotation (see Figure 1). The nodes along edges C and D are constrained in all translations and rotations, except the X-rotation. All other nodes are constrained in the X and Y-translations, and Z-rotation.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

		NAFEMS	Autodesk Inventor Nastran Diagonal Mass Formulation		Autodesk Inventor Nastran Coupled Mass Formulation	
Mode Number	**Element**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	CQUAD4	44.75	44.36	0.9	45.510	1.7
	CQUADR	44.75	44.36	0.9	45.510	1.7
2, 3	CQUAD4	112.94	106.97	5.3	114.04	0.9
	CQUADR	112.94	106.97	5.3	114.04	0.9
4	CQUAD4	170.28	160.34	5.9	177.56	4.3
	CQUADR	170.28	160.34	5.9	177.56	4.3
5, 6	CQUAD4	230.23	200.62	12.9	227.94	0.8
	CQUADR	230.23	200.62	12.9	227.94	0.8
7, 8	CQUAD4	274.19	242.87	11.4	286.52	4.5
	CQUADR	274.19	242.87	11.4	286.52	4.5
9	CQUAD4	355.98	308.88	13.2	386.48	8.6
	CQUADR	355.98	308.88	13.2	386.48	8.6
10	CQUAD4	355.98	311.69	12.4	386.48	8.6
	CQUADR	355.98	311.69	12.4	386.48	8.6

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 21b.

5.2.9 Clamped Thick Rhombic Plate

Problem Description

Figure 1 shows the thick rhombic plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the rhombic plate are determined. All dimensions are in meters.

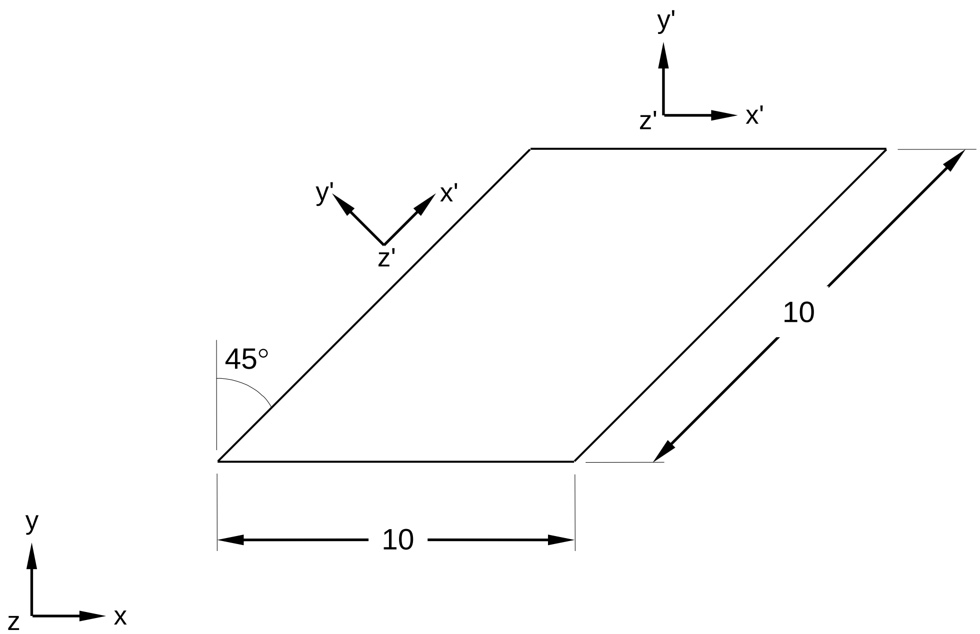


Figure 1. Clamped Thick Rhombic Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_2_9c4.nas – 5-DOF/node, CQUAD4, coupled mass formulation
- vm5_2_9d4.nas – 5-DOF/node, CQUAD4, diagonal mass formulation
- vm5_2_9cR.nas – 6-DOF/node, CQUADR, coupled mass formulation
- vm5_2_9dR.nas – 6-DOF/node, CQUADR, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Distorted elements.
- Test 1 and 2 (vm5_2_9c4 and vm5_2_9d4): 121 nodes, 100 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5_2_9cR and vm5_2_9dR): 121 nodes, 100 6-DOF/node quadrilateral plate elements.
- The plate is modeled with 10 elements on each edge.

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Thickness: t = 0.05 m

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.3

Boundary Conditions

The nodes along all four edges are fully constrained in all translations and rotations. All other nodes are constrained in the X and Y-translations, and Z-rotation.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

		NAFEMS	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Element**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	CQUAD4	137.80	133.46	3.1	136.71	0.8
	CQUADR	137.80	133.46	3.1	136.71	0.8
2	CQUAD4	218.48	202.48	7.3	214.62	1.8
	CQUADR	218.48	202.48	7.3	214.62	1.8
3	CQUAD4	295.42	265.73	10.1	291.70	1.3
	CQUADR	295.42	265.73	10.1	291.70	1.3
4	CQUAD4	296.83	279.28	5.9	293.71	1.0
	CQUADR	296.83	279.28	5.9	293.71	1.0
5	CQUAD4	383.56	331.75	13.5	380.35	0.8
	CQUADR	383.56	331.75	13.5	380.35	0.8
6	CQUAD4	426.59	376.13	11.8	416.71	2.3
	CQUADR	426.59	376.13	11.8	416.71	2.3

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 22.

5.2.10 Simply – Supported Thick Annular Plate

Problem Description

Figure 1 shows the thick annular plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the annular plate are determined. All dimensions are in meters.

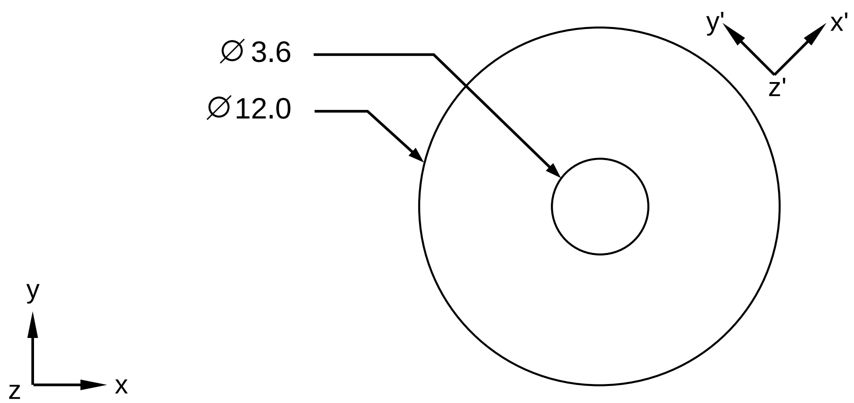


Figure 1. Simply – Supported Thick Annular Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_2_10c4.nas – 5-DOF/node, CQUAD4 elements, coupled mass formulation
- vm5_2_10d4.nas – 5-DOF/node, CQUAD4 elements, diagonal mass formulation
- vm5_2_10cR.nas – 6-DOF/node, CQUADR elements, coupled mass formulation
- vm5_2_10dR.nas – 6-DOF/node, CQUADR elements, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Curved boundary (skewed coordinate system). Repeated eigenvalues.
- Test 1 and 2 (vm5_2_10c4 and vm5_2_10d4): 192 nodes, 160 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5_2_10cR and vm5_2_10dR): 192 nodes, 160 6-DOF/node quadrilateral plate elements.
- The model has a mapped mesh with 32 elements on the inner and outer circles of the plate, respectively.

Units

meter/Newton/second

Model Geometry

Outer Radius: R_o = 6.0 m
Inner Radius: R_i = 1.8 m
Thickness: t = 0.05 m

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.3

Boundary Conditions

The nodes around the circumference are constrained in the X, Y and Z-translations, and X and Z-rotations. All other nodes are constrained in the X and Y-translations, and Z-rotation.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

		NAFEMS	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Element**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	CQUAD4	18.82	18.50	1.7	18.68	0.7
	CQUADR	18.82	18.50	1.7	18.68	0.7
2, 3	CQUAD4	49.82	49.44	0.8	50.13	0.6
	CQUADR	49.82	49.44	0.8	50.13	0.6
4, 5	CQUAD4	96.06	93.46	2.7	96.26	0.2
	CQUADR	96.06	93.46	2.7	96.26	0.2
6	CQUAD4	148.34	134.61	9.3	145.86	1.6
	CQUADR	148.34	134.61	9.3	145.86	1.6
7, 8	CQUAD4	153.68	146.67	4.5	154.71	0.7
	CQUADR	153.68	146.67	4.5	154.71	0.7
9	CQUAD4	174.52	161.03	7.7	174.52	0.0
	CQUADR	174.52	161.03	7.7	174.52	0.0

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 23.

5.2.11 Cantilevered Square Membrane

Problem Description

Figure 1 shows the square membrane. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the square membrane are determined. All dimensions are in meters.

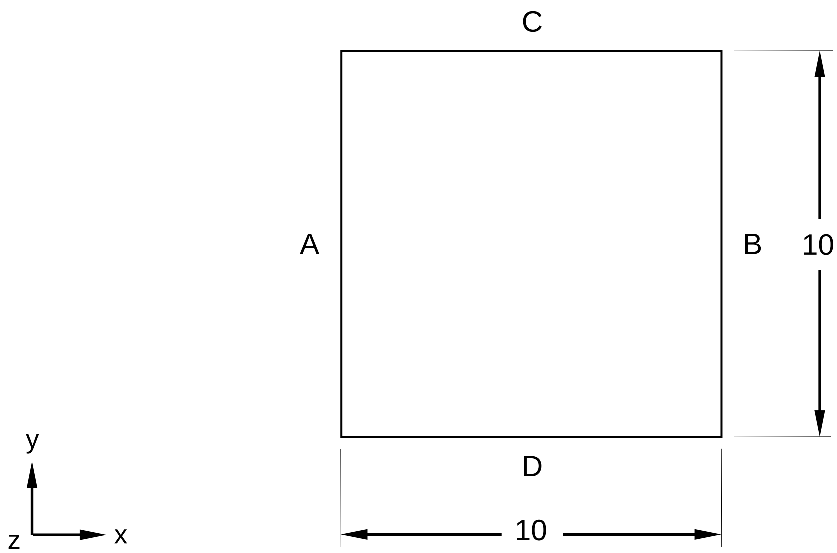


Figure 1. Cantilevered Square Membrane

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_2_11c4.nas – 5-DOF/node, CQUAD4 elements, coupled mass formulation
- vm5_2_11d4.nas – 5-DOF/node, CQUAD4 elements, diagonal mass formulation
- vm5_2_11cR.nas – 6-DOF/node, CQUADR elements, coupled mass formulation
- vm5_2_11dR.nas – 6-DOF/node, CQUADR elements, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Well established.
- Test 1 and 2 (vm5_2_11c4 and vm5_2_11d4): 81 nodes, 64 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5_2_11cR and vm5_2_11dR): 81 nodes, 64 6-DOF/node quadrilateral plate elements.
- The plate is modeled with 8 elements on each edge.

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Thickness: t = 0.05 m

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.3

Boundary Conditions

The nodes along the Y-axis are fully constrained in all translations and rotations (edge A in Figure 1). All other nodes are constrained in the Z-translation, and all rotations.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

		NAFEMS	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Element**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	CQUAD4	52.90	53.31	0.8	53.60	1.3
	CQUADR	52.90	53.36	0.9	53.66	1.4
2	CQUAD4	126.11	125.60	0.4	126.07	0.0
	CQUADR	126.11	125.62	0.4	126.09	0.0
3	CQUAD4	143.20	140.68	1.8	143.98	0.5
	CQUADR	143.20	140.81	1.7	144.11	0.6
4	CQUAD4	228.85	216.41	5.4	228.76	0.0
	CQUADR	228.85	217.00	5.2	229.30	0.2
5	CQUAD4	247.90	240.38	3.0	247.77	0.0
	CQUADR	247.90	240.61	2.9	247.97	0.0
6	CQUAD4	260.61	252.52	3.1	259.98	0.2
	CQUADR	260.61	252.85	3.0	260.33	0.1

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 31.

5.2.12 Cantilevered Tapered Membrane

Problem Description

Figure 1 shows the tapered membrane. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the tapered membrane are determined. All dimensions are in meters.

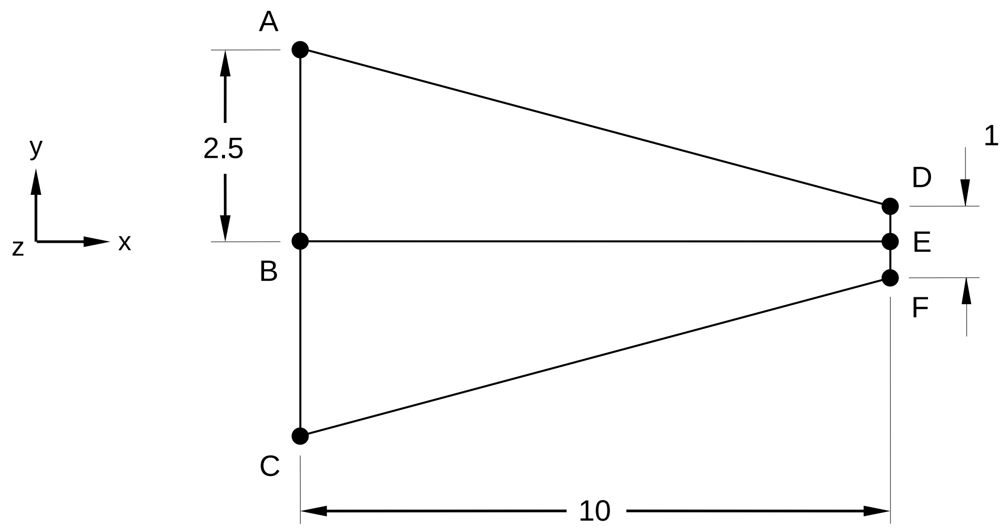


Figure 1. Cantilevered Tapered Membrane

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_2_12c4.nas – 5-DOF/node, CQUAD4 elements, coupled mass formulation
- vm5_2_12d4.nas – 5-DOF/node, CQUAD4 elements, diagonal mass formulation
- vm5_2_12cR.nas – 6-DOF/node, CQUADR elements, coupled mass formulation
- vm5_2_12dR.nas – 6-DOF/node, CQUADR elements, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Shear behavior. Irregular mesh. Symmetry.
- Test 1 and 2 (vm5_2_12c4 and vm5_2_12d4): 153 nodes, 128 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5_2_12cR and vm5_2_12dR): 153 nodes, 128 6-DOF/node quadrilateral plate elements.
- The plate is modeled with 4 elements on edges AB, BC, DE and EF, and 16 elements on edges AD, BE and CF (see Figure 1).

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Width: w_1 = 1 m and w_2 = 5 m
Thickness: t = 0.1 m

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.3

Boundary Conditions

Because the model is symmetric, only half of it is analyzed (defined by the points B, C, E, and F, as shown in Figure 1). The nodes along the Y-axis are fully constrained in all translations and rotations. All other nodes are constrained in all rotations.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

		Ref. Value	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Element**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	CQUAD4	44.62	44.56	0.1	44.65	0.1
	CQUADR	44.62	44.52	0.2	44.64	0.1
2	CQUAD4	130.03	129.68	0.3	131.04	0.8
	CQUADR	130.03	129.27	0.6	130.32	0.2
3	CQUAD4	162.70	162.61	0.1	162.80	0.0
	CQUADR	162.70	162.62	0.1	162.76	0.0
4	CQUAD4	246.05	244.41	0.7	250.35	1.7
	CQUADR	246.05	243.31	1.1	247.24	0.5
5	CQUAD4	379.90	374.87	1.3	391.59	3.1
	CQUADR	379.90	372.85	1.9	383.00	0.8
6	CQUAD4	391.44	389.80	0.4	393.10	0.4
	CQUADR	391.44	389.42	0.5	392.94	0.4

Note: Reference value (Ref. Value) refers to the accepted solution to this problem.

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 32.

5.2.13 Free Annular Membrane

Problem Description

Figure 1 shows the annular membrane. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method. The natural frequencies of the annular membrane are determined. All dimensions are in meters.

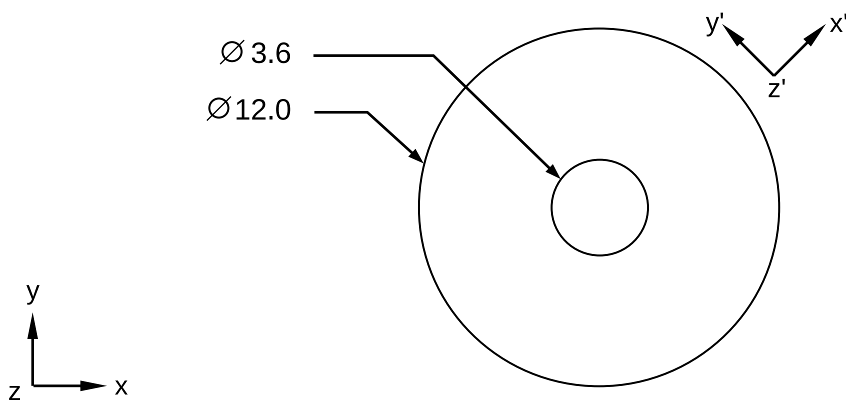


Figure 1. Free Annular Membrane

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_2_13c4.nas – 5-DOF/node, CQUAD4 elements, coupled mass formulation
- vm5_2_13d4.nas – 5-DOF/node, CQUAD4 elements, diagonal mass formulation
- vm5_2_13cR.nas – 6-DOF/node, CQUADR elements, coupled mass formulation
- vm5_2_13dR.nas – 6-DOF/node, CQUADR elements, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Repeated eigenvalues. Rigid body modes (three modes). Kinematically incomplete suppression.
- Test 1 and 2 (vm5_2_13c4 and vm5_2_13d4): 192 nodes, 160 5-DOF/node quadrilateral plate elements.
- Test 3 and 4 (vm5_2_13cR and vm5_2_13dR): 192 nodes, 160 6-DOF/node quadrilateral plate elements.
- The model has a mapped mesh with 32 elements on the inner and outer circles of the plate, respectively.

Units

meter/Newton/second

Model Geometry

Outer Radius: R_o = 6.0 m
Inner Radius: R_i = 1.8 m
Thickness: t = 0.06 m

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.3

Boundary Conditions

Nodes 254 and 286 (points A and B as shown in Figure 1) are constrained in the X and Y-translations. All other nodes are constrained in the Z-translation and all rotations.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

		NAFEMS	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Element**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
4, 5	CQUAD4	129.51	128.85	0.5	129.83	0.2
	CQUADR	129.51	129.05	0.4	130.03	0.4
6	CQUAD4	225.46	224.52	0.4	225.22	0.1
	CQUADR	225.46	224.55	0.4	225.25	0.1
7, 8	CQUAD4	234.92	230.88	1.7	236.16	0.5
	CQUADR	234.92	230.90	1.7	236.17	0.5
9, 10	CQUAD4	272.13	265.21	2.5	272.17	0.0
	CQUADR	272.13	265.69	2.4	272.62	0.2
11, 12	CQUAD4	340.34	329.45	3.2	341.00	0.2
	CQUADR	340.34	329.52	3.2	341.06	0.2
13, 14	CQUAD4	391.98	369.76	5.7	390.95	0.3
	CQUADR	391.98	371.12	5.7	392.16	0.0

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 33.

5.3 Axisymmetric Solid and Solid Element Test Cases

The following normal mode/eigenvalue verification problems using standard NAFEMS Benchmarks are performed using solid elements

5.3.1 Simply – Supported “Solid” Square Plate

Problem Description

Figure 1 shows the simply supported square plate. Normal modes/eigenvalue analysis is performed on the model using the subspace iterative method for both lumped mass and consistent mass. The natural frequencies of the simply supported square plate are determined. All dimensions are in meters.

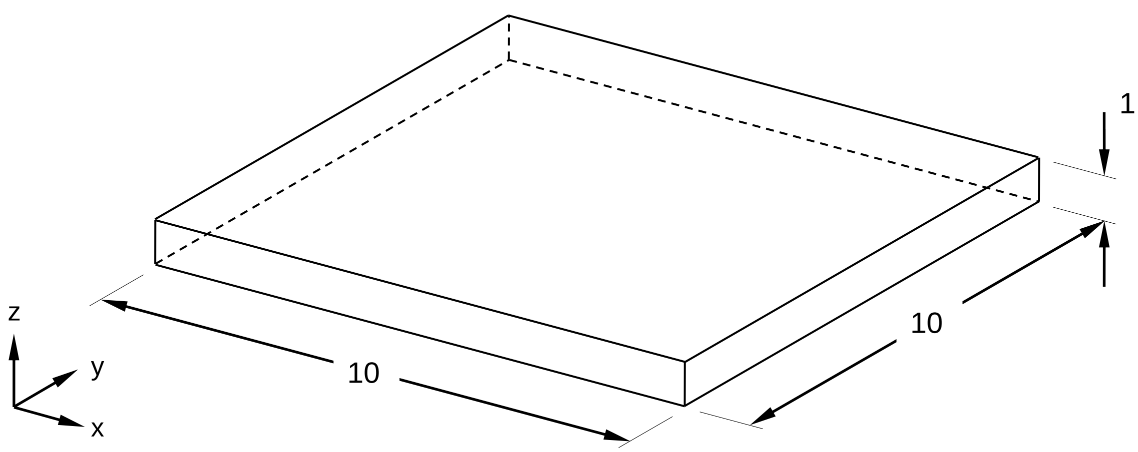


Figure 1. Simply – Supported “Solid” Square Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm5_3_1cl.nas – Linear solid brick elements, coupled mass formulation
- vm5_3_1dl.nas – Linear solid brick elements, diagonal mass formulation
- vm5_3_1cp.nas – Parabolic solid brick elements, coupled mass formulation
- vm5_3_1dp.nas – Parabolic solid brick elements, diagonal mass formulation

Model Data

Finite Element Modeling

- Attributes: Well established. Rigid body modes (three modes). Kinematically incomplete suppressions.
- Tests 1 and 2 (vm5_3_1cl, vm5_3_1dl): 324 nodes, 192 5 Linear solid brick elements.
- Tests 3 and 4 (vm5_3_1cp, vm5_3_1dp): 155 nodes, 16 parabolic solid brick elements.

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Width: w = 10.0 m
Thickness: t = 1 m

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Mass Density: ρ = 8000 kg/m^3
Poisson's Ratio: ν = 0.3

Boundary Conditions

Constraint set 1: All the nodes along the four edges on the XY plane at Z = -0.5 m are constrained in the Z-translation.

Constraint set 2 (Kinematic DOF): Tests 1 and 2: Nodes 36 and 264 are constrained in all translations. Tests 3 and 4: Nodes 27 and 219 are constrained in the X and Y-translations.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

		Ref. Value	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Element**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
4	Linear	45.90	44.11	3.9	45.32	1.3
	Parabolic	45.90	44.47	3.1	44.78	2.4
5, 6	Linear	109.44	106.72	2.5	113.96	4.1
	Parabolic	109.44	107.81	1.5	107.56	1.7
7	Linear	167.89	156.47	6.8	173.30	3.2
	Parabolic	167.89	161.13	4.0	169.90	1.2
8	Linear	193.59	193.58	0.0	196.77	1.6
	Parabolic	193.59	185.54	4.2	192.74	0.4
9, 10	Linear	206.19	200.14	2.9	209.56	1.6
	Parabolic	206.19	192.39	6.7	205.57	0.3

Note: Reference value (Ref. Value) refers to the accepted solution to this problem.

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov., 1987. Test No. 52.

6. Verification Test Cases from the Société Francaise des Mechaniciens

The purpose of these test cases is to verify the functionality of Autodesk Inventor Nastran using standard benchmarks published by SFM (Société Francaise des Mécaniciens, Paris, France) in “Guide de validation des progiciels de calcul de structures”.

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent common and well-known applications of the finite element method.

6.1 Mechanical Structures – Linear Statics Analysis with Bar or Rod Elements

The following mechanical structures verification problems using standard SFM Benchmarks are performed using linear static analysis with bar or rod elements.

6.1.1 Short Beam on Two Articulated Supports

Problem Description

Figure 1 shows the short beam with two fixed supports. Static analysis is performed on the model. The total translation at point B (node 7) of the beam is determined. All dimensions are in millimeters.

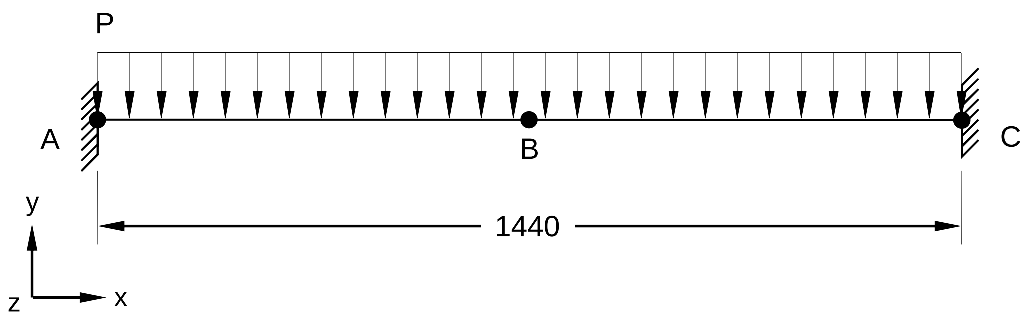


Figure 1. Short Beam on Two Articulated Supports

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_1_1.nas

Model Data

Finite Element Modeling

- 11 nodes, 10 bar elements

Units

meter/Newton/second

Model Geometry

Length: L = 1.440 m

Cross Sectional Properties

Area: A = 31.0 E-4 m^2
Moment of Inertia: I = 2810.0 E-8 m^4
Shear Area Ratio = 2.42

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Poisson's Ratio: ν = 0.3

Boundary Conditions

The two nodes at the ends of the beam (nodes 1 and 2, at points A and C, as shown in Figure 1) are constrained in all translations and rotations except for the Z-rotation. A load P = 1.0 E+5 N/m is applied to nodes 1-10 in the negative Y-direction.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Total Translation at Point B (node 7) (m)	-1.2593E-2	-1.2491E-2	0.8

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLL02/89.

6.1.2 Clamped Beams Linked by a Rigid Element

Problem Description

Figure 1 shows the clamped beams linked by a rigid element. Static analysis is performed on the model. The displacement, constraint moment, and constraint force on the beams are determined. All dimensions are in millimeters.

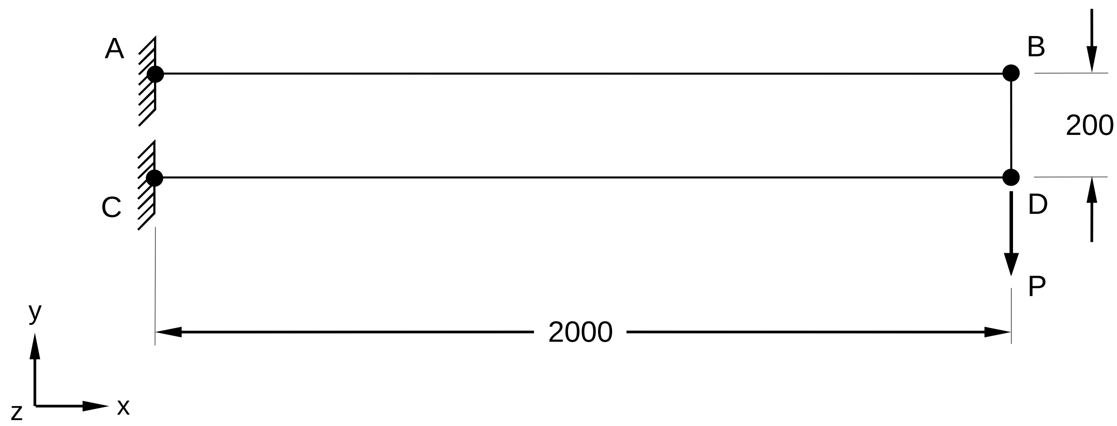


Figure 1. Clamped Beams Linked by a Rigid Element

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_1_2.nas

Model Data

Finite Element Modeling

- 22 nodes, 20 bar elements (10 on each edge, AB and CD, respectively), and 1 rigid element (on edge BD).

Units

meter/Newton/second

Model Geometry

Length: L = 2 m
Width: w = 0.2 m

Cross Sectional Properties

Moment of Inertia: I = 1.33 E-8 m^4

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2

Boundary Conditions

Points A (node 1) and C (node 4) are fully constrained in all translations and rotations. A nodal force P = 1000 N is set to point D (node 3) in the negative Y-direction.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Node**	**Point**	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Displacement Y (T2 Translation) (m)	6	B	-0.125	-0.125	0.0
Displacement Y (T2 Translation) (m)	3	D	-0.125	-0.125	0.0
Force Y (T2 Constraint Force) (N)	1	A	500	500	0.0
Moment R_z (R3 Constraint Force) (Nm)	1	A	500	500	0.0
Force Y (T2 Constraint Force) (N)	4	C	500	500	0.0
Moment R_z (R3 Constraint Force) (Nm)	4	C	500	500	0.0

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLL05/89.

6.1.3 Plane Bending Load on a Thin Arc

Problem Description

Figure 1 shows the model of the thin arc and applied force. Static analysis is performed on the model. The displacements of the thin arc are determined.

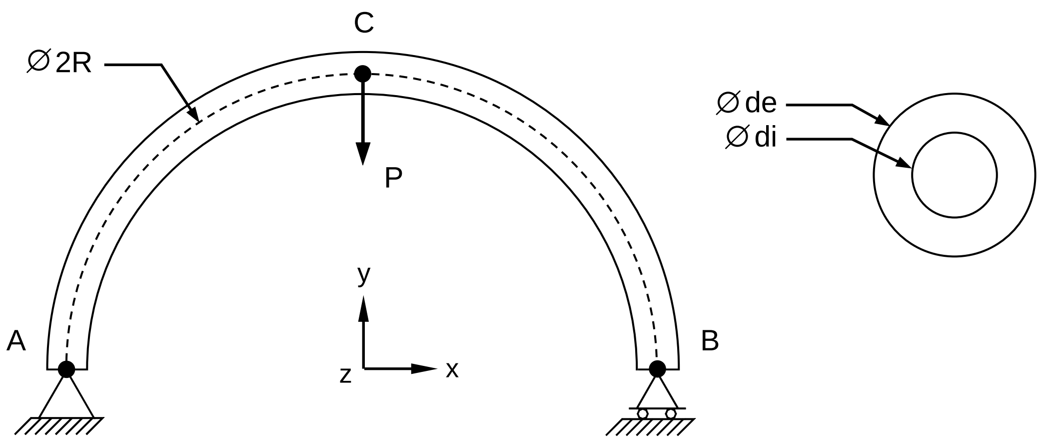


Figure 1. Plane Bending Load on a Thin Arc

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_1_3.nas

Model Data

Finite Element Modeling

- 11 nodes, 10 bar elements

Units

meter/Newton/second

Model Geometry

Radius of Arc: R = 1 m
External Diameter of Bar: d_e = 0.020 m
Inside Diameter of Bar: d_i = 0.016 m

Cross Sectional Properties

Area: A = 1.131 E-4 m^2
Moment of Inertia: I_x = 4.637 E-9 m^4

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Poisson's Ratio: ν = 0.3

Boundary Conditions

Point A (node 2) is constrained in all translations. Point B (node 1) is constrained in the Y and Z-translations. A nodal force P = 100 N is set to point C in the negative Y-direction.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Node**	**Point**	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
R_z (R3 Rotation) (rad)	2	A	-3.0774E-2	-3.1097E-2	1.0
R_z (R3 Rotation) (rad)	1	B	3.0774E-2	3.1097E-2	1.0
Y (T2 Translation) (m)	7	C	-1.9206E-2	-1.9342E-2	0.7
X (T1 Translation) (m)	1	B	5.3913E-2	5.3734E-2	0.3

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLL08/89.

6.1.4 Articulated Plane Truss

Problem Description

Figure 1 shows the articulated plane truss and applied forces. Static analysis is performed on the model. The displacement, and reaction force of the plane truss are determined. All dimensions are in meters.

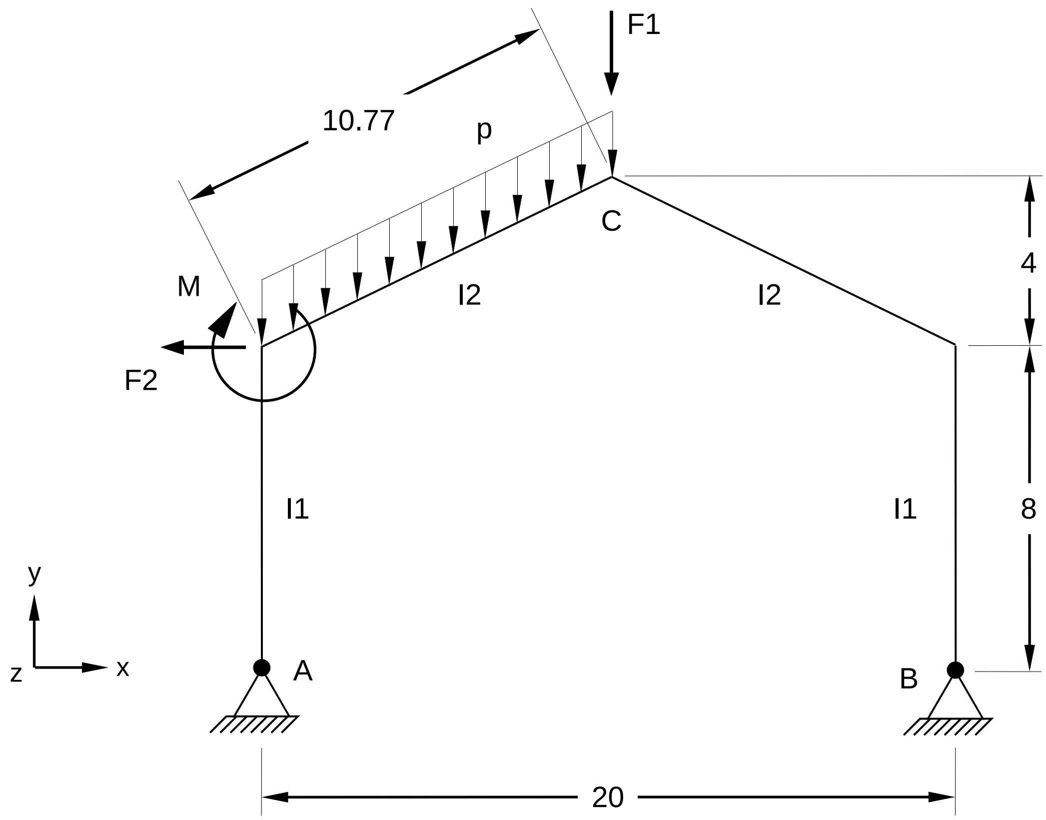


Figure 1. Articulated Plane Truss

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_1_4a.nas – 4 bar elements
- vm6_1_4b.nas – 10 bar elements

Model Data

Finite Element Modeling

- Test 1 (vm6_1_4a): 5 nodes, 4 bar elements (one on each of the four parts of the truss).
- Test 2 (vm6_1_4b): 11 nodes, 10 bar elements (2 on each vertical part of the truss and 3 on each inclined part of the truss).

Units

meter/Newton/second

Model Geometry

Length: L = 20 m
Height: h_1 = 8 m and h_2 = 4 m
Inside Diameter of Bar: d_i = 0.016 m

Cross Sectional Properties

Moment of Inertia: I_1 = 5.0 E-4 m^4
Moment of Inertia: I_2 = 2.5 E-4 m^4

Material Properties

Young's Modulus: E = 210.0 E+9 N/m^2

Boundary Conditions

Test 1 (vm6_1_4a): Points A (node 1) and B (node 4) are constrained in all translations. Nodes 2, 3, and 8 are constrained in the Z-translation. The forces and moments are set to the following numerical values: p = -3,000 N/m (on element 4); F1 = -20,000 N (on node 8); F2 = -10,000 N (on node 2), and M = -100,000 Nm (on node 2).

Test 2 (vm6_1_4b): Points A (node 1) and B (node 4) are constrained in all translations. Nodes 2, 3, 5-13 are constrained in the Z-translation. The forces and moments are set to the following numerical values: p = -3,000 N/m (on element 5-7); F1 = -20,000 N (on node 8); F2 = -10,000 N (on node 2), and M = -100,000 Nm (on node 2).

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	Node	Point	Theory	Test	Autodesk Inventor Nastran	Error (%)
Vertical (Y) reaction (T2 Constraint force) (N)	2	A	31,500	1	33,233	5.5
				2	33,233	5.5
Horizontal (X) reaction (T1 Constraint force) (N)	1	B	20,239	1	20,609	1.8
				2	20,609	1.8
Y (T2 Translation) (m)	7	C	-3.072E-2	1	-3.1609E-2	2.9
				2	-3.1609E-2	2.9

Note: The theoretical values neglect the shear effect.

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLL14/89.

6.2 Mechanical Structures – Linear Statics Analysis with Plate Elements

The following mechanical structures verification problems using standard SFM Benchmarks are performed using linear static analysis with plate elements.

6.2.1 Plane Shear and Bending Load on a Plate

Problem Description

Figure 1 shows the plate and applied plane shear load. Static analysis is performed on the model. The centerline displacement of the plate is determined. All dimensions are in millimeters.

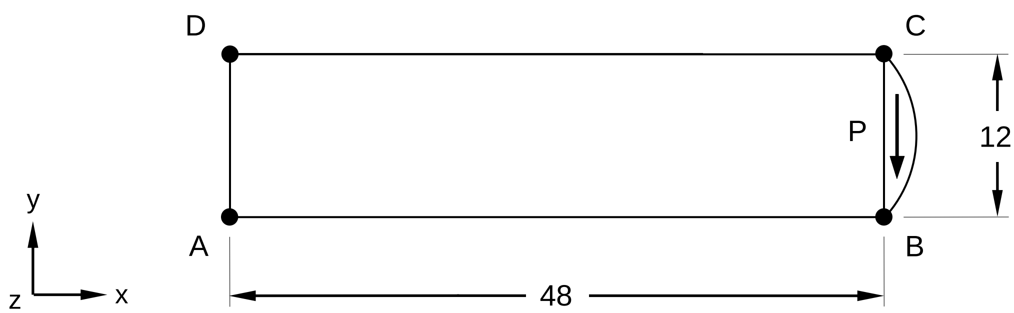


Figure 1. Plane Shear and Bending Load on a Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_2_1.nas

Model Data

Finite Element Modeling

- 126 nodes, 100 5-DOF/node quadrilateral plate elements (5 elements on AD and BC edges, and 20 elements on AB and CD edges, as shown in Figure 1).

Units

meter/Newton/second

Model Geometry

Length: L = 0.048 m
Width: w = 0.012 m
Thickness: t = 0.001 m

Material Properties

Young's Modulus: E = 30.0 E+9 N/m^2
Poisson's Ratio: ν = 0.25

Boundary Conditions

Nodes 121-126 (on the AD edge) are fully constrained in all translations and rotations. A shear force with parabolic distribution on the width and constant distribution on the thickness is set on the BC edge. The resultant force (P) is 40 N.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Point Coordinates**	**Node**	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Y centerline (T2 Translation) (mm)	(L,y)	3	0.3413	0.3408	0.15

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLP01/89.

6.2.2 Uniformly Distributed Load on a Circular Plate

Problem Description

Figure 1 shows the circular plate (¼ of it). Static analysis is performed on the model. The translation displacement for both test cases is determined. All dimensions are in meters.

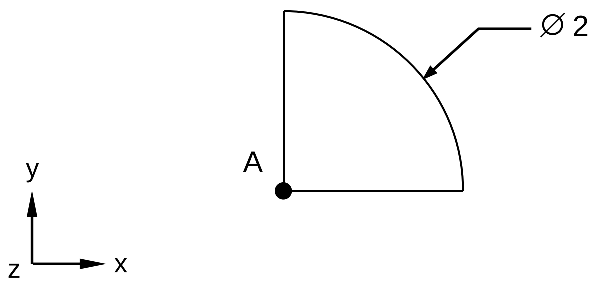


Figure 1. Uniformly Distributed Load on a Circular Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_2_2a.nas – 5-DOF/node quadrilateral elements
- vm6_2_2b.nas – 5-DOF/node triangular elements

Model Data

Finite Element Modeling

- Test 1 (vm6_2_2a): Free meshing 166 nodes, 144 5-DOF/node quadrilateral plate elements.
- Test 2 (vm6_2_2b): Free meshing 149 nodes, 254 5-DOF/node triangular plate elements.
- Only ¼ of the plate is meshed.

Units

meter/Newton/second

Model Geometry

Radius: R = 1 m
Thickness: t = 0.005 m

Material Properties

Young's Modulus: E = 210.0 E+9 N/m^2
Poisson's Ratio: ν = 0.3

Boundary Conditions

Node 1 (at point A) is constrained in all translations and rotations except the Z-translation. The nodes along the arc are fully constrained in all translations and rotations. The nodes along the X and Y-axis are constrained using symmetry boundary conditions. A uniform elemental pressure of p = -1,000 Pa is applied.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Test**	**Node**	**Point**	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
T3 Translation (Displacement Z) (m)	1	1	Center 0	-6.5E-3	-4.2E-3	35.4
T3 Translation (Displacement Z) (m)	2	1	Center 0	-6.5E-3	-5.8E-3	10.8

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLS03/89.

6.2.3 Torque Loading on a Square Tube

Problem Description

Figure 1 shows the square tube (0.1 by 0.1 m, and 5 mm thickness) with torque loading. Static analysis is performed on the model. The displacement and stress on the square tube is determined. All dimensions are in meters.

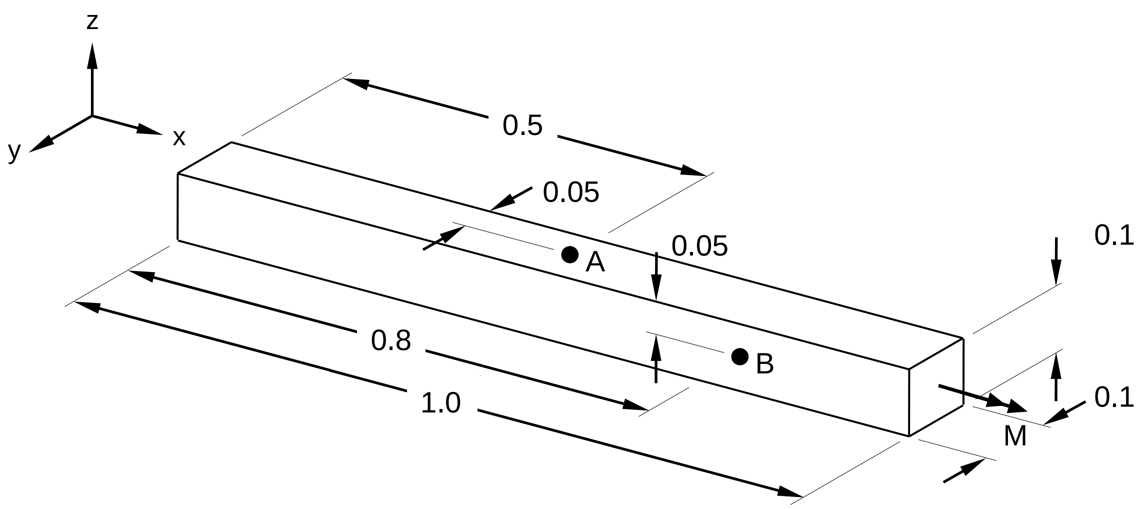


Figure 1. Torque Loading on a Square Tube

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_2_3.nas

Model Data

Finite Element Modeling

- Mapped meshing 176 nodes, 160 5-DOF/node quadrilateral plate elements (4 elements on the small edges and 10 elements on the long edges of the tube).

Units

meter/Newton/second

Model Geometry

Length: L = 1.0 m
Width: w = 0.1 m
Height: h = 0.1 m
Thickness: t = 0.005 m

Material Properties

Young's Modulus: E = 210.0 E+9 N/m^2
Poisson's Ratio: ν = 0.3

Boundary Conditions

Nodes 1-5, 112-115, and 167-169 (on the left end of the tube) are constrained in all translations and rotations. A torque M = 10 N-m is applied to the free end of the tube (on the right hand side of the tube).

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Node**	**Point**	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
T2 Translation (m)	193	A	-6.2E-7	-6.2E-7	0.0
R1 Rotation (rad)	193	A	1.2E+5	1.2E+5	0.0
Plate Bottom Minor Stress (Pa)	193	A	-110,000	-109,369	5.7
T2 Translation (m)	208	B	-9.9E-7	-9.9E-7	0.0
R1 Rotation (rad)	208	B	2.0E+5	2.0E+5	0.0
Plate Bottom Minor Stress (Pa)	208	B	-110,000	-108,558	13.1

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLS05/89.

6.2.4 Cylindrical Shell with Internal Pressure

Problem Description

Figure 1 shows the model of the cylindrical shell (1/8 model, with thickness = 20 mm). Static analysis is performed. The displacements and the stresses of the cylindrical shell with an internal pressure are determined. All dimensions are in meters.

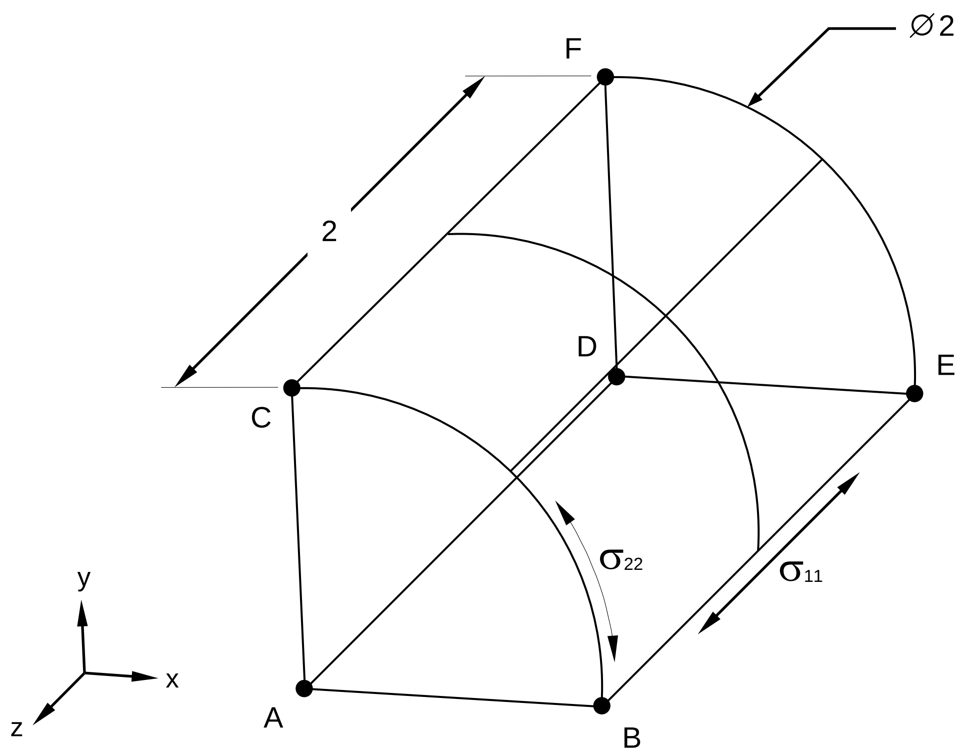


Figure 1. Cylindrical Shell with Internal Pressure

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_2_4a.nas – 6-DOF/node quadrilateral elements
- vm6_2_4b.nas – 6-DOF/node quadrilateral elements

Model Data

Finite Element Modeling

- Test 1 (vm6_2_4a): 121 nodes, 100 6-DOF/node quadrilateral plate elements (10 elements on each of the following edges: BC, EF, BE and CF).

- Test 2 (vm6_2_4b): 441 nodes, 400 6-DOF/node quadrilateral plate elements (20 elements on each of the following edges: BC, EF, BE and CF).

Units

meter/Newton/second

Model Geometry

Length: L = 2.0 m
Radius: R = 1.0 m
Thickness: t = 0.02 m

Material Properties

Young's Modulus: E = 210.0 E+9 N/m^2
Poisson's Ratio: ν =0.3

Boundary Conditions

The model is constrained using symmetry boundary conditions. One end of the model is constrained as a cap covering the end of the cylinder. The opposite side (edge BC) is constrained in X and Y-rotations because the CQUADR element needs these additional boundary conditions due to the hidden mid-side nodes. An internal elemental pressure of 10,000 Pa is applied to the model (from inside out).

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Test**	**Node**	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
σ 11 Plate Top Y Normal Stress (Pa)	1	11	0.0	1.48	NA
	2	21	0.0	0.63	NA
σ 22 Plate Top X Normal Stress (Pa)	1	111	500,000	498,464	0.3
	2	421	500,000	499,617	0.1
∆ R T1 Translation (m)	1	121	2.38E-6	2.37E-6	0.4
	2	441	2.38E-6	2.38E-6	0.0
∆ R T3 Translation (m)	1	121	-1.43E-6	-1.42E-6	0.7
	2	441	-1.43E-6	-1.43E-6	0.0

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLS06/89.

6.2.5 Gravity Loading on a Thin Wall Cylinder

Problem Description

Figure 1 shows the model of the thin wall cylinder (¼ of the model, with thickness = 20 mm). Static analysis is performed. The displacements and stresses of the thin wall cylinder with gravity loading are determined. All dimensions are in meters.

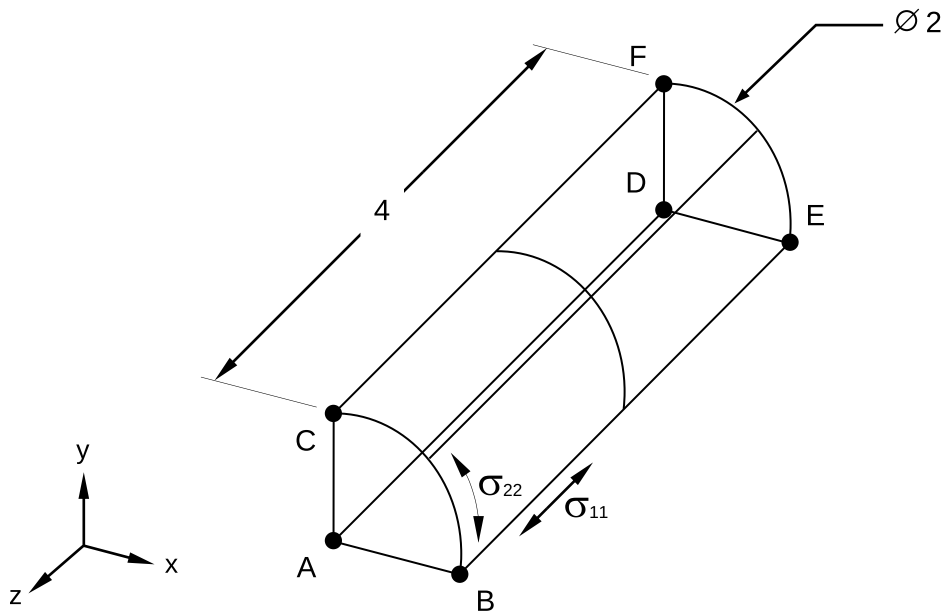


Figure 1. Gravity loading on a Thin Wall Cylinder

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_2_5.nas

Model Data

Finite Element Modeling

- Mapped Meshing: 84 nodes, 65 5-DOF/node quadrilateral plate elements.

Units

meter/Newton/second

Model Geometry

Length: L = 4.0 m
Radius: R = 1.0 m
Thickness: t = 0.02 m

Material Properties

Young's Modulus: E = 210.0 E+9 N/m^2
Poison's Ratio: ν = 0.3
Specific Weight: γ = 7.85 E+4 N/m^3
Mass Density: ρ = 8002.0 kg/m^3

Boundary Conditions

Symmetry is applied to the two radial edges (BE and CF). The top edge (EF) is constrained in the Z-translation only. A translational acceleration body load of 9.8 m/s^2 is applied in the negative Z-direction.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Node**	**Point**	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
σ11 Plate Top X Normal Stress (Pa)	2	X=0	314,000	302,227	3.7
σ22 Plate Top Y Normal Stress (Pa)	1	Any	0.0	0.0	0.0
ΔR T1 Translation (m)	2	X=0	-4.49E-7	-4.39E-7	2.0
Δz T3 Translation (m)	1	X=L	2.99E-6	2.99E-6	0.0
Ψ R2 Rotation (rad)	10	X-L	-1.12E-7	-1.12E-7	0.0

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique,1990. Test No. SSLS09/89.

6.2.6 Pinched Cylindrical Shell

Problem Description

Figure 1 shows the cylindrical shell (1/4 of the model, with thickness = 0.094 mm). Static analysis is performed. The displacements of the cylindrical shell are determined. All dimensions are in meters.

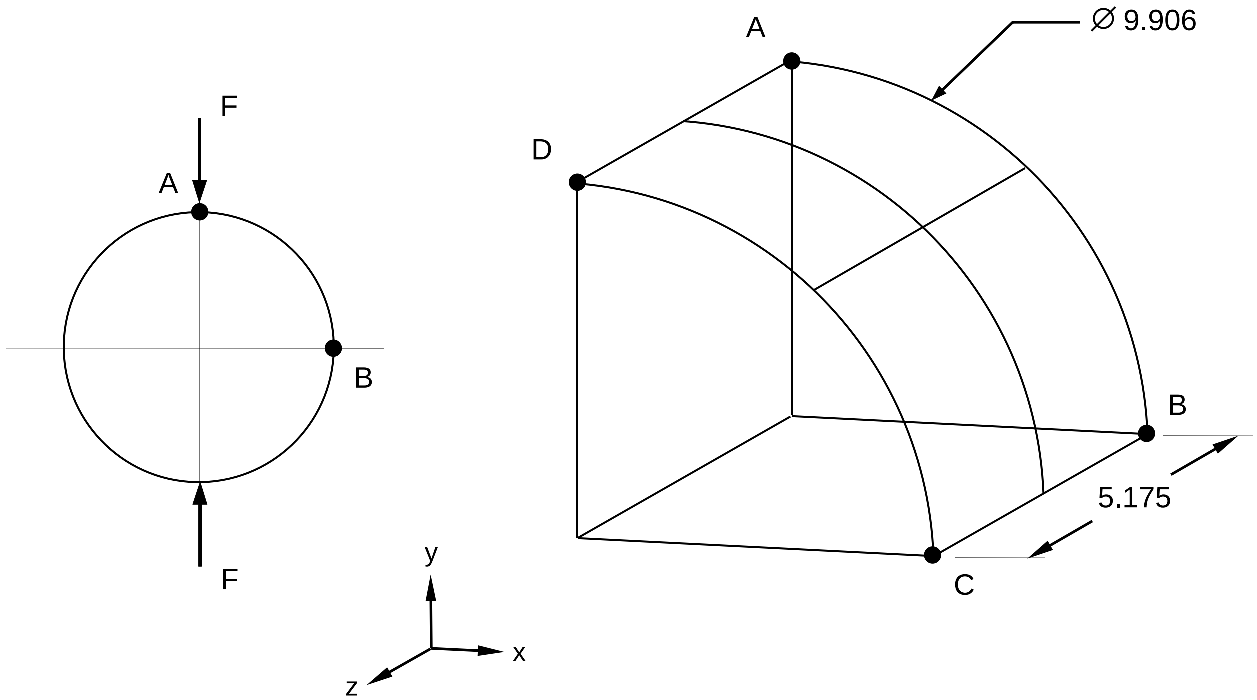


Figure 1. Pinched Cylindrical Shell

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_2_6a.nas – 5-DOF/node triangular plate elements
- vm6_2_6b.nas – 5-DOF/node quadrilateral plate elements

Model Data

Finite Element Modeling

- Test 1 (vm6_2_6a): Free meshing. 173 nodes, 296 5-DOF/node triangular plate elements.
- Test 2 (vm6_2_6b): Mapped meshing. 165 nodes, 140 5-DOF/node quadrilateral plate elements.

Units

meter/Newton/second

Model Geometry

Length: L = 5.175 m
Radius: R = 4.953 m
Thickness: t = 0.094 m

Material Properties

Young's Modulus: E = 10.5 E+6 N/m^2
Poisson's Ratio: ν = 0.3125

Boundary Conditions

Symmetry is applied about the XY, XZ and YZ-planes. Point C is fully constrained in all translations and rotations, except the X-translation. Point B is constrained in the Y-translation, and X and Z-rotations. Point D is fully constrained in all translations and rotations except the Y-translation. Point A is constrained in the X-translation, and Y and Z-rotations. Edge CD is constrained in the Z-translation, and X and Y-rotations. Nodal forces F = 25 N are applied at point D (in the negative Y-direction).

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Test**	**Point**	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Displacement Y (Node 3) (T2 Translation) (m)	1	D	-0.1139	-0.1067	6.3
Displacement Y (Node 3) (T2 Translation) (m)	2	D	-0.1139	-0.1126	1.1

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLS20/89.

6.2.7 Infinite Plate with a Circular Hole

Problem Description

Figure 1 shows ¼ of the plate model, with thickness = 0.001 m. Static analysis is performed. The plate top normal stress at points A, B and C are determined. All dimensions are in meters.

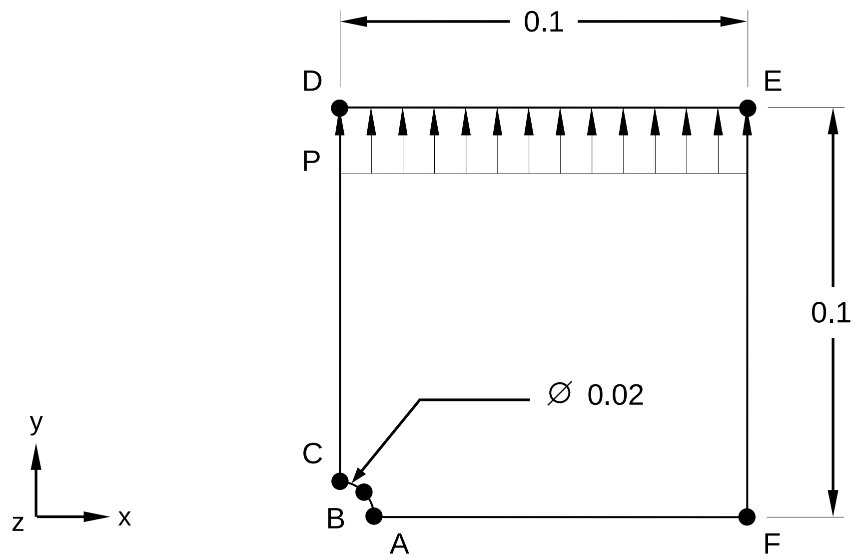


Figure 1. Infinite Plate with a Circular Hole

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_2_7

Model Data

Finite Element Modeling

- A ¼ model is created using 121 nodes, and 100 5-DOF/node quadrilateral plate elements.

Units

meter/Newton/second

Model Geometry

Length: L = 0.1 m
Radius: R = 0.01 m
Thickness: t = 0.001 m

Material Properties

Young's Modulus: E = 3.0 E+10 Pa
Poisson's Ratio: ν = 0.25

Boundary Conditions

The model is constrained using symmetry boundary conditions (on edges AF and CD, as shown in Figure 1). Edge AF is constrained in the Y and Z-translations, and X and Z-rotations. Edge CD is constrained in the X and Z-translations, and Y and Z-rotations. All other nodes are constrained in the Z-translation only. An edge force P = 2500 N/m in the negative Y-direction is applied on edge DE. This represents a uniform pressure on the edge of 2.5 N/mm^2.

Solution Type

Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Top Normal Stress at Point A (MPa)	7.5	7.55	0.7
Top Normal Stress at Point B (MPa)	2.5	2.52	0.8
Top Normal Stress at Point C (MPa)	-2.5	-2.45	2.0

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLS02/89.

6.3 Mechanical Structures – Linear Statics Analysis with Solid Elements

The following mechanical structures verification problems using standard SFM Benchmarks are performed using linear static analysis with solid elements.

6.3.1 Thick Plate Clamped at Edges

Problem Description

Figure 1 shows the thick plate (1/4 model is used with different thickness). Static analysis is performed. The translation at point C of the thick plate is determined. All dimensions are in meters.

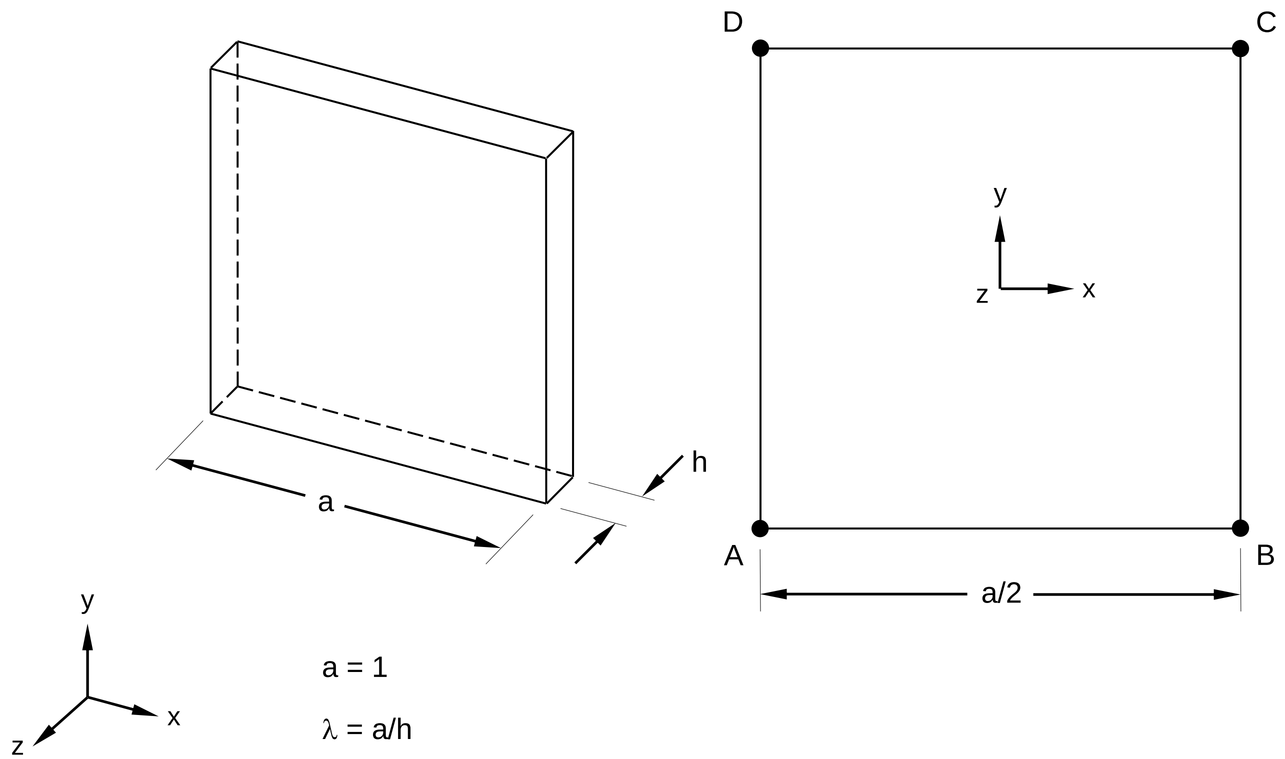


Figure 1. Thick Plate Clamped at Edges

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_3_1a10.nas – Test 1, 3-DOF/node brick, λ = 10
- vm6_3_1a20.nas – Test 1, 3-DOF/node brick, λ = 20
- vm6_3_1a50.nas – Test 1, 3-DOF/node brick, λ = 50
- vm6_3_1a75.nas – Test 1, 3-DOF/node brick, λ = 75
- vm6_3_1a100.nas – Test 1, 3-DOF/node brick, λ = 100
- vm6_3_1b10.nas – Test 2, 5-DOF/node plate, λ = 10
- vm6_3_1b20.nas – Test 2, 5-DOF/node plate, λ = 20
- vm6_3_1b50.nas – Test 2, 5-DOF/node plate, λ = 50
- vm6_3_1b75.nas – Test 2, 5-DOF/node plate, λ = 75
- vm6_3_1b100.nas – Test 2, 5-DOF/node plate, λ = 100

Model Data

Finite Element Modeling

- Test 1: Mapped meshing. 228 nodes, 25 3-DOF/node brick elements. $\lambda = 10$, $\lambda = 20$, $\lambda = 50$, $\lambda = 75$, $\lambda = 100$.
- Test 2: Mapped meshing. 36 nodes, 25 5-DOF/node quadrilateral plate elements. $\lambda = 10$ & t=0.1, $\lambda = 20$ & t = 0.05, $\lambda = 50$ & t = 0.02, $\lambda = 75$ & t = 0.013, $\lambda = 100$ & t = 0.01.

Units

meter/Newton/second

Material Properties

Young's Modulus: E = 2.1 E+11 N/m^2
Poisson's Ratio: ν = 0.3

Boundary Conditions

Constraints Test 1: The nodes on edges AB, A'B', AD, and A'D' are constrained in all translations and rotations. The nodes on edge BC and B'C' are constrained in the X-translation, and Y and Z-rotations. The corner nodes at C and C' are constrained in all translations and rotations, except for the Z-translation. The nodes on edge DC and D'C' are constrained in the Y-translation, and X and Z-rotations.

Constraints Test 2: The nodes on edges AB and AD are fully constrained in all translations and rotations. The nodes on edge BC are constrained in the X-translation, and Y and Z-rotations. The corner nodes at C are constrained in all translations and rotations except for the Z-translation. The nodes on edge DC are constrained in the Y-translation, and X and Z-rotations.

Load case 1: An elemental pressure p = 1E+6 Pa is applied in the negative Z-direction.

Load case 2: A nodal force is applied at point C of F = 1E+6 N in the negative Z-direction.

Solution Type

Static

Comparison of Results

The tabular results are given in Tables 1 and 2.

Table 1. Test Case 1 Results (T3 Translation at location C) (m)

Element	**λ***	**Load Case**	**Node**	**SFDM**	**Autodesk Inventor Nastran**	**Error** (%)
CHEXA	10	Pressure	242	-7.623E-5	-7.443E-5	2.4
CHEXA	10	Force	242	-4.2995E-4	-4.3582E-4	1.4
CHEXA	20	Pressure	242	-5.3833E-4	-5.3189E-4	1.2
CHEXA	20	Force	242	-2.5352E-3	-2.4781E-3	2.3
CHEXA	50	Pressure	242	-8.0286E-3	-7.9292E-3	1.2
CHEXA	50	Force	242	-3.5738E-2	-3.4992E-2	2.1
CHEXA	75	Pressure	242	-2.6861E-2	-2.6443E-2	1.6
CHEXA	75	Force	242	-1.1837E-1	-1.1551E-1	2.4
CHEXA	100	Pressure	36	-6.3389E-2	-6.2405E-2	1.6
CHEXA	100	Force	36	-2.7794E-1	-2.7118E-1	2.4

***Note:** λ = length/thickness

Table 2. Test Case 2 Results (T3 Translation at location C) (m)

Filename	**λ***	**Load Case**	**Node**	**SFDM**	**Autodesk Inventor Nastran**	**Error** (%)
CQUAD4	10	Pressure	1	-7.8661E-5	-7.8029E-5	0.8
CQUAD4	10	Force	1	-4.1087E-4	-3.8270E-4	6.9
CQUAD4	20	Pressure	36	-5.5574E-4	-5.4897E-4	1.2
CQUAD4	20	Force	36	-2.5946E-3	-2.4829E-3	4.3
CQUAD4	50	Pressure	36	-8.3480E-3	-8.2171E-3	1.6
CQUAD4	50	Force	36	-3.7454E-2	-3.6097E-2	3.6
CQUAD4	75	Pressure	36	-2.8053E-2	-2.7586E-2	1.7
CQUAD4	75	Force	36	-1.2525E-1	-1.2064E-1	3.7
CQUAD4	100	Pressure	1	-6.6390E-2	-6.5143E-2	1.9
CQUAD4	100	Force	1	-2.9579E-1	-2.8424E-1	3.9

***Note:** λ = length/thickness

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SSLV09/89.

6.4 Mechanical Structures – Normal Modes/Eigenvalue Analysis

The following mechanical structures verification problems using standard SFM Benchmarks are performed using normal modes/eigenvalue analysis.

6.4.1 Cantilever Beam with a Variable Rectangular Section

Problem Description

Figure 1 shows the cantilever beam with a variable rectangular section. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the beam are determined. All dimensions are in meters.

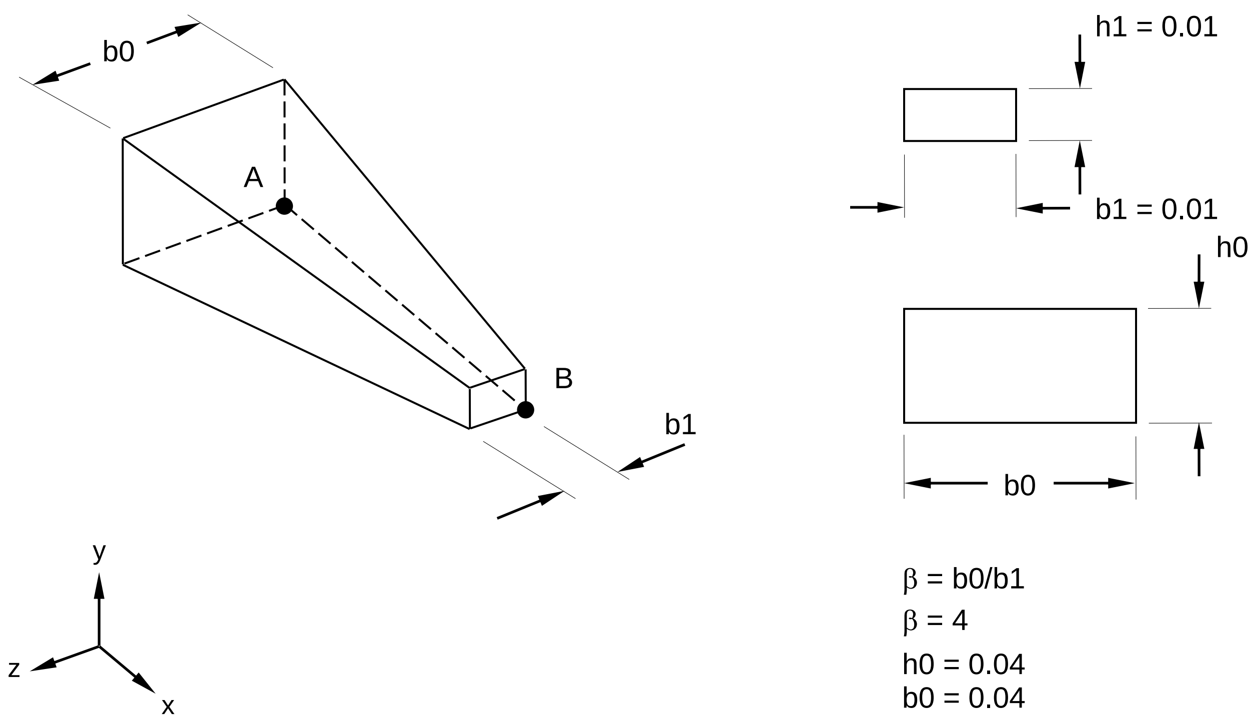


Figure 1. Cantilever Beam with a Variable Rectangular Section

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_4_1d.nas – Diagonal mass formulation
- vm6_4_1c.nas – Coupled mass formulation

Model Data

Finite Element Modeling

- 11 nodes, 10 beam elements (tapered)

Units

meter/Newton/second

Material Properties

Young's Modulus: E = 2.0 E+11 Pa
Mass Density: ρ = 7800 kg/m^3

Boundary Conditions

Point A (node 1) is fully constrained in all translations and rotations. All other nodes are constrained in the Z-translation and X and Y-rotations only.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

		NAFEMS	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**β**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	4	54.18	54.33	0.3	54.15	0.1
2	4	171.94	170.71	0.7	172.02	0.2
3	4	384.40	373.30	0.3	383.92	0.1
4	4	697.24	657.59	5.7	693.82	0.5
5	4	1,112.28	1,017.22	8.5	1,102.04	0.9

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SDLL09/89.

6.4.2 Thin Circular Ring Clamped at Two Points

Problem Description

Figure 1 shows the circular ring. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the circular ring are determined. All dimensions are in meters.

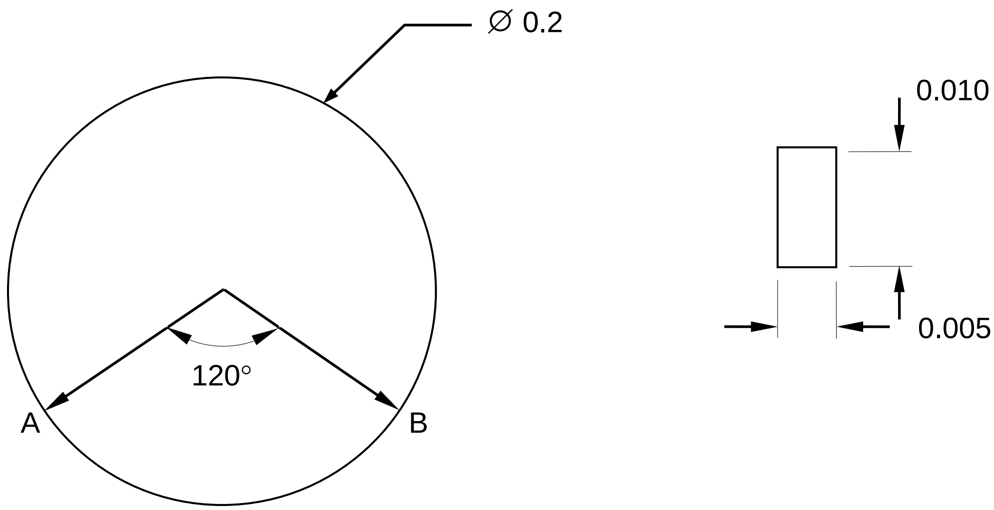


Figure 1. Thin Circular Ring Clamped at Two Points

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_4_2d.nas – Diagonal mass formulation
- vm6_4_2c.nas – Coupled mass formulation

Model Data

Finite Element Modeling

- 29 nodes, 29 bar elements

Units

meter/Newton/second

Model Geometry

Radius: R = 0.1 m

Cross Sectional Properties

Rectangular Cross Section = (0.010 m x 0.005 m)

Material Properties

Young's Modulus: E = 7.2 E+10 Pa
Poison's Ratio: ν = 0.3
Mass Density: ρ = 2700 kg/m^3

Boundary Conditions

Points A and B (nodes 1 and 2) are fully constrained in all translations and rotations. All other nodes are constrained the Z-translation and X and Y-rotations only.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Frequency Results

	Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	235.3	235.9	0.3	236.0	0.3
2	575.3	576.5	0.2	576.1	0.1
3	1,105.7	1,106.1	0.0	1,105.6	0.0
4	1,405.6	1,406.8	0.0	1,405.2	0.0
5	1,751.1	1,751.4	0.0	1,747.5	0.2
6	2,557.0	2,556.9	0.0	2,550.0	0.3
7	2,801.5	2,751.0	1.8	2,774.2	1.0

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SDLL12/89.

6.4.3 Vibration Modes of a Thin Pipe Elbow

Problem Description

Figure 1 shows the thin pipe elbow. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the thin pipe elbow are determined. All dimensions are in meters.

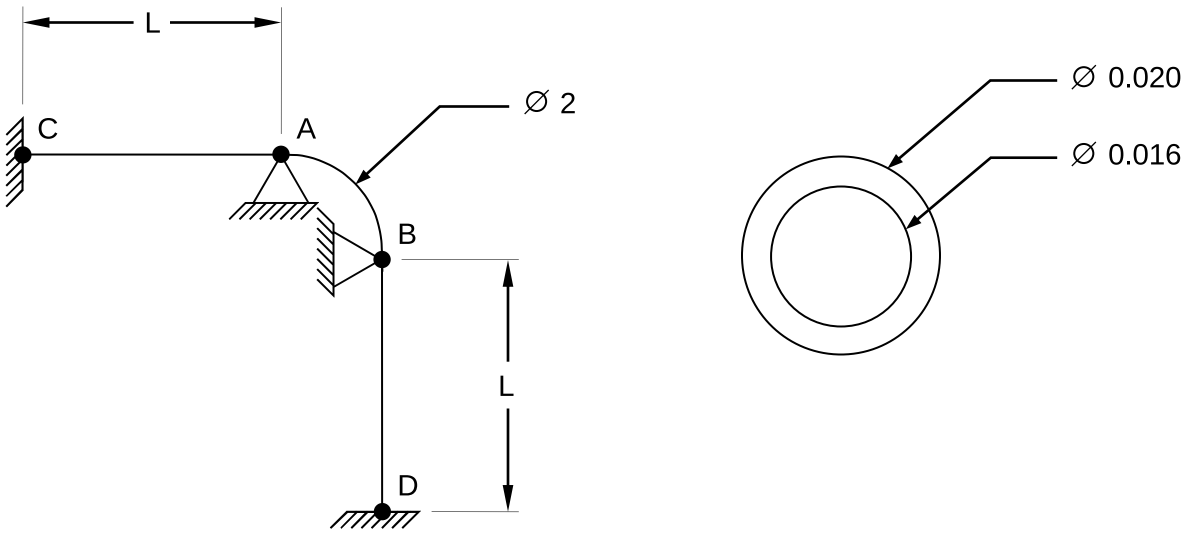


Figure 1. Thin Pipe Elbow

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_4_3ad.nas – Diagonal mass formulation
- vm6_4_3ac.nas – Coupled mass formulation
- vm6_4_3bd.nas – Diagonal mass formulation
- vm6_4_3bc.nas – Coupled mass formulation
- vm6_4_3cd.nas – Diagonal mass formulation
- vm6_4_3cc.nas – Coupled mass formulation

Model Data

Finite Element Modeling

- Test 1 (vm6_4_3a): L = 0.0 m, 19 nodes, 18 bar elements.
- Test 2 (vm6_4_3b): L = 0.6 m, 19 nodes, 18 bar elements.
- Test 3 (vm6_4_3c): L = 2.0 m, 29 nodes, 28 bar elements.

Units

meter/Newton/second

Model Geometry

Radius of Arc: $R = 1$ m
External Radius of Pipe: $R_e = 0.010$ m
Inside Radius of Pipe: $R_i = 0.008$ m

Material Properties

Young's Modulus: $E = 2.1$ E+11 Pa
Poisson's Ratio: $\nu = 0.3$
Mass Density: $\rho = 7800$ kg/m^3

Boundary Conditions

Test 1 (vm6_4_3a): Points C and D (nodes 1 and 2) are fully constrained in all translations and rotations.

Test 2 (vm6_4_3b): Points C and D (nodes 1 and 4) are fully constrained in all translations and rotations. Point B (node 2) is constrained in the X and Z-translations. Point C (node 3) is constrained in the Y and Z-translations.

Test 3 (vm6_4_3c): Points C and D (nodes 1 and 4) are fully constrained in all translations and rotations. Point B (node 2) is constrained in the X and Z-translations. Point C (node 3) is constrained in the Y and Z-translations.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Tables 1, 2 and 3.

Table 1. Test 1 (vm643a) Frequency Results

		NAFEMS	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**L**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	0	44	44	0.0	44	0.0
2	0	119	119	0.0	119	0.0
3	0	125	125	0.0	125	0.0
4	0	227	224	1.3	225	0.9

Table 2. Test 2 (vm643b) Frequency Results

		NAFEMS	Autodesk Inventor Nastran Diagonal Mass Formulation		Autodesk Inventor Nastran Coupled Mass Formulation	
Mode Number	**L**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	1	33	33	0.0	33	0.0
2	1	94	94	0.0	94	0.0
3	1	100	98	2.0	99	1.0
4	1	180	182	1.1	183	1.7

Table 3. Test 3 (vm643c) Frequency Results

		NAFEMS	Autodesk Inventor Nastran Diagonal Mass Formulation		Autodesk Inventor Nastran Coupled Mass Formulation	
Mode Number	**L**	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	2	17.9	17.6	1.7	17.7	1.1
2	2	24.8	24.4	1.6	24.4	1.6
3	2	25.3	24.9	1.6	24.9	1.6
4	2	27.0	26.6	1.5	26.7	1.1

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SDLL14/89.

6.4.4 Thin Square Plate (Clamped or Free)

Problem Description

Figure 1 shows the square plate. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies are determined.

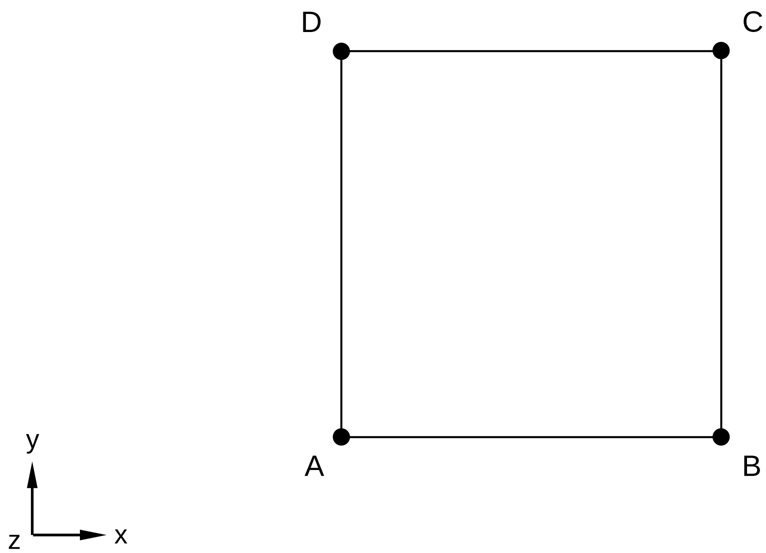


Figure 1. Thin Square Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_4_4ad.nas – Diagonal mass formulation
- vm6_4_4ac.nas – Coupled mass formulation
- vm6_4_4bd.nas – Diagonal mass formulation
- vm6_4_4bc.nas – Coupled mass formulation

Model Data

Finite Element Modeling

- 121 nodes, 100 5-DOF/node quadrilateral plate elements

Units

meter/Newton/second

Model Geometry

Length: L = 1 m
Width: w = 1 m

Material Properties

Young's Modulus: E = 2.1 E+11 Pa
Poisson's Ratio: ν = 0.3
Mass Density: ρ = 7800 kg/m^3

Boundary Conditions

Test 1 (vm6_4_4a): The nodes along side AB are constrained in all translations and rotations.

Test 2 (vm6_4_4b): Unconstrained plate; only the lower left, lower right, and upper right corner nodes are constrained in all translations and rotations (points A, B and C).

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Tables 1 and 2.

Table 1. Test 1 (vm6_4_4a) Frequency Results

	Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	8.73	8.64	1.0	8.68	0.6
2	21.30	21.04	1.2	21.30	0.0
3	53.55	52.25	2.4	53.78	0.4
4	68.30	66.15	3.1	68.86	0.8
5	77.74	75.82	2.5	78.50	1.0
6	136.05	131.05	3.7	139.28	2.4

Table 2. Test 2 (vm6_4_4b) Frequency Results

	Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
7	33.71	33.10	1.8	33.72	0.0
8	49.45	47.34	4.6	49.35	0.2
9	61.05	59.03	3.3	61.38	0.5
10, 11	87.52	83.84	4.2	88.01	0.6

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique,1990. Test No. SDLS01/89.

6.4.5 Simply – Supported Rectangular Plate

Problem Description

Figure 1 shows the rectangular plate, with a thickness of 0.01 meters. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the rectangular plate are determined. All dimensions are in meters.

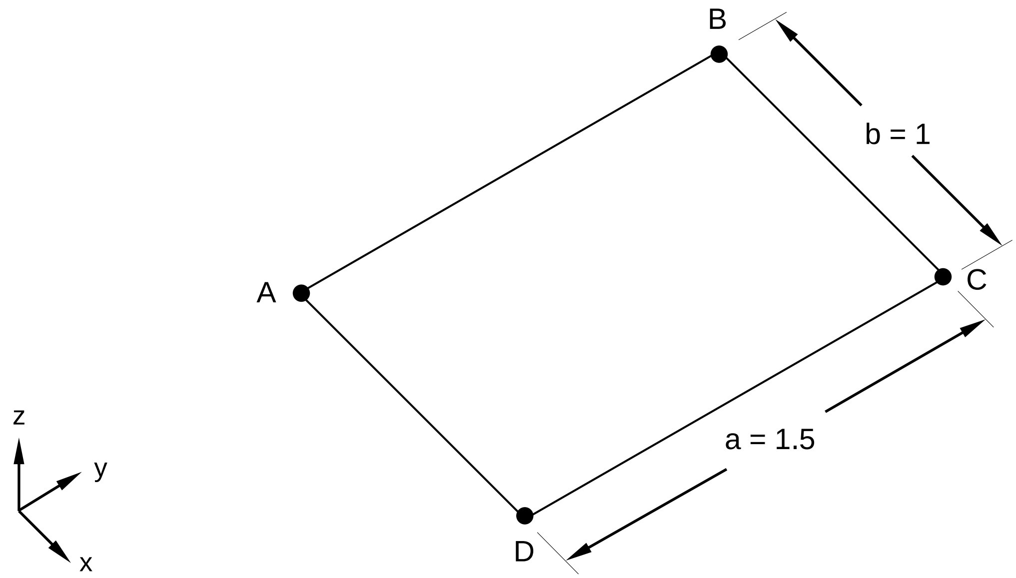


Figure 1. Simply – Supported Rectangular Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_4_5d.nas – Diagonal mass formulation
- vm6_4_5c.nas – Coupled mass formulation

Model Data

Finite Element Modeling

- 176 nodes, 150 5-DOF/node quadrilateral plate elements

Units

meter/Newton/second

Model Geometry

Length: L = 1.5 m
Width: w = 1.0 m

Material Properties

Young's Modulus: E = 2.1 E+11 Pa
Poisson's Ratio: ν = 0.3
Mass Density: ρ = 7800 kg/m^3

Boundary Conditions

The nodes on all sides of the plate are constrained in the Z-translation. Nodes 47, 55 and 119 are constrained in all translations and rotations except for the Z-translation.

Solution Type

Normal Modes/Eigenvalue – Lanczos method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Frequency Results

	Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
4	35.63	35.57	0.2	35.99	1.0
5	68.51	68.39	0.2	69.97	2.1
6	109.62	109.88	0.3	114.00	4.0
7	123.32	123.48	0.1	128.64	4.3
8	142.51	142.38	0.1	149.27	4.7

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SDLS03/89.

6.4.6 Bending of a Symmetric Truss

Problem Description

Figure 1 shows the symmetric truss. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the truss are determined. All dimensions are in meters.

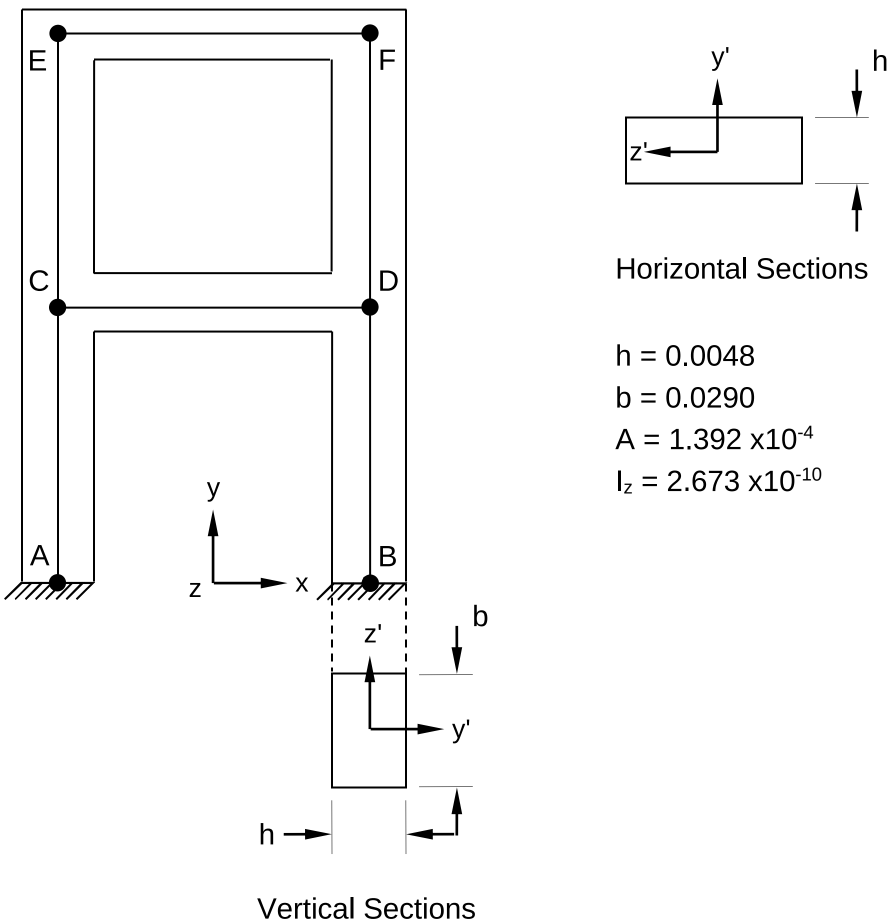


Figure 1. Symmetric Truss

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_4_6d.nas – Diagonal mass formulation
- vm6_4_6c.nas – Coupled mass formulation

Model Data

Finite Element Modeling

- 24 nodes, 24 bar elements

Units

meter/Newton/second

Material Properties

Young's Modulus: E = 2.1 E+11 Pa
Poisson's Ratio: ν = 0.3
Mass Density: ρ = 7800 kg/m^3

Boundary Conditions

Points A and B (nodes 1 and 4) are fully constrained in all translations and rotations. Nodes 2–3 and 5–24 are constrained in the Z-translation and X and Y-rotations.

Solution Type

Normal Modes/Eigenvalue – Subspace iterative method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Frequency Results

	Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	8.8	8.8	0.0	8.8	0.0
2	29.4	29.3	0.3	29.4	0.0
3	43.8	43.8	0.0	43.8	0.0
4	56.3	56.2	0.2	56.3	0.0
5	96.2	95.4	0.2	96.2	0.0
6	102.6	102.4	0.2	102.7	0.1
7	147.1	146.1	0.7	147.3	0.1
8	174.8	173.0	1.0	175.3	0.3
9	178.8	177.3	0.8	179.2	0.2
10	206.0	202.7	1.6	206.7	0.3
11	266.4	262.3	1.6	267.9	0.6
12	320.0	309.6	3.3	322.3	0.7
13	335.0	321.8	4.0	338.5	1.0

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SDLX01/89.

6.4.7 Hovgaard's Problem – Pipes with Flexible Elbows

Problem Description

Figure 1 shows the pipe. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the pipe are determined. All dimensions are in meters.

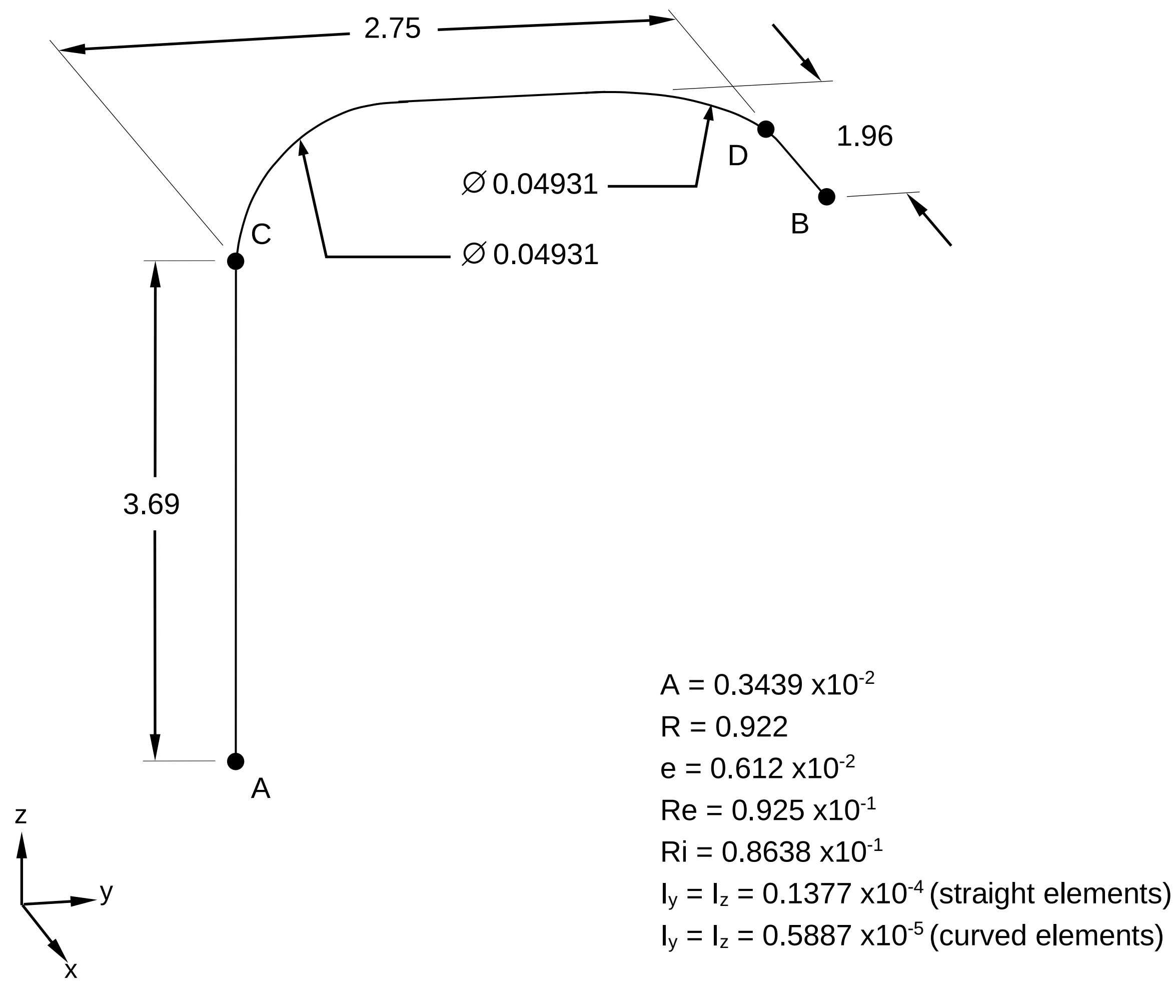


Figure 1. Pipe with Flexible Elbows

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_4_7d.nas – Diagonal mass formulation
- vm6_4_7c.nas – Coupled mass formulation

Model Data

Finite Element Modeling

- 26 nodes, 25 bar elements

Units

meter/Newton/second

Material Properties

Young's Modulus: E = 1.658 E+11 Pa
Poisson's Ratio: ν = 0.3
Mass Density: ρ = 13404.106 kg/m^3

Boundary Conditions

Points A and B (nodes 1 and 6) are fully constrained in all translations and rotations.

Solution Type

Normal Modes/Eigenvalue – Lanczos method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Frequency Results

	Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
1	10.18	10.52	3.3	10.39	3.4
2	19.54	20.03	2.5	19.87	2.5
3	25.47	25.56	0.4	25.35	0.4
4	48.09	48.46	0.8	47.82	0.5
5	52.86	52.21	1.2	51.86	1.3
6	75.94	84.01	10.6	83.16	10.1
7	80.11	86.27	7.7	85.28	7.6
8	122.34	127.60	4.2	126.15	3.5
9	123.15	129.60	5.2	127.97	4.9

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SDLX02/89.

6.4.8 Rectangular Plates

Problem Description

Figure 1 shows the rectangular plates. Normal modes/eigenvalue analysis is performed using the subspace iterative method. The natural frequencies of the rectangular plates are determined. All dimensions are in meters.

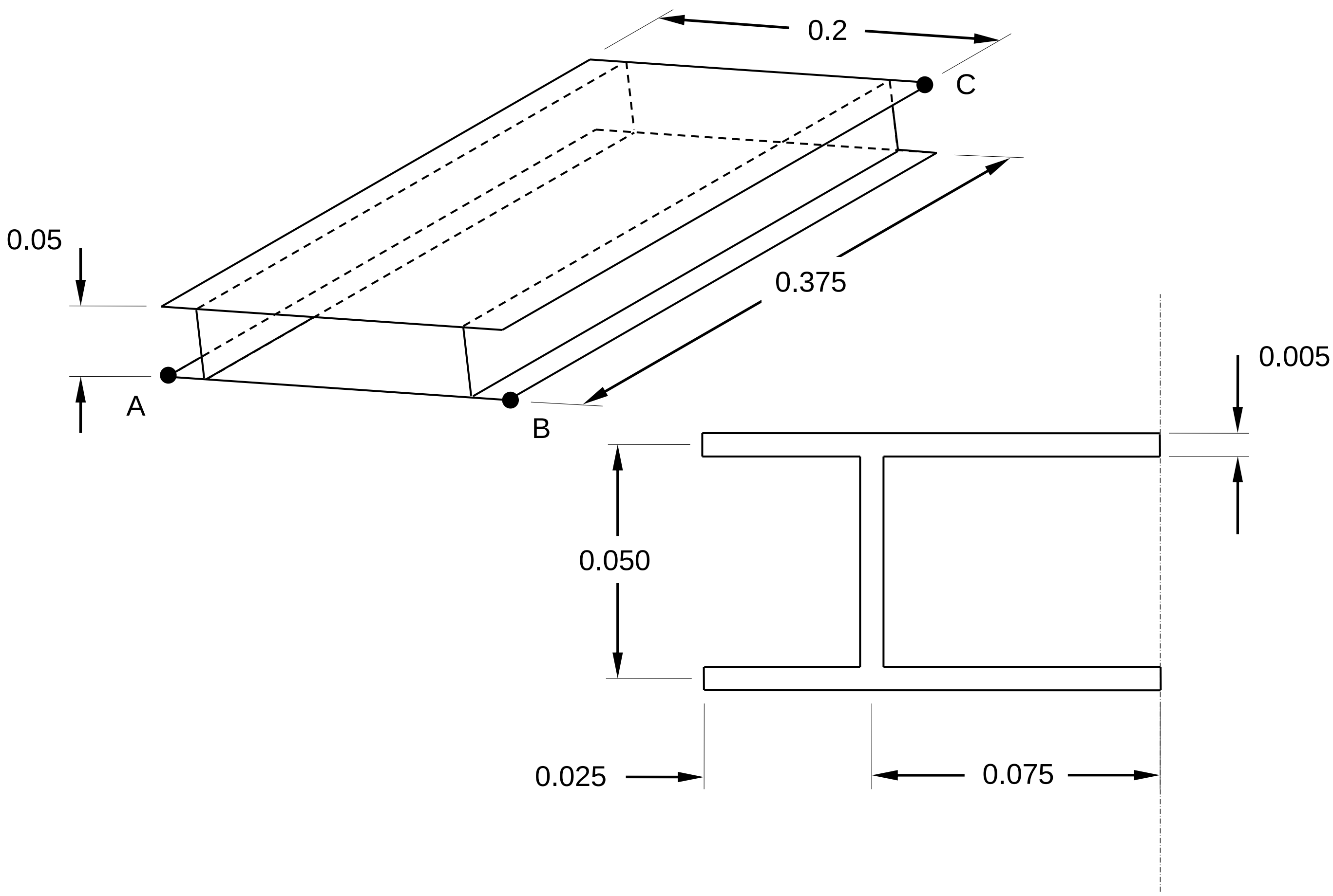


Figure 1. Rectangular Plates

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_4_8d.nas – Diagonal mass formulation
- vm6_4_8c.nas – Coupled mass formulation

Model Data

Finite Element Modeling

- 320 nodes, 300 5-DOF/node quadrilateral plate elements

Units

meter/Newton/second

Model Geometry

Length: L = 0.375 m
Width: w = 0.200 m
Thickness: t = 0.050 m

Material Properties

Young's Modulus: E = 2.1 E+11 Pa
Poisson's Ratio: ν = 0.3
Mass Density: ρ = 7800 kg/m^3

Boundary Conditions

Constraint Set 1: Nodes 2, 69, and 84 (points A, B, C) are fully constrained in all translations and rotations.

Solution Type

Normal Modes/Eigenvalue – Lanczos method

Comparison of Results

The tabular results are given in Table 1.

Table 1. Frequency Results

	Theory	**Autodesk Inventor Nastran Diagonal Mass Formulation**		**Autodesk Inventor Nastran Coupled Mass Formulation**	
Mode Number	**Natural Frequency** (Hz)	**Natural Frequency** (Hz)	**Error** (%)	**Natural Frequency** (Hz)	**Error** (%)
7	584	584	0.0	594	1.7
8	826	818	1.0	844	2.2
9	855	847	0.9	873	2.1
10	911	901	1.2	932	2.3
11	1,113	1,087	2.5	1,137	2.1
12	1,136	1,146	0.8	1,188	4.6

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. SDLX03/89.

6.5 Stationary Thermal Tests – Steady-State Heat Transfer Analysis

The following stationary thermal verification problems using standard SFM Benchmarks are performed using steady-state heat transfer analysis.

6.5.1 L-Plate

Problem Description

Figure 1 shows the L-plate. A steady-state heat transfer analysis is performed. The temperature of the L-plated is determined. All dimensions are in meters.

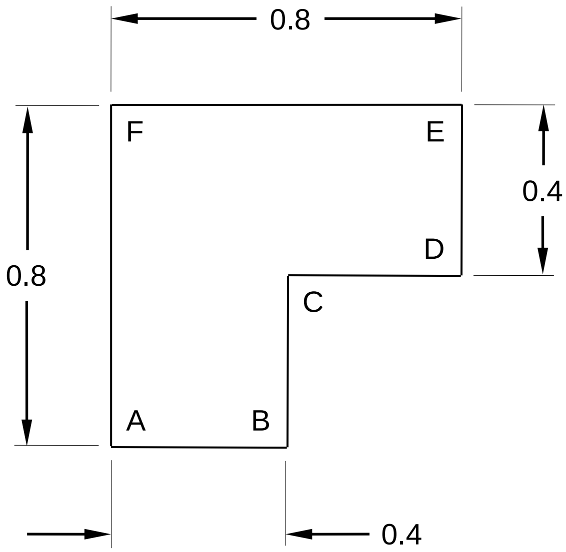


Figure 1. L-Plate

Autodesk Inventor Nastran Analysis Model Filenames

- vm6_5_1.nas

Model Data

Finite Element Modeling

- 21 nodes, 12 5-DOF/node quadrilateral plate elements

Units

meter/Newton/second

Model Geometry

Length: L_1 = 0.8 m and L_2 = 0.4 m

Material Properties

λ = 1.0 W/m °C

Boundary Conditions

Nodal Temperatures. AF side: temperature is set to 10 °C. DE side: temperature is set to 0 °C.

Solution Type

Steady-State Heat Transfer

Comparison of Results

The tabular results are given in Table 1.

Table 1. Temperature Results (°C)

Node	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
8	7.869	7.861	0.1
9	5.495	5.502	0.1
10	2.816	2.845	1.0
19	8.018	8.025	0.1
18	5.680	5.670	0.2
20	2.881	2.960	2.7
17	8.514	8.505	0.1
6	6.667	6.667	0.0
16	2.972	2.990	0.6
21	9.001	9.015	0.1
15	8.640	8.661	0.2
14	9.316	9.294	0.2
5	9.009	8.996	0.1

References

1. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris, Afnor Technique, 1990. Test No. TPLP01/89.

7. Buckling Verification Using Theoretical Solutions

The purpose of these buckling test cases is to verify the functionality of Autodesk Inventor Nastran using theoretical solutions of well-known engineering buckling problems. The test cases are basic in form and have closed-form theoretical solutions.

The theoretical solutions given in these examples are from reputable engineering texts. For each case, a specific reference is cited. All theoretical reference texts are listed in Appendix A.

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

For most cases, discrepancies between Autodesk Inventor Nastran computed and theoretical results are minor and can be considered negligible. To produce exact results, for most cases, a larger number of elements would need to be used. Element quantity is chosen to achieve reasonable engineering accuracy in a reasonable amount of time.

7.1 Buckling of a Thin Walled Cylinder

Problem Description

Figure 1 shows the half modeled thin walled cylinder. A buckling analysis is performed on the model. Critical buckling load is determined. All dimensions are in inches.

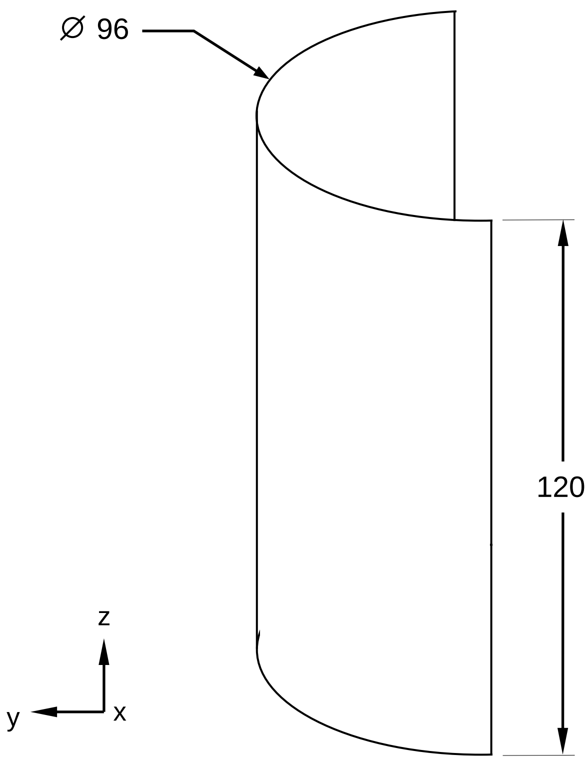


Figure 1. Half Cylinder Model

Autodesk Inventor Nastran Analysis Model Filenames

- vm7_1a.nas – Test 1
- vm7_1b.nas – Test 2
- vm7_1c.nas – Test 3

Model Data

Finite Element Modeling

- Test 1 (vm7_1a): 861 nodes, 800 5-DOF/node quadrilateral plate elements
- Test 2 (vm7_1b): 3321 nodes, 3200 5-DOF/node quadrilateral plate elements
- Test 3 (vm7_1c): 13041 nodes, 12800 5-DOF/node quadrilateral plate elements

Units

inch/pound/second

Model Geometry

Length: L = 120 in
Radius: R = 48 in

Material Properties

Young's Modulus: E = 10.0 E+6 psi
Poisson's Ratio: ν = 0.3

Boundary Conditions

Cylindrical coordinates are used for the boundary conditions in which Z-axis is running vertically through the center of the cylinder, R-axis is projected outward normal to the nodes, and T-axis is tangential to the elements. The half model is constrained along the sliced edges using symmetry boundary conditions. One end of the cylinder is constrained in all translations. The other end is constrained in the R and T-translations only. A 1 psi pressure load is applied to all elements acting normal to the elements and toward the center of the cylinder. This acts as a crushing load. A separate load set is created and a downward nodal load is applied along all the nodes on the top edge of the cylinder of 1000 lb and 500 lb for the two end nodes. These nodal forces are applied to the end of the cylinder which contains the R and T-translation constraints.

Solution Type

Buckling

Comparison of Results

Results are calculated for full cylinder model.

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Critical Buckling Load (Test 1) (lbs)	380,276	382,746	0.6
Critical Buckling Load (Test 2) (lbs)	380,276	382,496	0.6
Critical Buckling Load (Test 3) (lbs)	380,276	379,680	0.2

References

1. NASA SP-8007 Buckling of Thin-Walled Circular Cylinders.

7.2 Buckling of a Bar with Hinged Ends

Problem Description

Figure 1 shows the bar model. A buckling analysis is performed on the model. Critical buckling load is determined. All dimensions are in inches.

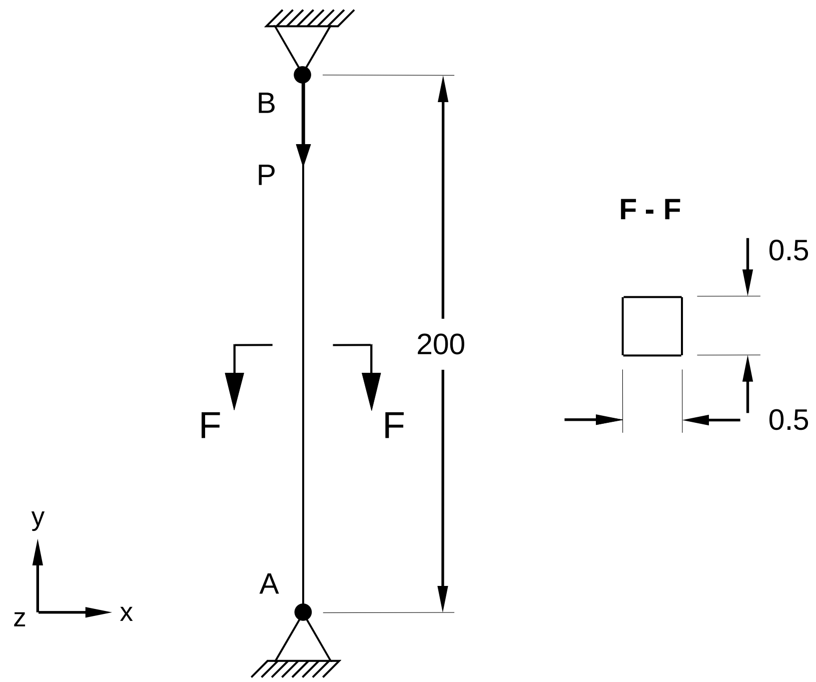


Figure 1. Bar Model

Autodesk Inventor Nastran Analysis Model Filenames

- vm7_2.nas

Model Data

Finite Element Modeling

- 11 nodes, 10 6-DOF/node beam elements

Units

inch/pound/second

Model Geometry

Length: L = 200 in
Width: w = 0.5 in
Thickness: t = 0.5 in

Cross Sectional Properties

Area: $A = 0.25\ \text{in}^2$

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Poisson's Ratio: $\nu = 0.3$

Boundary Conditions

Supports are pinned: the bottom node is constrained in all translations, and in the Y-rotation, while the upper node is constrained in the X and Z-translations. A unit load P is applied at upper end in the negative Y-direction.

Solution Type

Buckling

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Critical Buckling Load (lbs)	38.553	38.551	0.0

References

1. Timoshenko, S., *Strength of Materials, Part 2, Elementary Theory and Problems*, 3rd Edition. New York: D. VanNostrand Co., Inc., 1956.

7.3 Buckling of a Bar with Hinged Ends Using Plates

Problem Description

Figure 1 shows the bar model using plates. A buckling analysis is performed on the model. Critical buckling load is determined. All dimensions are in inches.

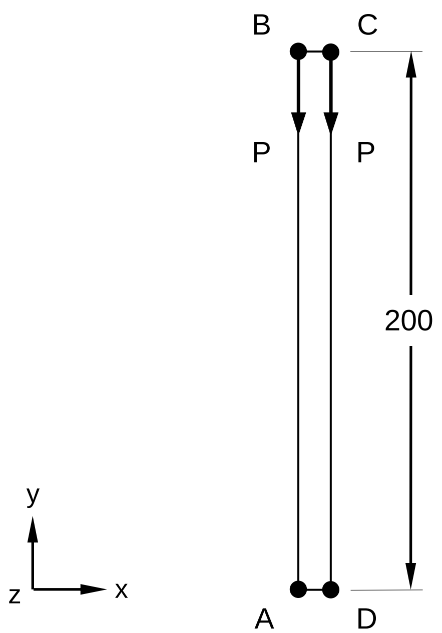


Figure 1. Bar with Hinged Ends Using Plates

Autodesk Inventor Nastran Analysis Model Filenames

- vm7_3.nas

Model Data

Finite Element Modeling

- 42 nodes, 20 5-DOF/node quadrilateral plate elements

Units

inch/pound/second

Model Geometry

Length: L = 200 in
Width: w = 0.5 in
Thickness: t = 0.5 in

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Poisson's Ratio: ν = 0.3

Boundary Conditions

Supports are pinned: the bottom left node (point A) is constrained in all translations, and in the Y-rotation, while the bottom right node (point D) is constrained in the Y and Z-translation and the Y-rotation. The top left node (point B) is constrained in the X and Z-translations, while the top right node (point C) is constrained in the Z-translation. A half unit load P is applied at each node at the upper end in the negative Y-direction.

Solution Type

Buckling

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Critical Buckling Load (lbs)	38.553	38.639	0.2

References

1. Timoshenko, S., *Strength of Materials, Part 2, Elementary Theory and Problems*, 3rd Edition. New York: D. VanNostrand Co., Inc., 1956.

7.4 Buckling of a Bar with Hinged Ends Using Solids

Problem Description

Figure 1 shows the bar model using solids. A buckling analysis is performed on the model. Critical buckling load is determined. All dimensions are in inches.

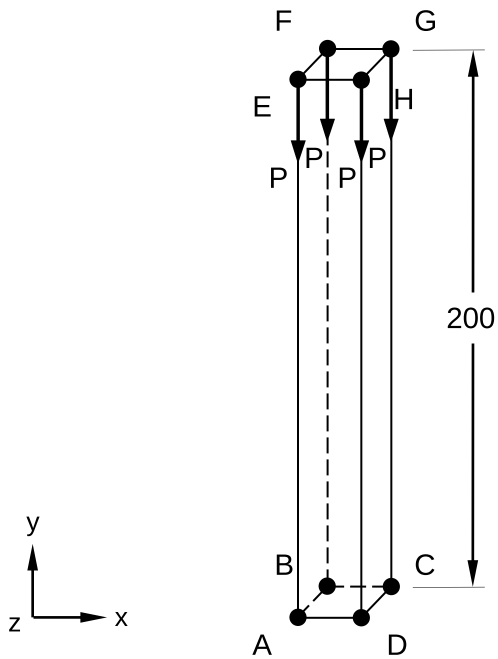


Figure 1. Bar with Hinged Ends Using Solids

Autodesk Inventor Nastran Analysis Model Filenames

- vm7_4.nas

Model Data

Finite Element Modeling

- 84 nodes, 20 3-DOF/node brick elements

Units

inch/pound/second

Model Geometry

Length: L = 200 in
Width: w = 0.5 in
Thickness: t = 0.5 in

Cross Sectional Properties

Area: $A = 0.25\ \text{in}^2$

Material Properties

Young's Modulus: $E = 30.0\text{E+6}$ psi
Poisson's Ratio: $\nu = 0.3$

Boundary Conditions

Supports are pinned: the bottom left hand side nodes (points A and B) are constrained in all translations, while the upper left hand side nodes (points E and F) are constrained in the X and Z-translations. A quarter unit load P is applied at each node at the upper end in the negative Y-direction.

Solution Type

Buckling

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Critical Buckling Load (lbs)	38.553	38.724	0.4

References

1. Timoshenko, S., *Strength of Materials, Part 2, Elementary Theory and Problems*, 3rd Edition. New York: D. VanNostrand Co., Inc., 1956.

7.5 Buckling of a Rectangular Plate Under Concentrated Center Loads

Problem Description

Figure 1 shows the plate model. A buckling analysis is performed on the model. Critical buckling load is determined. All dimensions are in meters.

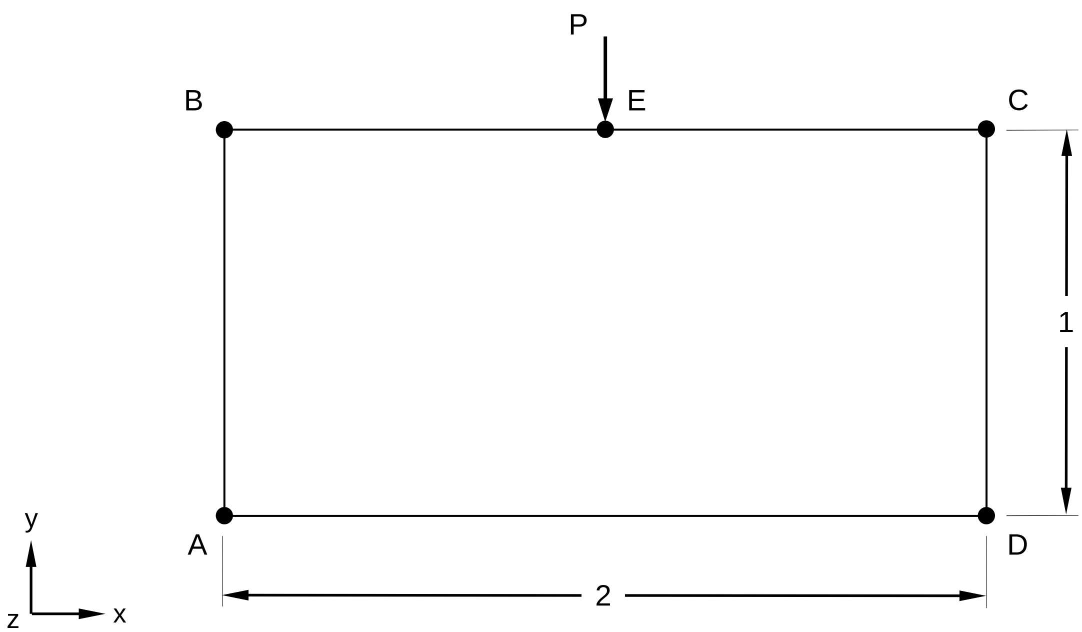


Figure 1. Plate Model

Autodesk Inventor Nastran Analysis Model Filenames

- vm7_5.nas

Model Data

Finite Element Modeling

- 84 nodes, 72 5-DOF/node quadrilateral plate elements

Units

meter/Newton/second

Model Geometry

Length: L = 2.0 m
Width: w = 1.0 m
Thickness: t = 0.01 m

Material Properties

Young's Modulus: $E = 200.0$ GPa
Poisson's Ratio: $\nu = 0.3$

Boundary Conditions

Point A is constrained in all translations and rotations. Points B and C are constrained in the Z-translation and all rotations. Point D is constrained in Y and Z-translations and all rotations. All other nodes on edges AB and CD are constrained in Z-translation, X and Z-rotations. All other nodes on edge BC are constrained in Z-translation, Y and Z-rotations. All other nodes on edge AD are constrained in Y and Z-translations, X and Z-rotations. A uniform load P of 1.0 kN/m at center of edge BC (point E) is applied in the negative Y-direction.

Solution Type

Buckling

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Critical Buckling Load (kN)	330	325	1.5

References

1. Timoshenko, S. and Gere, J. M., *Theory of Elastic Stability*, 2nd Edition. New York: McGraw-Hill Book Co., Inc., 1961.

7.6 Buckling of a Rectangular Plate Under End Uniform Load

Problem Description

Figure 1 shows the plate model. A buckling analysis is performed on the model. Critical buckling load is determined. All dimensions are in meters.

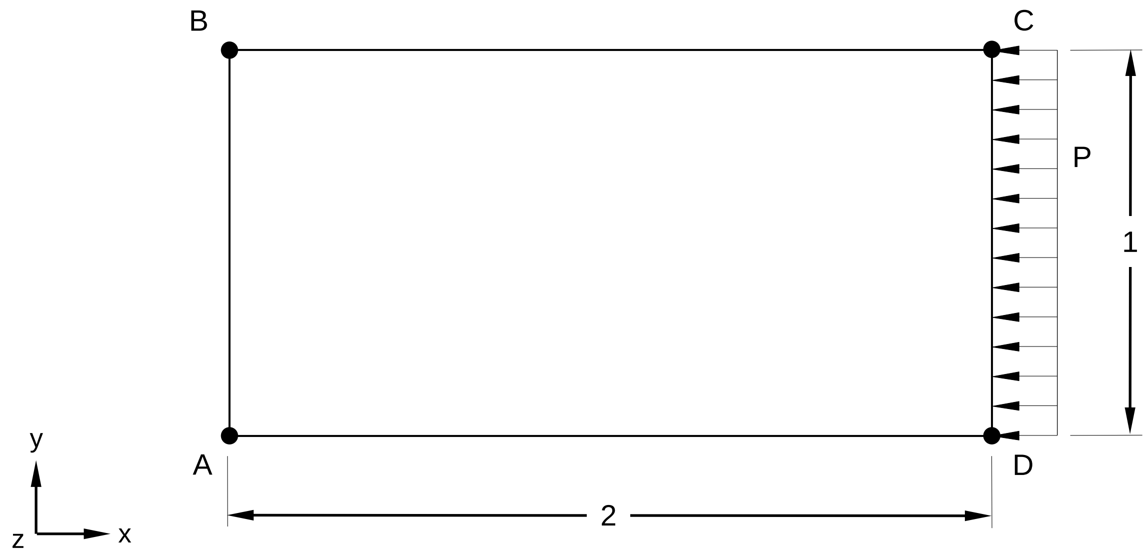


Figure 1. Plate Model

Autodesk Inventor Nastran Analysis Model Filenames

- vm7_6.nas

Model Data

Finite Element Modeling

- 91 nodes, 72 5-DOF/node quadrilateral plate elements

Units

meter/Newton/second

Model Geometry

Length: L = 2.0 m
Width: w = 1.0 m
Thickness: t = 0.01 m

Material Properties

Young's Modulus: E = 200.0 GPa
Poisson's Ratio: ν = 0.3

Boundary Conditions

Point A is constrained in all translations and rotations. Point B is constrained in the X and Z-translations, and all rotations. Point C is constrained in the Z-translation and all rotations. Point D is constrained in Y and Z-translations, and all rotations. All other nodes on edge AB are constrained in X and Z-translations and X and Z-rotations, while all other nodes on edge CD are constrained in Z-translation, and X and Z-rotations. All other nodes on edge BC are constrained in Z-translation, Y and Z-rotations. All other nodes on edge AD are constrained in Y and Z-translations, Y and Z-rotations. A uniform load P = 6.0 kN at edge CD is applied in the negative X-direction.

Solution Type

Buckling

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Critical Buckling Load (kN)	723	723	0.0

References

1. Young, W. C., *Roark's Formulas for Stress and Strain*, 6th Edition. New York: McGraw-Hill Co., 1989.

8. Dynamics Verification Using Standard NAFEMS Benchmarks

The purpose of these dynamic test cases is to verify the functionality of Autodesk Inventor Nastran using standard benchmarks published by NAFEMS (National Agency for Finite Element Methods and Standards, National Engineering Laboratory, Glasgow, U.K.).

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

8.1 Transient Forced Vibration Response – Deep Simply Supported Beam

Problem Description

Figure 1 shows the beam model. A transient dynamic time history analysis is performed on the model. Peak deflection and stress are determined. All dimensions are in meters.

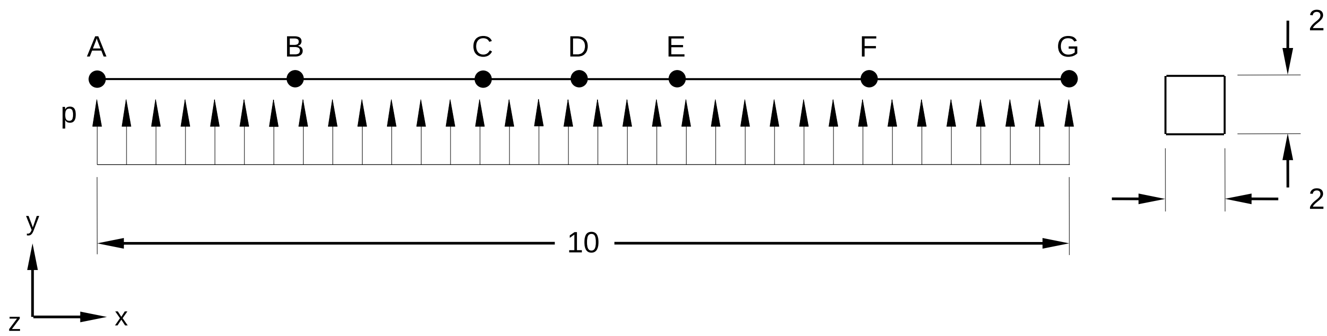


Figure 1. Beam Cross-Section Model

Autodesk Inventor Nastran Analysis Model Filenames

- vm8_1.nas

Model Data

Finite Element Modeling

- 7 nodes, 6 6-DOF/node beam elements

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m

Cross Sectional Properties

Area: $A = 4$ m^2
Square Cross Section = (2.0 m x 2.0 m)

Material Properties

Young's Modulus: E = 200.0 E+6 MPa
Poisson's Ratio: $\nu = 0.3$
Mass Density: $\rho = 8000$ kg/m^3

Boundary Conditions

The beam is loaded with a suddenly applied distributed step load p = 1 E+6 N/m. One end of the beam is fully constrained in all translations and rotations except the Z-rotation. The other end is constrained in the Y and Z-translations, and X and Y- rotations. A 2% critical damping is also applied.

Dynamic Parameters

Number of Time Steps: 1000
Time per Step: 0.0001
Output Intervals: 2

Solution Type

Transient Dynamic/Time History

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Peak Displacement (mm)	1.043	1.030	1.2
Time of Peak Displacement (s)	1.17E+1	1.18E+1	0.9
Peak Stress (N/mm^2)	18.76	18.78	0.1

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Forced Vibration*. Glasgow: NAFEMS, Mar., 1993. Test No. 5T.

8.2 Periodic Forced Vibration Response – Deep Simply Supported Beam

Problem Description

Figure 1 shows the beam model. A transient dynamic time history analysis is performed on the model. Peak deflection and translation are determined. All dimensions are in meters.

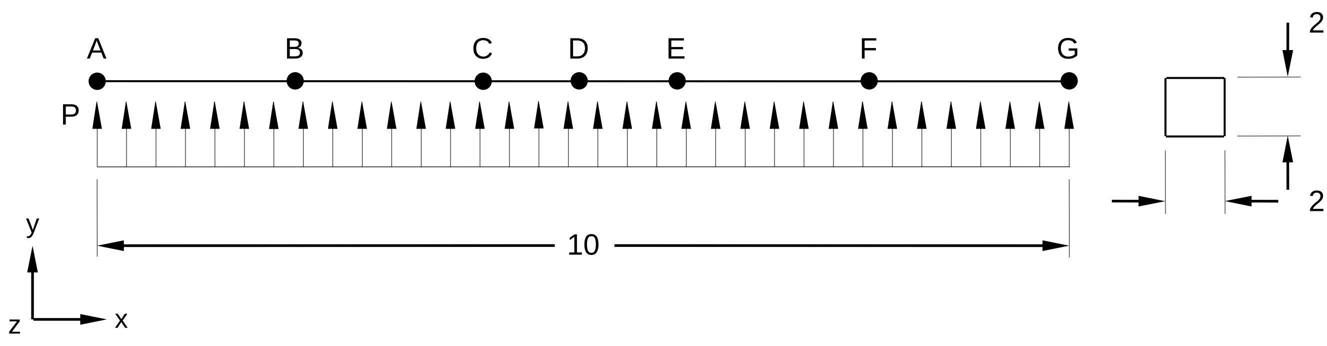


Figure 1. Beam Cross-Section Model

Autodesk Inventor Nastran Analysis Model Filenames

- vm8_2.nas

Model Data

Finite Element Modeling

- 7 nodes, 6 6-DOF/node beam elements

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m

Cross Sectional Properties

Area: $A = 4\ m^2$
Square Cross Section = (2.0 m x 2.0 m)

Material Properties

Young's Modulus: E = 200.0 E+6 MPa
Poisson's Ratio: $\nu = 0.3$
Mass Density: $\rho = 8000\ kg/m^3$

Boundary Conditions

The beam is loaded with a steady state periodic function acting as a distributed load of the form

$$P = P_o(\sin\omega t - \sin 3\omega t)$$

where:

$$P_o = 10^6 N/m$$
$$\omega = 2\pi f$$
$$f = 20Hz$$

The input function is shown in Figure 2.

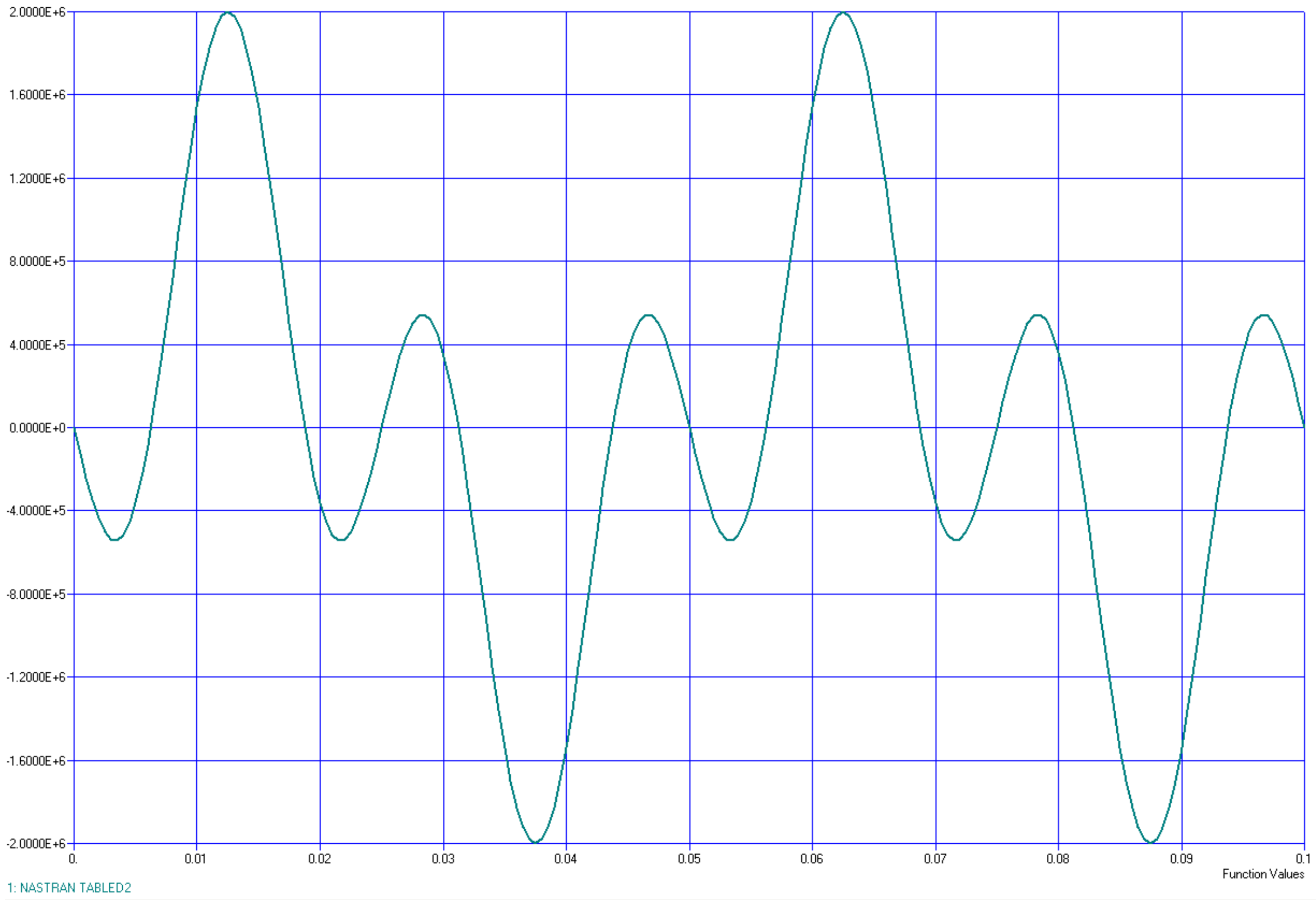


Figure 2. Input Loading Function

One end of the beam is fully constrained in all translations and rotations except the Z-rotation. The other end is constrained in the Y and Z- translations, and X and Y-rotations. A 2% critical damping is also applied.

Dynamic Parameters

Number of Time Steps: 5000
Time per Step: 0.0005
Output Intervals: 5

Solution Type

Transient Dynamic/Time History

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Peak Displacement (mm)	0.951	0.934	1.8
Peak Stress (N/mm^2)	16.97	17.16	1.1

References

1. NAFEMS, *Selected Benchmarks for Forced Vibration*, (J. Maguire, D.J. Dawswell, L. Gould) Test No. 5P.

8.3 Modal Transient Forced Vibration Response – Simply Supported Plate

Problem Description

Figure 1 shows the plate model. A transient dynamic time history analysis is performed on the model. Peak deflection and stress are determined. All dimensions are in meters.

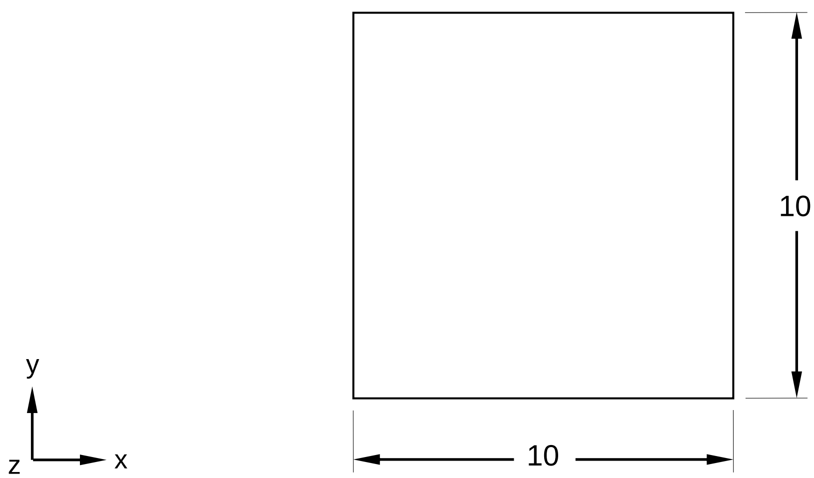


Figure 1. Plate Model

Autodesk Inventor Nastran Analysis Model Filename

- vm8_3.nas

Model Data

Finite Element Modeling

- Mesh (20 x 20): 441 nodes, 400 5-DOF/node quadrilateral plate elements

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Width: w = 10.0 m
Thickness: t = 0.05 m

Cross Sectional Properties

Area: A = 100.0 m^2
Square Cross Section = (10.0 m x 10.0 m)

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Poisson's Ratio: ν = 0.3
Mass Density: ρ = 8000 kg/m^3

Boundary Conditions

The boundary conditions are as follows: fixed Tx, Ty, and Rz at all nodes; fixed Tz along all 4 edges; fixed Rx along edges X = 0 and X = 10 m (left and right edges); fixed Ry along Y = 0 and Y = 10 m (top and bottom edges). The plate is loaded with a suddenly applied step load acting as a pressure load of 100 N/m^2 over the whole plate.

Dynamic Parameters

Number of time steps: 200
Time per step: 0.002
Output Intervals: 0

Solution Type

Transient Dynamic/Time History

Solution Method

Modal Transient

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Peak Displacement (mm)	3.523	3.436	2.5
Time of Peak Displacement (s)	0.210	0.210	0.0
Peak Stress (N/mm^2)	2.484	2.476	0.3

References

1. NAFEMS, *The International Association for the Engineering Analysis Company*, Report No. E1261/R002, Issue 03/9th, February 1989.

8.4 Harmonic Forced Vibration Response – Simply Supported Plate

Problem Description

Figure 1 shows the plate model. A harmonic forced vibration response analysis is performed on the model. Peak deflection and stress are determined, as well as frequencies. All dimensions are in meters.

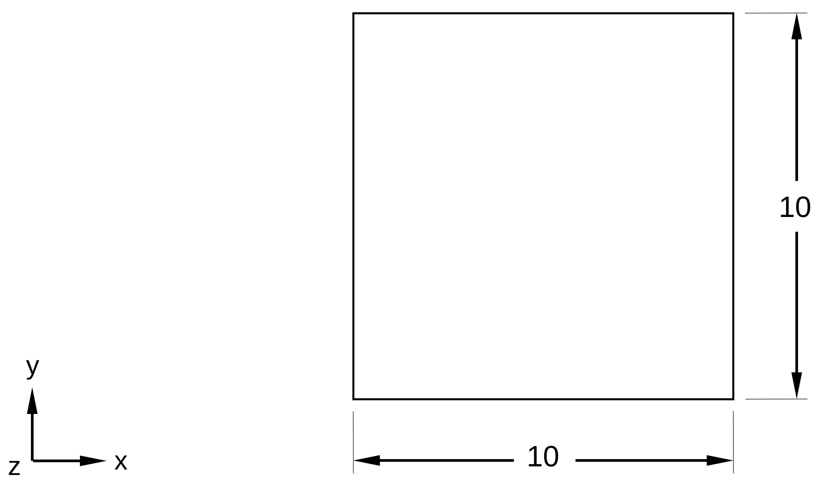


Figure 1. Plate Model

Autodesk Inventor Nastran Analysis Model Filename

- vm8_4.nas

Model Data

Finite Element Modeling

- Mesh (20 x 20): 441 nodes, 400 5-DOF/node quadrilateral plate elements

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Width: w = 10.0 m
Thickness: t = 0.05 m

Cross Sectional Properties

Area: A = 100.0 m^2
Square Cross Section = (10.0 m x 10.0 m)

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Poisson's Ratio: ν = 0.3
Mass Density: ρ = 8000 kg/m^3

Boundary Conditions

The boundary conditions are as follows: fixed Tx, Ty, and Rz at all nodes; fixed Tz along all 4 edges; fixed Rx along edges X = 0 and X = 10 m (left and right edges); fixed Ry along Y = 0 and Y = 10 m (top and bottom edges). A step loading of magnitude 100 N/m^2 is applied to the entire plate, as pressure load. Damping is 2% critical across all modes. A frequency dependent function will be applied where frequency range from 0 to 4.16 Hz. The effects of gravity are not considered in this model.

Dynamic Parameters

Number of Modes: 16

Solution Type

Frequency/Harmonic Response

Solution Method

Modal Frequency

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Peak Displacement (mm)	45.42	45.25	0.4
Frequency (Hz)	2.377	2.377	0.0
Peak Stress (N/mm^2)	30.03	31.64	5.1

References

1. NAFEMS, *The International Association for the Engineering Analysis Company*, Report No. E1261/R002, Issue 03/9th, February 1989.

8.5 Random Forced Vibration Response – Simply Supported Plate

Problem Description

Figure 1 shows the plate model. A random forced vibration response analysis is performed on the model. Peak deflection and PSD stress are determined, as well as frequencies. All dimensions are in meters.

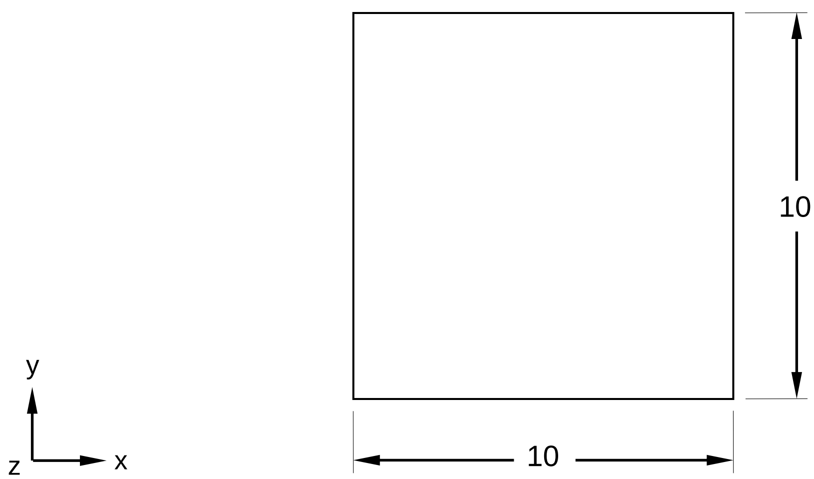


Figure 1. Plate Model

Autodesk Inventor Nastran Analysis Model Filename

- vm8_5.nas

Model Data

Finite Element Modeling

- Mesh (20 x 20): 441 nodes, 400 5-DOF/node quadrilateral plate elements

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Width: w = 10.0 m
Thickness: t = 0.05 m

Cross Sectional Properties

Area: A = 100.0 m^2
Square Cross Section = (10.0 m x 10.0 m)

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Poisson's Ratio: ν = 0.3
Mass Density: ρ = 8000 kg/m^3

Boundary Conditions

The boundary conditions are as follows: fixed Tx, Ty, and Rz at all nodes; fixed Tz along all 4 edges; fixed Rx along edges X = 0 and X = 10 m (left and right edges); fixed Ry along Y = 0 and Y = 10 m (top and bottom edges). A random forcing with uniform power spectral density (of force) PSD is applied to the entire plate (PSD = (100 N/m^2)^2/Hz). Damping is 2% critical across all modes. A frequency dependent function will be applied where frequency range from 0 to 4.16 Hz. The effects of gravity are not considered in this model.

Dynamic Parameters

Number of Modes: 16

Solution Type

Random Response

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Peak Displacement PSD (mm^2/Hz)	2,063.20	2,047.86	0.7
Frequency (Hz)	2.377	2.377	0.0
Peak Stress PSD (N/mm^2)^2/Hz	1,025.44	1,000.80	2.4

References

1. NAFEMS, *The International Association for the Engineering Analysis Company*, Report No. E1261/R002, Issue 03/9th, February 1989.

8.6 Direct Transient Forced Vibration Response – Simply Supported Plate

Problem Description

Figure 1 shows the plate model. Two transient dynamic time history analyses are performed on the model, one with structural damping and another one with Rayleigh damping. Peak deflection and stress are determined. All dimensions are in meters.

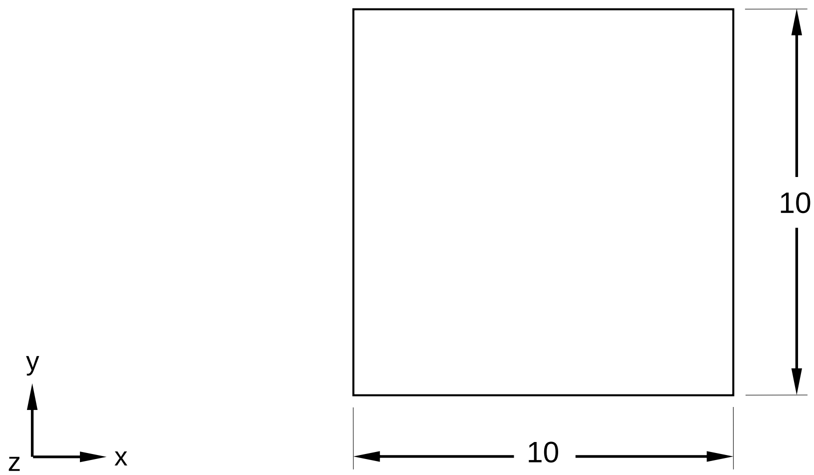


Figure 1. Plate Model

Autodesk Inventor Nastran Analysis Model Filename

- vm8_6.nas

Model Data

Finite Element Modeling

- Mesh (20 x 20): 441 nodes, 400 5-DOF/node quadrilateral plate elements

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Width: w = 10.0 m
Thickness: t = 0.05 m

Cross Sectional Properties

Area: A = 100.0 m^2
Square Cross Section = (10.0 m x 10.0 m)

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Poisson's Ratio: ν = 0.3
Mass Density: ρ = 8000 kg/m^3

Boundary Conditions

The boundary conditions are as follows: fixed Tx, Ty, and Rz at all nodes; fixed Tz along all 4 edges; fixed Rx along edges X = 0 and X = 10 m (left and right edges); fixed Ry along Y = 0 and Y = 10 m (top and bottom edges). A time varying pressure load of 100 N/m^2 is applied on the entire plate. An overall damping coefficient (G) of 0.04 and a frequency for System Damping of 2.377 Hz is applied in the analysis. The frequency was extracted from a normal mode analysis. The effects of gravity are not considered in this model.

Another run is performed with Rayleigh damping, using a Rayleigh damping stiffness matrix scale factor, alpha, of 1.339 E-3, and a Rayleigh damping mass matrix scale factor, beta, of 2.99 E-1.

Dynamic Parameters

Number of Time Steps: 2000
Time per Step: 0.0002
Output Intervals: 0

Solution Type

Transient Dynamic/Time History

Solution Method

Direct Transient

Comparison of Results

The tabular results for the run with the structural damping are given in Table 1, while the tabular results for the run with the Rayleigh damping are given in Table 2.

Table 1. Results for Structural Damping

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Peak Displacement (mm)	3.523	3.459	1.8
Time of Peak Displacement (s)	0.210	0.210	0.0
Peak Stress (N/mm^2)	2.484	2.431	0.3

Table 2. Results for Rayleigh Damping

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Peak Displacement (mm)	3.523	3.459	1.8
Time of Peak Displacement (s)	0.210	0.210	0.0
Peak Stress (N/mm^2)	2.484	2.432	0.3

References

1. NAFEMS, *The International Association for the Engineering Analysis Company*, Report No. E1261/R002, Issue 03/9th, February 1989.

8.7 Direct Frequency Response – Simply Supported Plate

Problem Description

Figure 1 shows the plate model. Two direct frequency response analyses are performed on the model, with structural damping and another one with Rayleigh damping. Peak deflection and stress are determined, as well as frequencies. All dimensions are in meters.

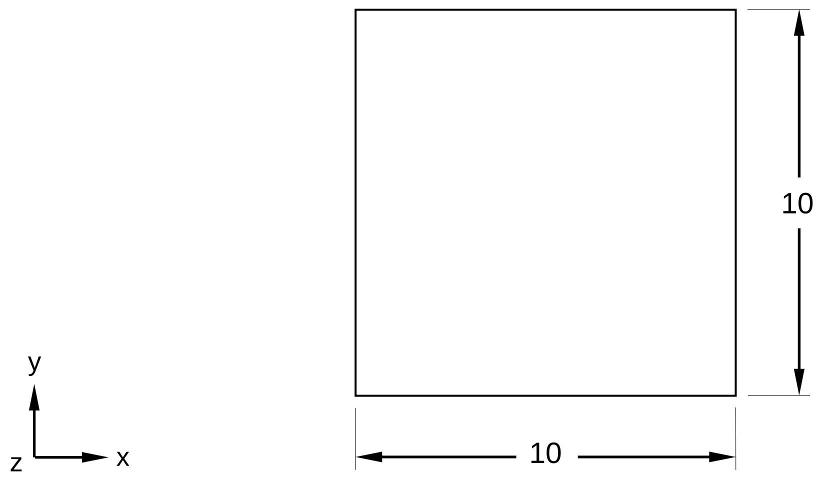


Figure 1. Plate Model

Autodesk Inventor Nastran Analysis Model Filename

- vm8_7.nas

Model Data

Finite Element Modeling

- Mesh (20 x 20): 441 nodes, 400 5-DOF/node quadrilateral plate elements

Units

meter/Newton/second

Model Geometry

Length: L = 10.0 m
Width: w = 10.0 m
Thickness: t = 0.05 m

Cross Sectional Properties

Area: A = 100.0 m^2
Square Cross Section = (10.0 m x 10.0 m)

Material Properties

Young's Modulus: E = 200.0 E+9 N/m^2
Poisson's Ratio: ν = 0.3
Mass Density: ρ = 8000 kg/m^3

Boundary Conditions

The boundary conditions are as follows: fixed Tx, Ty, and Rz at all nodes; fixed Tz along all 4 edges; fixed Rx along edges X = 0 and X = 10 m (left and right edges); fixed Ry along Y = 0 and Y = 10 m (top and bottom edges). A frequency forcing function with pressure load of magnitude 100 N/m^2 is applied on the entire plate. An overall structural damping coefficient (G) of 0.04 is applied. The effects of gravity are not considered in this model.

Another run is performed with Rayleigh damping, using a Rayleigh damping stiffness matrix scale factor, alpha, of 1.339E-3, and a Rayleigh damping mass matrix scale factor, beta, of 2.99 E-1.

Solution Type

Frequency/Harmonic Response

Solution Method

Direct Frequency

Comparison of Results

The tabular results for the run with the structural damping are given in Table 1, while the tabular results for the run with the Rayleigh damping are given in Table 2.

Table 1. Results for Structural Damping

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Peak Displacement (mm)	45.42	45.24	0.4
Frequency (Hz)	2.377	2.377	0.0
Peak Stress (N/mm^2)	30.03	31.63	5.1

Table 2. Results for Rayleigh Damping

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Peak Displacement (mm)	45.42	45.22	0.4
Frequency (Hz)	2.377	2.377	0.0
Peak Stress (N/mm^2)	30.03	31.61	5.0

References

1. NAFEMS, *The International Association for the Engineering Analysis Company*, Report No. E1261/R002, Issue 03/9th, February 1989.

9. Dynamics Verification Using Theoretical Solutions

The purpose of these dynamic test cases is to verify the functionality of Autodesk Inventor Nastran using theoretical solutions of well-known engineering dynamic problems. The test cases are basic in form and have closed-form theoretical solutions.

The theoretical solutions given in these examples are from reputable engineering texts. For each case, a specific reference is cited. All theoretical reference texts are listed in Appendix A.

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

For most cases, discrepancies between Autodesk Inventor Nastran computed and theoretical results are minor and can be considered negligible. To produce exact results, for most cases, a larger number of elements would need to be used. Element quantity is chosen to achieve reasonable engineering accuracy in a reasonable amount of time.

9.1 Seismic Response of a Beam Structure

Problem Description

Figure 1 shows the beam model. Response spectrum analysis is performed on the model. Assume zero damping and lumped mass. The fundamental frequency, fundamental displacement, and the maximum bending stress for the beam structure subjected to a seismic displacement response spectrum are determined. All dimensions are in inches.

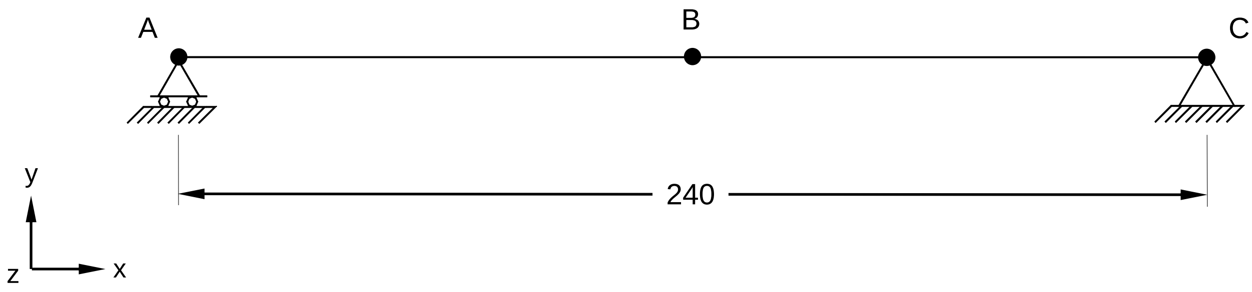


Figure 1. Beam Model

Autodesk Inventor Nastran Analysis Model Filename

- vm9_1.nas

Model Data

Finite Element Modeling

- 26 nodes, 25 beam elements

Units

inch/pond/second

Model Geometry

Length: L = 240 in
Height: h = 14 in

Cross Sectional Properties

Area: A = 273.9726 in^2

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Poisson's Ratio: ν = 0.33
Mass: m = 0.2 lb-sec^2/ in

Boundary Conditions

One end of the beam (point A) is restrained in Z-translation, X and Y-rotations, while the other end of the beam (point C) is restrained in X and Z-translations, and X and Y-rotations. All other nodes are restrained in Z-translation, X and Y-rotations. The middle node of the beam (point B) is also constrained in the Y-translation. A response spectrum dynamic loading is applied as follows:

Frequency (Hz)	**Displacement (in)**
0.1	0.44
10.0	0.44

Solution Type

Modal Summation

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Frequency (Hz)	6.0979	6.0979	0.0
Displacement (in)	-0.560	-0.5594	0.1
Maximum Bending Stress (psi)	20,158	20,158	0.0

References

1. Biggs, J. M., *Introduction to Structural Dynamics*. New York: McGraw-Hill Co., 1964.

9.2 Cantilever Beam Subjected to Sine Plus Tip Load

Problem Description

Figure 1 shows the beam model. A transient dynamic time history analysis using time history forcing functions is performed on the model. The peak displacement at the tip of the beam due to a half-sine pulse tip load is determined. All dimensions are in inches.

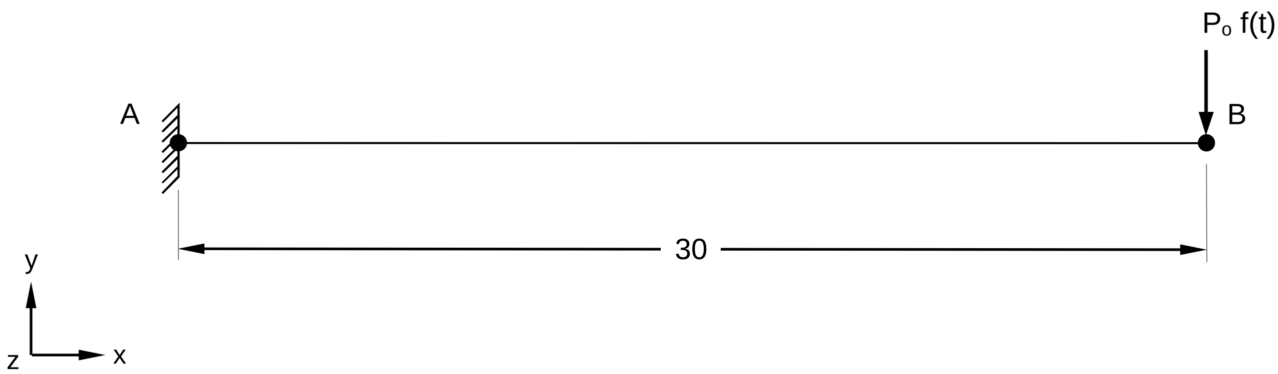


Figure 1. Beam Model

Autodesk Inventor Nastran Analysis Model Filename

- vm9_2.nas

Model Data

Finite Element Modeling

- 7 nodes, 6 beam elements

Units

inch/pound/second

Model Geometry

Length: L = 30 in

Cross Sectional Properties

Area: A = 0.04 in^2
Moment of Inertia: I = 1.33333 E-4 in^4

Material Properties

Young's Modulus: E = 10.0 E+6 psi
Poisson's Ratio: ν = 0.3
Mass Density: ρ = 0.1 lbf/in^3

Boundary Conditions

The beam is loaded with a forcing function which is half-sine pulse:

$$P_o f(t)$$

where:

$$P_o = -0.1 lb$$
$$f(t) = \sin \pi t / T$$
$$T = 2\pi / \omega_1$$

The input function is shown in Figure 2.

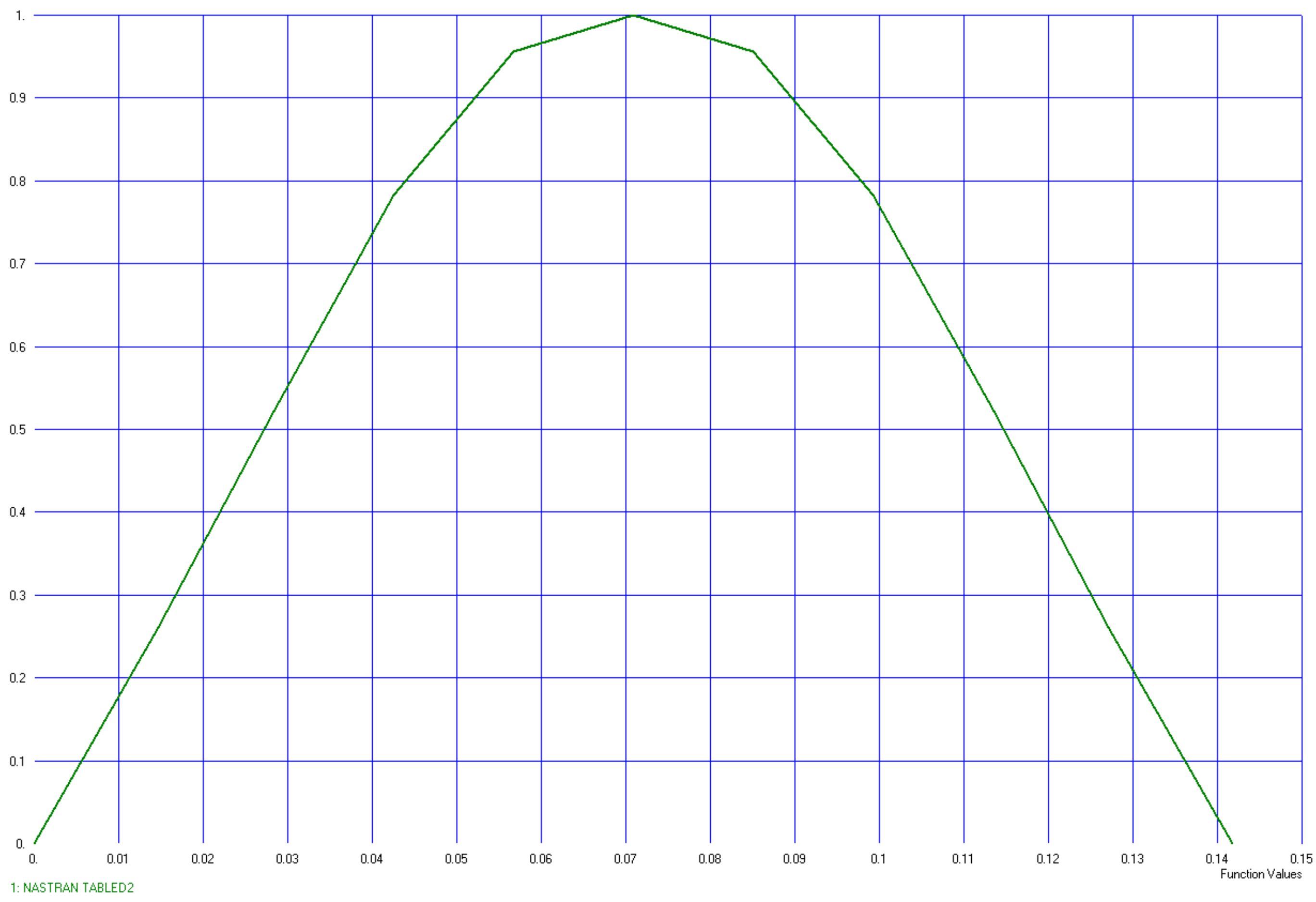


Figure 2. Input Loading Function

One end of the beam (point A) is restrained in all translations and rotations. All other nodes are restrained in Z-translation, X and Y-rotations.

Dynamic Parameters

Number of Time Steps: 283
Time per Step: 0.001
Output Intervals: 1

Solution Type

Modal Transient Response

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Displacement (in)	-1.151	-1.148	0.3

References

1. Warburton, G. B., *The Dynamic Behavior of Structures*. Pergamon Press, 1964.

9.3 Simply Supported Beam, Ramped Nodal Forcing Function, Transient Forced Vibration

Problem Description

Figure 1 shows the beam model. A transient dynamic time history analysis using time history forcing functions is performed on the model. The peak displacement at the tip of the beam and the time at which the pick occurs, and the peak dynamic stress are determined. All dimensions are in inches.

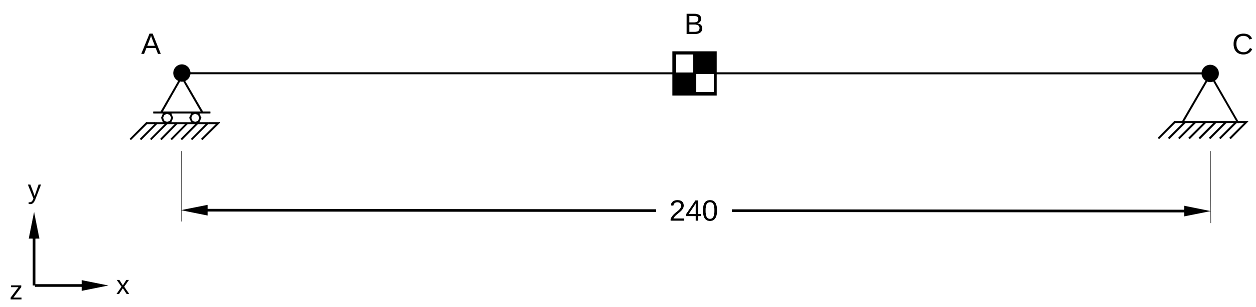


Figure 1. Beam Model

Autodesk Inventor Nastran Analysis Model Filename

- vm9_3.nas

Model Data

Finite Element Modeling

- 3 nodes, 3 beam elements and 1 mass element

Units

inch/pound/second

Model Geometry

Length: L = 240 in
Width: w = 18 in

Cross Sectional Properties

Area: A = 180 in^2
Moment of Inertia: I = 800.6 in^4

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Poisson's Ratio: ν = 0.3
Mass: m = 25.9067 lbf-sec^2/ in

Boundary Conditions

One end of the beam (point A) is restrained in the Y and Z-translations, and X and Y-rotations, while the other end of the beam (point C) is restrained in all translations, and X and Y-rotations. The middle node of the beam (point B) is also constrained in the Z-translation, X and Y-rotations. The beam weight is assumed negligible. A mass m = 25.9067 lbf-sec^2/ in is considered at the center of the beam. A time step force data is applied as follows:

Time (sec)	Force (lbf)
0.000	0
0.075	20000
0.100	20000

Dynamic Parameters

Number of Time Steps: 100
Time per Step: 0.001
Output Intervals: 1

Solution Type

Modal Transient Response

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	Theory	Autodesk Inventor Nastran	Error (%)
Peak Displacement (in)	0.331	0.336	1.5
Time of Peak Displacement (s)	0.092	0.093	1.1
Peak Stress (psi)	18,600	18,438	0.9

References

1. Biggs, J. M., *Introduction to Structural Dynamics*. New York: McGraw-Hill Co., 1964.

9.4 Tower Structure Under a Harmonic Excitation Force

Problem Description

Figure 1 shows the tower structure and its beam equivalent. A transient dynamic time history analysis using time history forcing functions is performed on the structure that is subjected to a lateral harmonic excitation force. The peak displacement at the tip due to a half-sine pulse tip load and the first natural mode are determined. All dimensions are in inches.

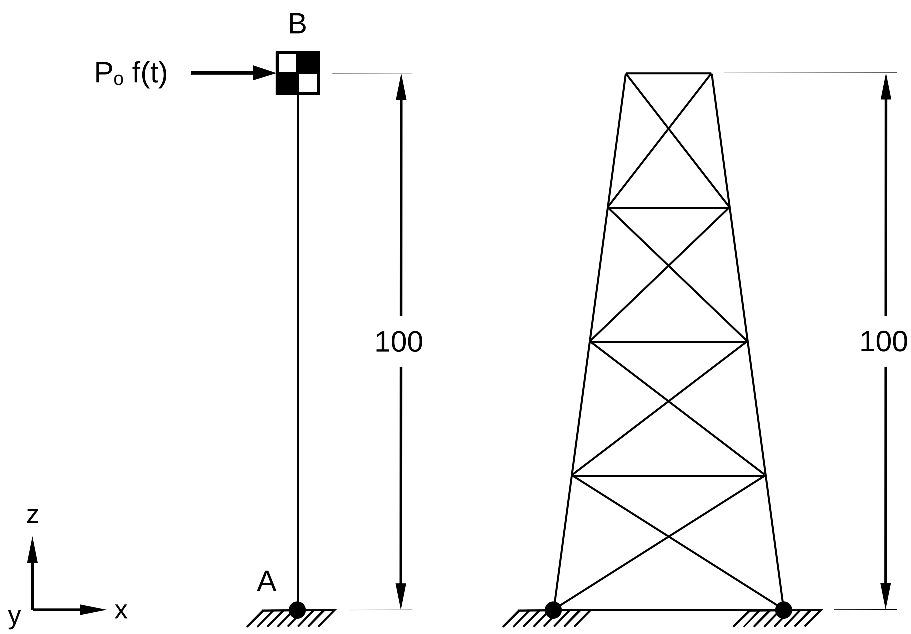


Figure 1. Tower Model

Autodesk Inventor Nastran Analysis Model Filename

- vm9_4.nas

Model Data

Finite Element Modeling

- 11 nodes, 10 beam elements and 1 mass element

Units

inch/pound/second

Model Geometry

Length: L = 100 in

Cross Sectional Properties

Area: A = 1 in^2
Moment of Inertia: I = 1666.667 in^4

Material Properties

Young's Modulus: E = 20.0 E+6 psi
Poisson's Ratio: ν = 0.3
Mass: m = 100 lbf-sec^2/ in (in the X-direction)

Boundary Conditions

A harmonic excitation force is applied at the top node (point B) in the X-direction:

$$P(t) = P_o \sin \omega t \text{ lbf}$$

where:

$$P_o = 1.0\,\mathrm{E} + 5\,\text{lbf}$$
$$\omega = 30\,\text{rad/sec}$$

The input function is shown in Figure 2.

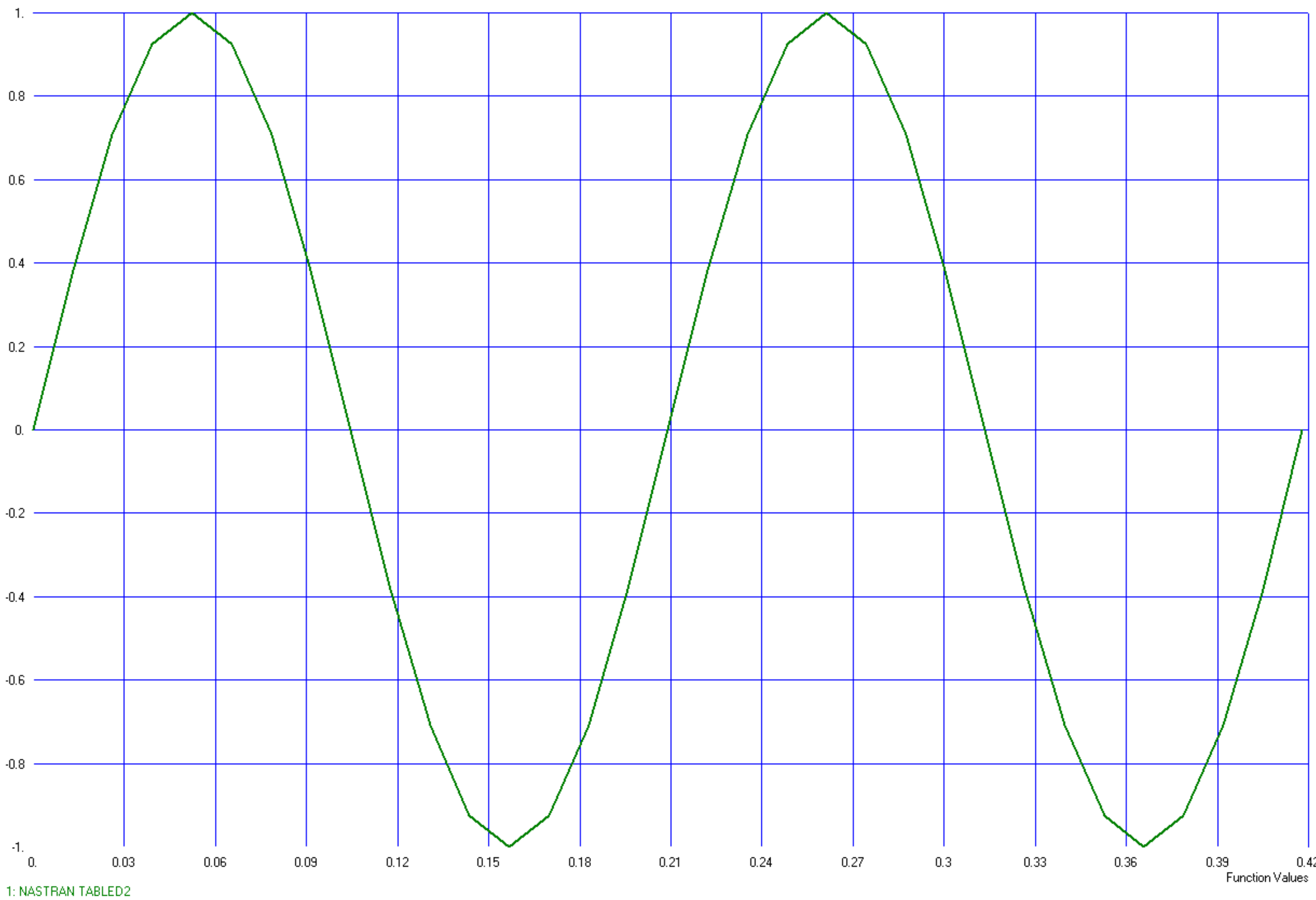


Figure 2. Input Loading Function

One end of the beam (point A) is restrained in all translations and rotations, while all the other nodes are restrained in Y and Z-translations, and all rotations. A mass m = 100 lbf-sec^2/ in is considered at the top of the beam. Damping ratio is zero.

Dynamic Parameters

Number of Time Steps: 300
Time per Step: 0.001
Output Intervals: 1

Solution Type

Modal Transient Response

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Peak Displacement (in) at t = 0.1 sec	1.608	1.585	0.1
Peak Displacement (in) at t = 0.2 sec	-3.187	-3.146	1.3
Peak Displacement (in) at t = 0.3 sec	4.742	4.655	1.8
First Natural Frequency (cycles/sec=1/Hz)	0.200	0.199	0.5

References

1. Paz, M., *Structural Dynamics; Theory and Computation*. 3rd Edition. New York: D. VanNostrand Co., Inc., 1991.

9.5 Simply Supported Beam Subjected to a Traveling Dynamic Load

Problem Description

Figure 1 shows the simply supported beam model. A transient dynamic time history analysis using time history forcing functions and multiple arrival times is performed on the model that is subjected to a dynamic force traveling along the span at a constant velocity. The displacements at the mid-span of the beam with a time history are determined. All dimensions are in inches.

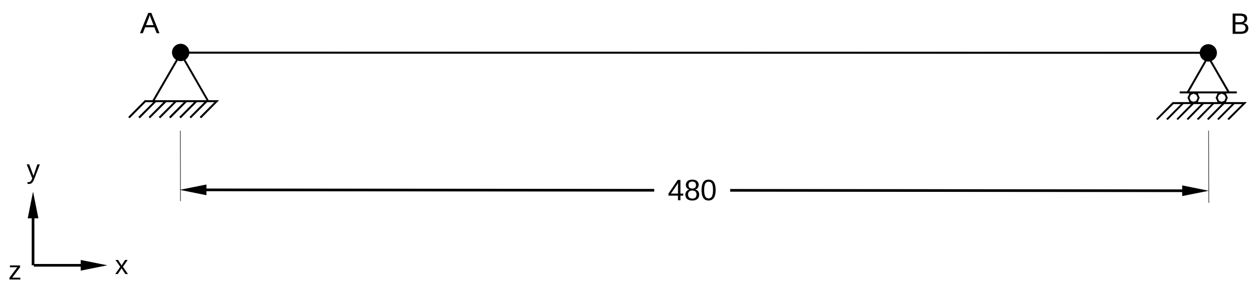


Figure 1. Beam Model

Autodesk Inventor Nastran Analysis Model Filename

- vm9_5.nas

Model Data

Finite Element Modeling

- 21 nodes, 20 beam elements

Units

inch/pound/second

Model Geometry

Length: L = 480 in

Cross Sectional Properties

Area: A = 1 in^2
Moment of Inertia: I = 0.083333 in^4

Material Properties

Young's Modulus: E = 2.4 E+11 psi
Poisson's Ratio: ν = 0.3
Specific Weight: γ = 0.1 lbf/in^3

Boundary Conditions

A triangular excitation force is applied in the Z-direction and travels at 600 in/sec in the X-direction:

$P(t) = P_o$ lbf

where:

$P_o = 1.0$ lbf

The triangular load is shown in Figure 2.

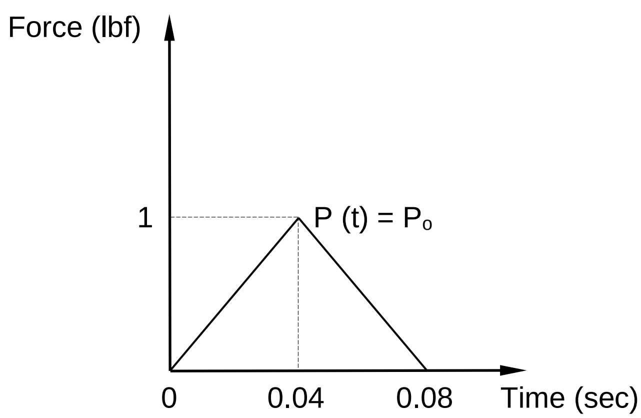


Figure 2. Input Loading Function

One end of the beam (point A) is restrained in all translations, X and Y- rotations, while the other end of the beam (point B) is restrained in the Y and Z-translations, X and Y-rotations. All the other nodes are restrained in the Z-translations, and X and Y-rotations. Damping ratio is zero.

Dynamic Parameters

Number of time steps: 800
Time per step: 0.001
Output Intervals: 1

Solution Type

Modal Transient Response

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Displacement at mid span (in)	-1.100	1.087	1.2
Time at Max. Displacement (sec)	0.280	0.311	11.1
Velocity at mid span (in/sec)	7.500	7.896	5.3
Time at Max. Velocity (sec)	0.670	0.712	6.3

References

1. Biggs, J. M., *Introduction to Structural Dynamics*. New York: McGraw-Hill Co., 1964.

10. Nonlinear Static Verification Using Theoretical Solutions

The purpose of these nonlinear static test cases is to verify the functionality of Autodesk Inventor Nastran using theoretical solutions of nonlinear static problems. The test cases are basic in form and have closed-form theoretical solutions.

The theoretical solutions given in these examples are either from reputable engineering texts or are calculated by Autodesk, Inc. For each applicable case, a specific reference is cited. All theoretical reference texts are listed in Appendix A.

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

For most cases, discrepancies between Autodesk Inventor Nastran computed and theoretical results are minor and can be considered negligible. To produce exact results, for most cases, a larger number of elements would need to be used. Element quantity is chosen to achieve reasonable engineering accuracy in a reasonable amount of time.

10.1 Nonlinear Cable Tension

Problem Description

Figure 1 shows the cable model. A nonlinear static analysis is performed on the model. The cable force of the tensioned cable is determined. All dimensions are in inches.

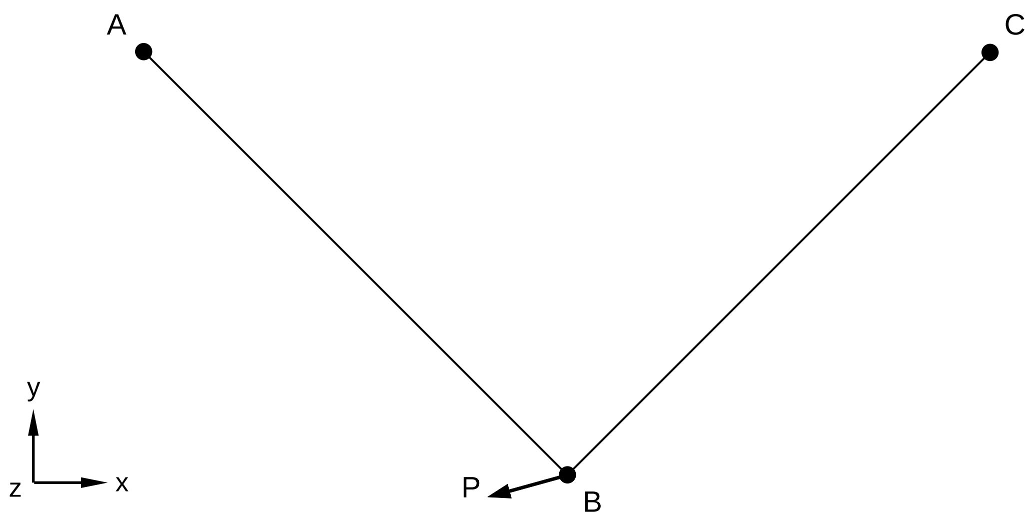


Figure 1. Cable Model

Autodesk Inventor Nastran Analysis Model Filename

- vm10_1.nas

Model Data

Finite Element Modeling

- 3 nodes, 2 1-DOF/node cable elements

Units

inch/pound/second

Model Geometry

Length:
Global X component = 2347.5 in
Global Y component = 2000.0 in

Cross Sectional Properties

Area: A = 706 in^2

Boundary Conditions

The cables are loaded at the joining node with a point load P = 8417.2 lb. The load's X-component Fx is 7700 lb in the negative direction, while the Y-component Fy is 3400 lb in the negative direction. The upper ends of the cables (points A and C) are constrained in all translations and rotations except the Z-rotation. The joined end of the cables (point B) is constrained in the Z-translation and X and Y-rotations.

Nonlinear Parameters

Static
Number of Increments: 5

Solution Type

Nonlinear Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Cable Force of Tensioned Cable (lb)	8,417.2	8,417.4	0.0

Note: The theoretical solution is simply the input load. One cable should slacken while the other takes all of the load, thus the cable force should also equal the input force.

10.2 Cable Supporting Hanging Loads

Problem Description

Figure 1 shows the cable model. Nonlinear static analysis is performed on the plate. The horizontal and vertical reaction forces at support A and the maximum tension in the cable are determined. All dimensions are in feet.

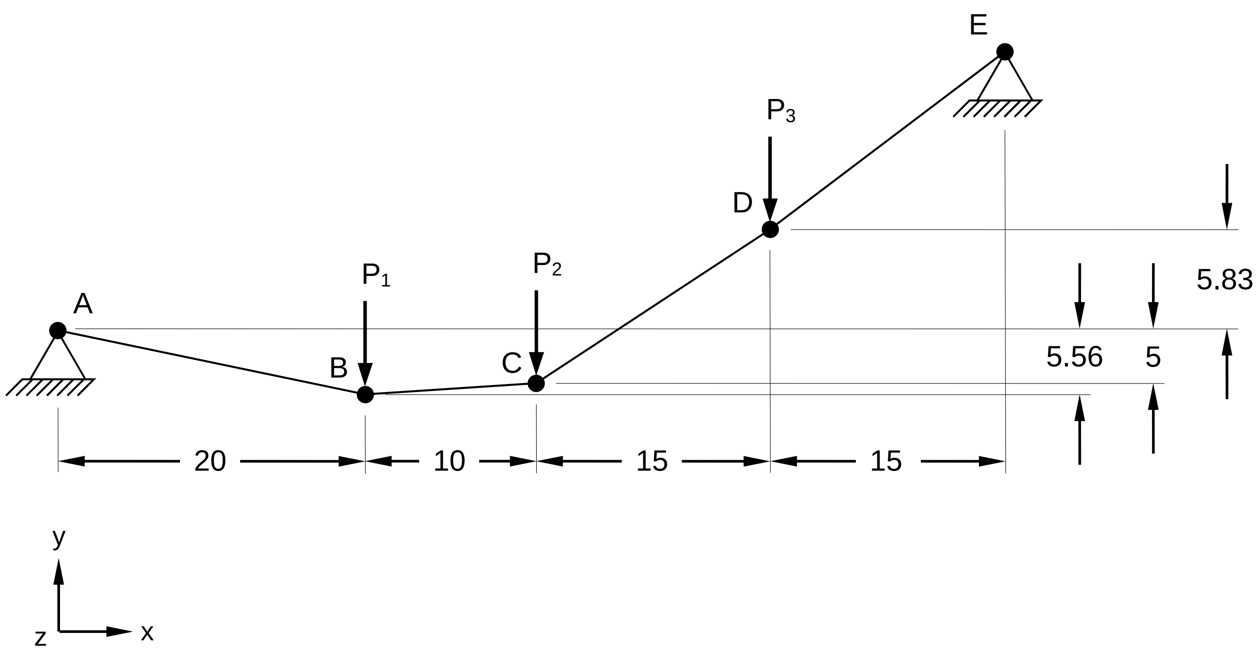


Figure 1. Cable Model Supporting Hanging Loads

Autodesk Inventor Nastran Analysis Model Filename

- vm10_2.nas

Model Data

Finite Element Modeling

- 5 nodes, 4 1-DOF/node cable elements

Units

feet/Kip/second

Model Geometry

Length: L = 60 ft (total horizontal length)

Cross Sectional Properties

Area: A = 0.1 ft^2

Material Properties

Young's Modulus: E = 2.0 E+7 ksf
Poisson's Ratio: ν = 0.3

Boundary Conditions

All translations and rotations are restrained for point A. Points B, C and D have the Z-translation, and the X and Y-rotations constrained, while point E is restrained in all translations. Vertical point loads P1 = 6 Kip, P2 = 12 Kip, and P3 = 4 Kip are applied to the cable model (see Figure 1).

Nonlinear Parameters

Static
Number of Increments: 1

Solution Type

Nonlinear Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	Theory	Autodesk Inventor Nastran	Error (%)
Reaction Force in X-direction (Kips)	18.00	18.00	0.0
Reaction Force in Y-direction (Kips)	5.00	5.00	0.0
Cable Force (Kips)	24.76	24.76	0.0

References

1. Beer, F. P., and Johnston, Jr. E. R., *Vector Mechanics for Engineers, Statics and Dynamics.* New York: McGraw-Hill, Inc., 1962.

10.3 Ten Story Plane Frame

Problem Description

Figure 1 shows the plane frame model. Nonlinear static analysis is performed on the beam (to simulate the P-delta analysis with 2 iterations). The maximum lateral displacement of the 10th story after 2 iterations of the P-delta analysis is determined. All dimensions are in inches.

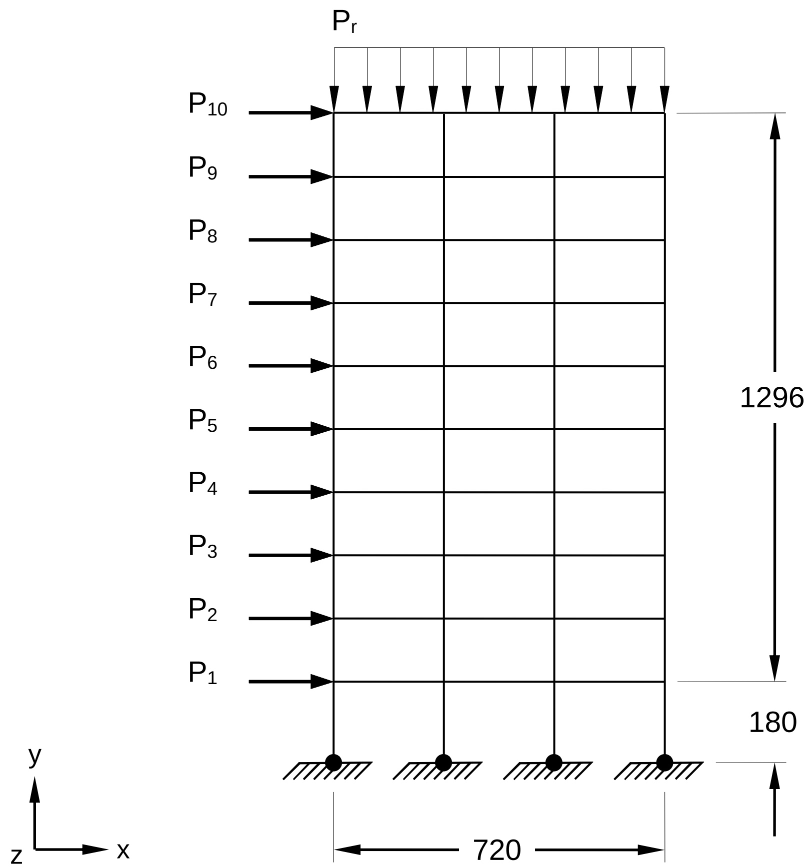


Figure 1. Ten Story Plane Frame Model

Autodesk Inventor Nastran Analysis Model Filename

- vm10_3.nas

Model Data

Finite Element Modeling

- 44 nodes, 70 beam elements

Units

inch/Kip/second

Model Geometry

Length: L = 720 in (3 x 240 in)
Height: h = 1476 in (1 x 180 in and 9 x 144 in)

Material Properties

Young's Modulus: E = 2.9 E+7 psi
Poisson's Ratio: ν = 0.3

Boundary Conditions

All translations and rotations are restrained at supports. All the other nodes are restrained in the Z-translation and X and Y-rotations. A typical roof gravity load P_r = 0.25 Kip/in and floor gravity load P_f =f 0.30 Kip/in (on each floor) are applied to the frame. Also point loads are applied to the left hand side beams, as shown in Figure 1: P_1 =2.79 Kip, P_2 = 5.34 Kip, P_3 = 7.71 Kip, P_4 = 10.08 Kip, P_5 = 12.45 Kip, P_6 = 14.83 Kip, P_7 = 17.20 Kip, P_8 = 19.57 Kip, P_9 = 21.94 Kip and P_{10} = 30.22 Kip.

Nonlinear Parameters

Static
Number of Increments: 1

Solution Type

Nonlinear Static

Comparison of Results

Tabular results are given in Table 1.

Table 1. Results

Description	Theory	Autodesk Inventor Nastran	Error (%)
Lateral Displacement at 10th Story (in)	8.51	8.61	1.2

References

1. Naeim, F., *The Seismic Design Handbook*. New York: VanNostrand Reinhold Co., 1989.

10.4 Straight Cantilever with Axial End Point Load

Problem Description

Figure 1 shows the cantilever model. A nonlinear static analysis is performed on the model. The free end tip displacement and rotations at different load increments are determined. All dimensions are in inches.

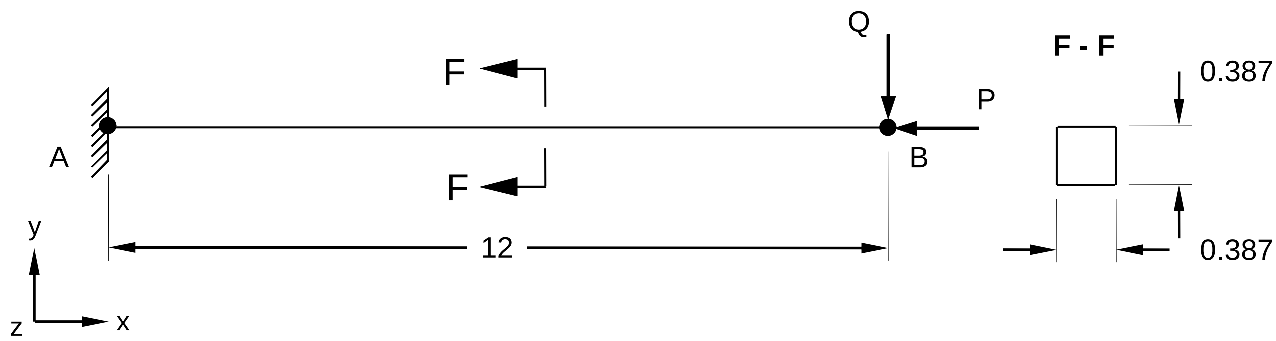


Figure 1. Cantilever Model

Autodesk Inventor Nastran Analysis Model Filename

- vm10_4.nas

Model Data

Finite Element Modeling

- 13 nodes, 12 bar elements

Units

inch/pound/second

Model Geometry

Length: L = 12 in

Cross Sectional Properties

Area: A = 0.15 in^2
Square Cross Section = (0.387 in x 0.387 in)

Material Properties

Young's Modulus: E = 2.0 E+7 psi
Shear Modulus of Elasticity: G = 1.0 E+7 psi
Poisson's Ratio: ν = 0.0

Boundary Conditions

The cantilever is constrained in all translations and rotations at point A. All other nodes are constrained in the Z-translation, X and Y-rotations. An axial force (P) and shear force (Q) at the free end are applied in increments up to a maximum value of P/P_{cr} = 9.116, where $P_{cr} = (\pi^2 EI) / 4L^2$ = 96.4 pounds, and Q = P/10.

Nonlinear Parameters

Static
Number of Increments: 15

Solution Type

Nonlinear Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Applied Force P/P_{cr} = 1.152	1.152	1.152	N/A
Applied Force P/P_{cr} = 2.541	2.541	2.541	N/A
Applied Force P/P_{cr} = 9.116	9.116	9.116	N/A
Tip Displacement in X-direction U_x/L (P/P_{cr} = 1.152)	0.741	0.743	0.3
Tip Displacement in X-direction U_x/L (P/P_{cr} = 2.541)	0.107	0.105	1.9
Tip Displacement in X-direction U_x/L (P/P_{cr} = 9.116)	0.575	0.575	0.0
Tip Displacement in Y-direction U_y/L (P/P_{cr} = 1.152)	0.593	0.592	0.2
Tip Displacement in Y-direction U_y/L (P/P_{cr} = 2.541)	0.750	0.752	0.3
Tip Displacement in Y-direction U_y/L(P/P_{cr} = 9.116)	0.421	0.424	0.7
Tip Rotation φ (P/P_{cr} = 1.152) (degrees)	60.0	59.8	0.3
Tip Rotation φ (P/P_{cr} = 2.541) (degrees)	140.0	140.0	0.0
Tip Rotation φ (P/P_{cr} = 9.116) (degrees)	176.0	176.1	0.0

References

1. Timoshenko, S. and Gere, J. M., *Theory of Elastic Stability*, 3rd Edition. New York: McGraw-Hill Book Co., Inc., 1970.

10.5 Residual Stress Problem

Problem Description

Figure 1 shows the three tie rods model. A nonlinear static analysis is performed on the model. The displacement at point A under load P1 and the residual stress in the center rod at load P2, where the three rods become fully plastic and then the load is fully released, are determined. All dimensions are in inches.

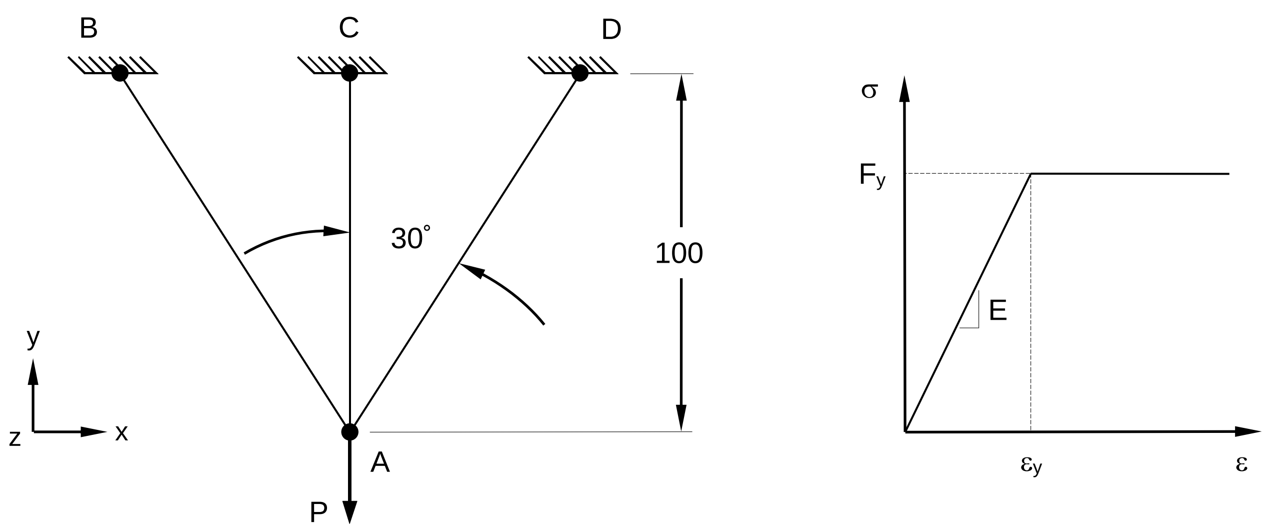


Figure 1. Three Tie Rods Model

Autodesk Inventor Nastran Analysis Model Filename

- vm10_5.nas

Model Data

Finite Element Modeling

- 4 nodes, 3 rod elements

Units

inch/pound/second

Model Geometry

Length: L = 100 in

Cross Sectional Properties

Area: A = 1 in^2

Material Properties

Young's Modulus: E = 30.0 E+6 psi
Poisson's Ratio: ν = 0.3
Yield Force: F_y = 30.0 E+3 psi

Boundary Conditions

All the top nodes (points B, C, and D) are constrained in all translations and rotations. A force P1 = 51,961.52422 lb and P2 = 81,961.52422lb is applied at point A in the negative Y-direction; then unload the applied load in the same increments.

Nonlinear Parameters

Static
Number of Increments: 50

Solution Type

Nonlinear Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Tip Deflection @ Point A (in)	-7.533E-2	-7.531E-2	0.0
Residual Stress @ Center Rod (psi)	-5,660	-5,639	0.4

References

1. Crandall, S. H., and Dahl, N. C., *An Introduction to the Mechanics of Solids*. New York: McGraw-Hill Book Co., Inc., 1959.

11. Nonlinear Static Verification Using Standard NAFEMS Benchmarks

The purpose of these nonlinear static test cases is to verify the functionality of Autodesk Inventor Nastran using benchmarks published by NAFEMS (National Agency for Finite Element Methods and Standards, National Engineering Laboratory, Glasgow, U.K.).

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

11.1 Elastic Large Deformation Response of a Z-shaped Cantilever Under an End Load

Problem Description

Figure 1 shows the Z-shaped cantilever model. A nonlinear static analysis is performed on the model. The tip displacement at different load increments is determined. All dimensions are in inches.

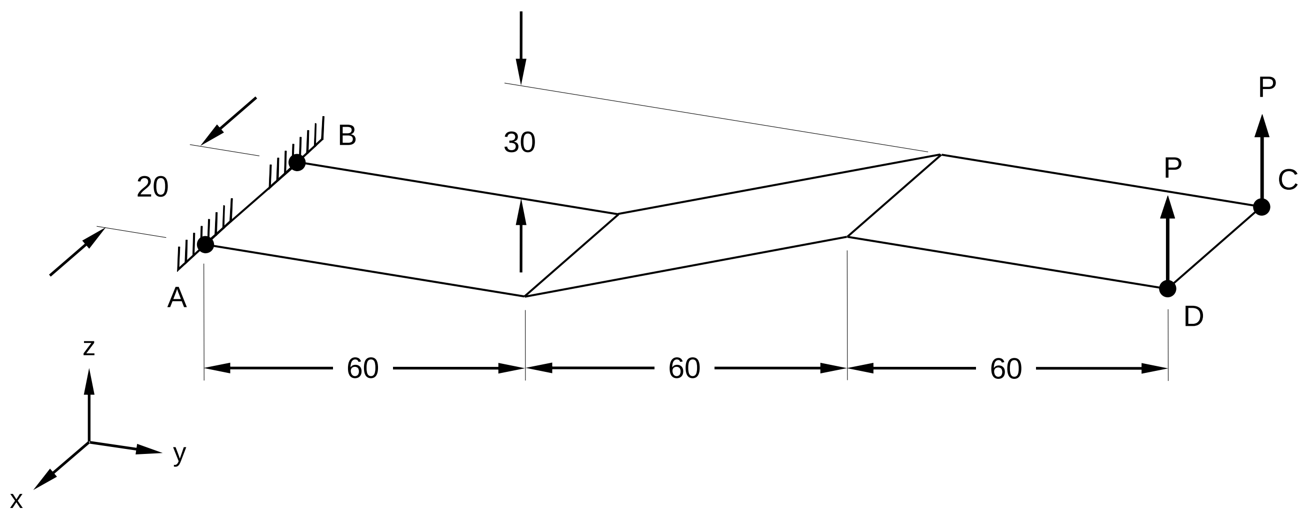


Figure 1. Z-shaped Cantilever Model

Autodesk Inventor Nastran Analysis Model Filename

- vm11_1.nas

Model Data

Finite Element Modeling

- 95 nodes, 72 5-DOF/node quadrilateral plate elements

Units

inch/pound/second

Model Geometry

Length: L = 2000 in
Width: w = 20 in
Thickness: t = 1.7 in

Material Properties

Young's Modulus: E = 2.05 E+5 psi
Poisson's Ratio: ν = 0.25

Boundary Conditions

The plate is constrained in all translations and rotations along edge AB. A uniform load P = 4000 lbs is applied at the free end (edge CD) in the positive Z-direction.

Nonlinear Parameters

Static
Number of Increments: 100

Solution Type

Nonlinear Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Tip Displacement at 104.5 lbs Applied Load (in)	80.42	78.43	2.5
Tip Displacement at 1263 lbs Applied Load (in)	133.10	132.10	0.8
Tip Displacement at 4000 lbs Applied Load (in)	143.50	142.70	0.6

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Geometric Non-Linear Behavior of Shells*. Glasgow: NAFEMS, Publication R0024. Test 3DNLG-1.

11.2 Straight Cantilever with End Moment

Problem Description

Figure 1 shows the cantilever beam model. A nonlinear static analysis is performed on the model. The free end tip displacement and rotations at different load increments are determined. All dimensions are in meters.

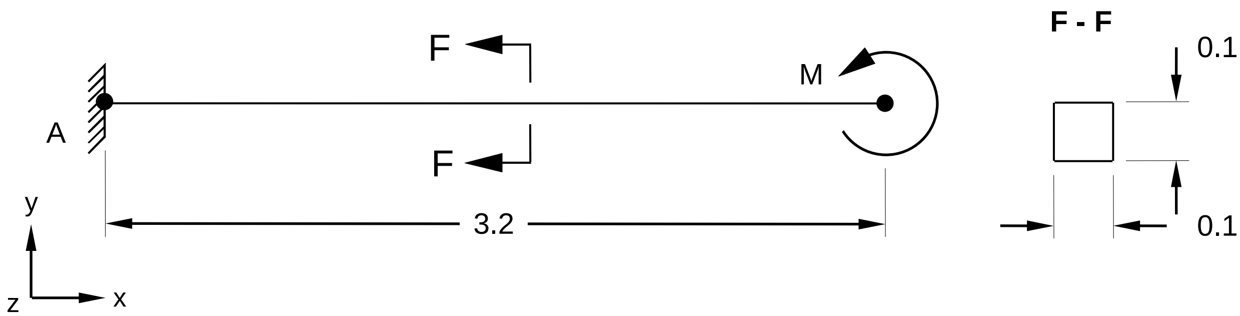


Figure 1. Cantilever Model

Autodesk Inventor Nastran Analysis Model Filename

- vm11_2.nas

Model Data

Finite Element Modeling

- 21 nodes, 20 beam elements

Units

meter/Newton/second

Model Geometry

Length: L = 3.2 m
Width: w = 0.1 m
Thickness: t = 0.1 m

Cross Sectional Properties

Area: A = 0.01 m^2
Moment of Inertia: I = 8.333 E-6 m^4

Material Properties

Young's Modulus: E = 210.0 GPa
Shear Modulus of Elasticity: G = 105.0 GPa
Poisson's Ratio: ν = 0.0

Boundary Conditions

The beam is constrained in all translations and rotations at point A. All other nodes are constrained in Z-translation, X and Y-rotations. A moment M is applied at the free end in increments up to $ML/2\pi EI = 1.0$. Thus the moment $M = 1.0 \times 2 \times \pi \times E \times I / L = 1.0 \times 2 \times \pi \times 210 \times 10^9 \times 8.333 \times 10^{-6} / 3.2 = 3435979.0$

Nonlinear Parameters

Static
Number of Increments: 50

Solution Type

Nonlinear Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Applied Moment $ML/2\pi EI$ (at increment = 0.5)	0.5	0.5	N/A
Applied Moment $ML/2\pi EI$ (at increment = 1.0)	1.0	1.0	N/A
Tip Displacement in X-direction U_x/L (at increment = 0.5)	1.0	1.0	0.0
Tip Displacement in X-direction U_x/L (at increment = 1.0)	1.0	1.0	0.0
Tip Displacement in Y-direction U_y/L (at increment = 0.5)	0.637	0.637	0.0
Tip Displacement in Y-direction U_y/L (at increment = 1.0)	0.0	0.0	0.0
Tip Rotation $\varphi/2\pi$ (at increment = 0.5)	0.5	0.5	0.0
Tip Rotation $\varphi/2\pi$ (at increment = 1.0)	1.0	1.0	0.0

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Non-Linear Behavior of 3D-Beams*. Glasgow: NAFEMS, Publication NNB, Rev. 1, Oct. 1989. Test NL5.

11.3 Lee's Frame Buckling Problem

Problem Description

Figure 1 shows the right angle frame model. A nonlinear static analysis is performed on the model. The displacement at point of loading at different load increments are determined. All dimensions are in meters.

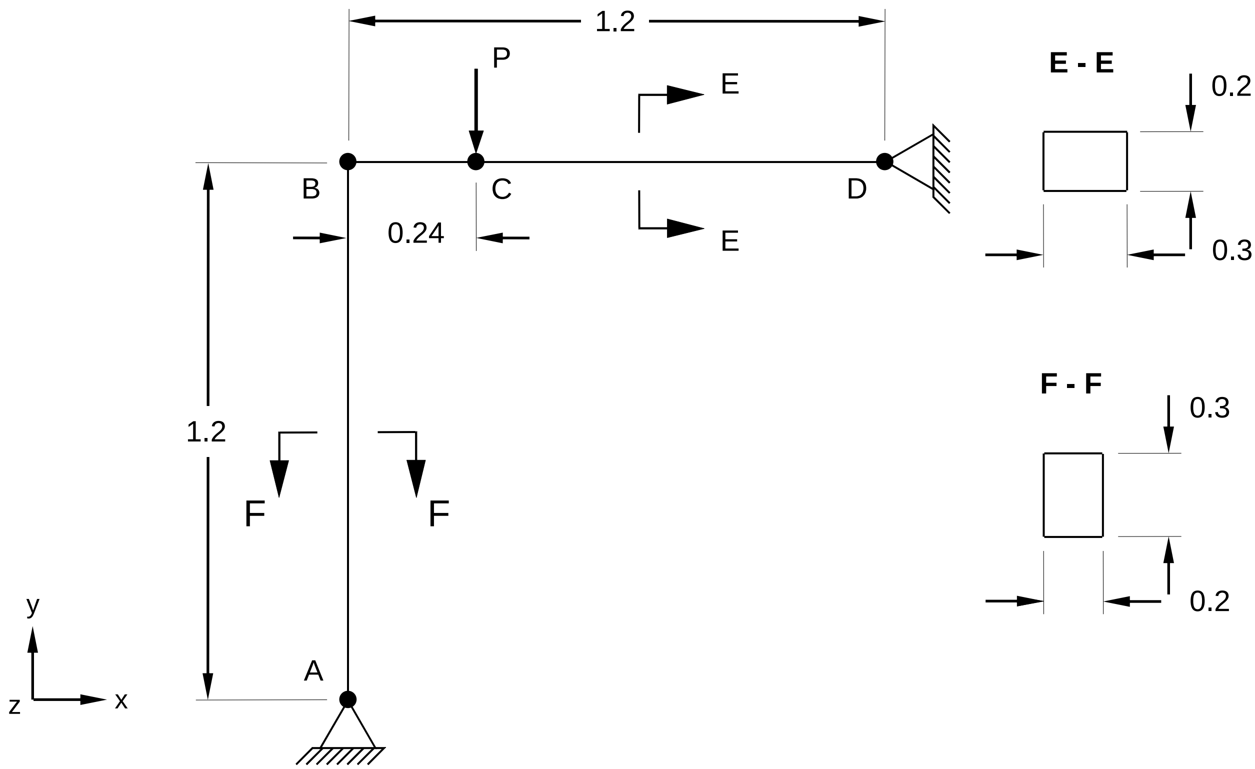


Figure 1. Right Angle Frame Model

Autodesk Inventor Nastran Analysis Model Filename

- vm11_3.nas

Model Data

Finite Element Modeling

- 21 nodes, 20 beam elements

Units

meter/Newton/second

Model Geometry

Length: L = 1.2 m

Cross Sectional Properties

Square Cross Section 1 = (0.03 m x 0.02 m)
Square Cross Section 2 = (0.02 m x 0.03 m)

Material Properties

Young's Modulus: E = 71.74 GPa
Poisson's Ratio: $\nu = 0.3$

Boundary Conditions

Points A and D are constrained in all translations, X and Y-rotations. All other nodes are constrained in the Z-translation, X and Y-rotations. A shear force P is applied at point C in increments up to a maximum value of $PL^2/EI = 31.887$.

Nonlinear Parameters

Static
Number of Increments: 50

Solution Type

Nonlinear Static

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Applied Force PL^2/EI	18.552	18.552	N/A
Applied Force PL^2/EI	31.887	31.887	N/A
Tip Displacement in Y-direction U_y/L	0.407	0.408	0.2
Tip Displacement in Y-direction U_y/L	0.784	0.782	0.3

References

1. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Non-Linear Behavior of 3D-Beams*. Glasgow: NAFEMS, Publication NNB, Rev. 1, Oct. 1989. Test NL5.

12. Nonlinear Dynamic Verification Using Theoretical Solutions

The purpose of these nonlinear dynamic test cases is to verify the functionality of Autodesk Inventor Nastran using theoretical solutions of well-known engineering nonlinear dynamic problems. The test cases are basic in form and have closed-form theoretical solutions.

The theoretical solutions given in these examples are from reputable engineering texts. For each case, a specific reference is cited. All theoretical reference texts are listed in Appendix A.

The finite element method is very broad in nature and is by no means exhausted by the verification tests provided in this manual. These examples, rather, represent basic, common and well-known applications of the finite element method.

For most cases, discrepancies between Autodesk Inventor Nastran computed and theoretical results are minor and can be considered negligible. To produce exact results, for most cases, a larger number of elements would need to be used. Element quantity is chosen to achieve reasonable engineering accuracy in a reasonable amount of time.

12.1 Impact Load on a Rod by a Mass at a Constant Velocity

Problem Description

Figure 1 shows the rod and mass model. A nonlinear transient analysis with impact loading is performed on the model. The strain energy, time to contact and maximum stress on the bar are determined. All dimensions are in inches.

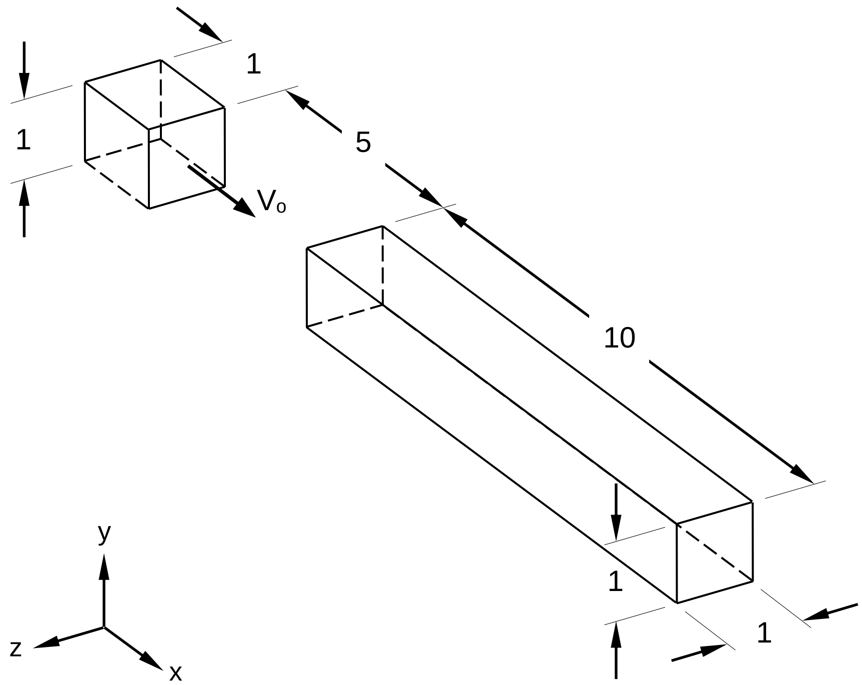


Figure 1. Rod and Mass Model

Autodesk Inventor Nastran Analysis Model Filename

- vm12_1.nas

Model Data

Finite Element Modeling

- 240 nodes, 117 hex solid elements, and 16 gap elements

Units

inch/pound/second

Model Geometry

Length Rod: L_r = 10 in
Length Mass: L_m = 1 in
Distance Mass to Rod: L = 5 in
Width: w = 1 in
Height: h = 1 in

Cross Sectional Properties

Area Rod: $A_r = 1\ in^2$
Area Mass: $A_m = 1\ in^2$
Square Cross Section Rod = (1 in x 1 in)
Square Cross Section Mass = (1 in x 1 in)

Material Properties

Young's Modulus Rod: E_r = 30.0 E+6 psi
Young's Modulus Mass: E_m = 1.0 E+5 psi
Poisson's Ratio: $\nu = 0.3$

Physical Properties

Mass Rod: m_r = 0.0001 lb-sec/in
Mass Moving Body: m_b = 0.2588 lb-sec/in

Boundary Conditions

One end of the beam is constrained in all translations and all rotations. The mass has the Y and Z-translations, and all rotations constrained, to be guided in the X-direction to impact on the beam. An initial velocity V_o = 100 in/sec is applied to the mass.

Nonlinear Parameters

Transient
Subcase 1:
Number of Increments: 50
Time Increment: 0.001

Subcase 2:
Number of Increments: 200
Time Increment: 0.0001

Solution Type

Nonlinear Transient (impact loading)

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Strain Energy Um (in-lb)	1,294	1,366	5.6
Time to Contact (sec)	5E-2	5E-2	0.0
Maximum Stress (psi)	44,057	44,551	1.1

References

1. Beer, F. P. and Johnston, Jr. E. R., *Mechanics of Materials.* New York: McGraw-Hill, Inc., 1981.

12.2 Impact of a Block on a Spring Scale

Problem Description

Figure 1 shows the rod and mass model. A nonlinear transient analysis with impact loading is performed on the model. The maximum deflection of the scale pan, time to contact and total deflection of the block are determined. All dimensions are in inches.

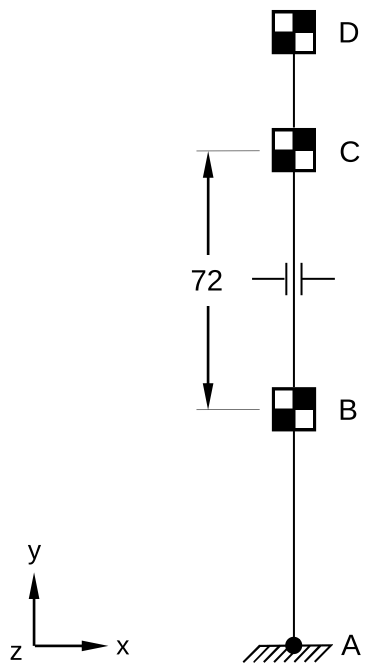


Figure 1. Rod and Mass Model

Autodesk Inventor Nastran Analysis Model Filename

- vm12_2.nas

Model Data

Finite Element Modeling

- 4 nodes, 1 rod element (AB), 3 mass elements, 1 rigid body element (CD), 1 gap element (BC)

Units

inch/pound/second

Model Geometry

Distance Block to Scale: *h* = 72 in

Material Properties

Young's Modulus Rod: E_r = 10.0 E+2 psi
Poisson's Ratio: ν = 0.25
Weight Rod (scale pan): W_p = 25.0 lbs
Weight Falling Body: W_b = 50.0 lbs
Spring Stiffness (scale): k_1 = 100.0 lbs/in

Boundary Conditions

One end of the rod (point A) is constrained in all translations and all rotations. All other nodes are constrained in the X and Z-translations and all rotations. Gravity load is applied in the negative Y-direction (1G acceleration).

Nonlinear Parameters

Transient
Subcase 1:
Number of Increments: 100
Time Increment: 0.006135

Subcase 2:
Number of Increments: 1000
Time Increment: 0.0001

Solution Type

Nonlinear Transient (impact loading)

Comparison of Results

The tabular results are given in Table 1.

Table 1. Results

Description	**Theory**	**Autodesk Inventor Nastran**	**Error** (%)
Deflection of Scale Pan (in)	-7.700	-7.445	3.3
Total Deflection of Block (in)	-79.450	-79.442	0.0
Time to Contact	0.611	0.613	0.3

References

1. Beer, F. P. and Johnston, Jr. E. R., *Vector Mechanics for Engineers.* New York: McGraw-Hill, Inc., 1962.

APPENDIX A – REFERENCES

1. ASME Pressure Vessel and Piping Division, Committee, on Computer Technology, *Program Verification and Qualification Library*, 1972.
2. Bathe, K. J. and Wilson, E. L., *Large Eigenvalue Problems in Dynamic Analysis*. Journal of the Engineering Mechanics Division, ASCE, Vol. 98, No. EM6, Paper 9433, 1972.
3. Beer, F. P. and Johnston, Jr. E. R., *Mechanics of Materials.* New York: McGraw-Hill, Inc., 1981.
4. Beer and Johnston, *Mechanics of Materials*. New York: McGraw-Hill, Inc., 1992.
5. Beer, F. P. and Johnston, Jr. E. R., *Vector Mechanics for Engineers, Statics and Dynamics.* New York: McGraw-Hill, Inc., 1962.
6. Belvins, R., *Formulas For Natural Frequency and Mode Shape*, 1st Edition. New York: VanNostrand Reinhold Co., 1979.
7. Biggs, J. M., *Introduction to Structural Dynamics*. New York: McGraw-Hill Co., 1964.
8. Crandall, S. H., and Dahl, N. C., *An Introduction to the Mechanics of Solids*. New York: McGraw-Hill Book Co., Inc., 1959.
9. Davies, G. A. O., Fenner, R. T., and Lewis, R. W., *Background to Benchmarks*. Glasgow: NAFEMS, 1993.
10. Harris, C. M. and Crede, C. E., *Shock and Vibration Handbook*. New York: McGraw-Hill, Inc., 1976.
11. Harris, C. O., *Introduction to Stress Analysis.* New York: The Macmillan Co., 1959.
12. Hsu, T. H., *Stress and Strain Data Handbook*. Houston, London, Paris, Tokyo: Gulf Publishing Company, Book Division.
13. McCormac, J. C., *Structural Analysis*, 3rd Edition. New York: In Text Educational Publishers, 1975.
14. McNeal, R. H. and Harder, R. C., *A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Element in Analysis and Design 1*, North Holland, 1985.
15. Naeim, F., *The Seismic Design Handbook*. New York: VanNostrand Reinhold Co., 1989.
16. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Forced Vibration*. Glasgow: NAFEMS, Mar., 1993. Test No. 5T.
17. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Geometric Non-Linear Behavior of Shells*. Glasgow: NAFEMS, Publication R0024. Test 3DNLG-1.
18. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Natural Frequency Analysis*. Glasgow: NAFEMS, Nov. 1987.
19. NAFEMS Finite Element Methods & Standards, Abbassian, F., Dawswell, D. J., and Knowles, N. C., *Selected Benchmarks for Non-Linear Behavior of 3D-Beams*. Glasgow: NAFEMS, Publication NNB, Rev. 1, Oct. 1989. Test NL5.
20. NAFEMS Finite Element Methods & Standards, *The Standard NAFEMS Benchmarks*, Rev. 3. Glasgow: NAFEMS, 1990.
21. NAFEMS Report R0031*, Laminated Strip*. Test No. R0031/1, Date issued 17/12/98/1.
22. NAFEMS Report R0031*, Wrapped Thick Cylinder*. Test No. R0031/2, Date issued 17/12/98/1.
23. NASA SP-8007 Buckling of Thin-Walled Circular Cylinders.
24. Paz, M., S*tructural Dynamics; Theory and Computation*. 3rd Edition. New York: D. VanNostrand Co., Inc., 1991.
25. Peterson, F. E., *EASE2, Elastic Analysis for Structural Engineering, Example Problem Manual*, Engineering Analysis Corporation, Berkeley, California, 1981.
26. Roark, R. and Young, W., *Formulas for Stress and Strain*, 5th Edition. New York: McGraw-Hill Co., 1975.

27. Scordelis, A. C. and Lo, K. S., *Computer Analysis of Cylindrical Shells*, Journal of the American Concrete Institute, Vol. 61, May 1964.
28. Seely, F. B. and Smith, J. O., *Advanced Mechanics of Materials*, 2nd Edition. John Wiley and Sons, 1955.
29. Shigley, J. and Mitchel, L., *Mechanical Engineering Design*, 4th Edition. New York: McGraw-Hill Co., 1983.
30. Société Francaise des Mécaniciens, *Guide de validation des progiciels de calcul de structures*. Paris: Afnor Technique, 1990.
31. Thompson, W. T., *Vibration Theory and Applications,* 2nd Edition. New Jersey, Englewood Cliff: Prentice-Hall, Inc., 1965.
32. Timoshenko, S. and Gere, J. M., *Theory of Elastic Stability*, 2nd Edition. New York: McGraw-Hill Book Co., Inc., 1961.
33. Timoshenko, S. and Gere, J. M., *Theory of Elastic Stability*, 3rd Edition. New York: McGraw-Hill Book Co., Inc., 1970.
34. Timoshenko, S. and Young, D. H., *Vibration Problems in Engineering*, 3rd Edition. New York: D. VanNostrand Co., Inc., 1955.
35. Timoshenko, S., *Strength of Materials, Part 1, Elementary Theory and Problems*, 3rd Edition. New York: D. VanNostrand Co., Inc., 1955.
36. Timoshenko, S., *Strength of Materials, Part 2, Elementary Theory and Problems*, 3rd Edition. New York: D. VanNostrand Co., Inc., 1956.
37. Tse, F., Morse, I., and Hinkle, R., *Mechanical Vibrations,* 2nd Edition. Boston: Allyn and Bacon, Inc., 1978.
38. Tse, F., Morse, I., and Hinkle, R., *Mechanical Vibrations, Theory and Applications*. Boston: Allyn and Bacon, Inc., 1978.
39. Young, W. C., *Roark's Formulas for Stress and Strain*, 6th Edition. New York: McGraw-Hill Co., 1989.
40. Warburton, G. B., *The Dynamic Behavior of Structures*. Pergamon Press, 1964.