ENERCALC 3D
Verifications Manual

ENERCALC, INC

ENERCALC 3D

Verifications Manual Build 20

A product of

ENERCALC, INC.

ENERCALC 3D Verifications Manual
Build 20

ENERCALC, Inc.

P.O. Box 2208
Newport Beach, CA 92659
(949) 645-0151
(800) 424-2252

Sales: info@enercalc.com
Support : support@enercalc.com
Web : www.enercalc.com

Publisher

ENERCALC, INC.

Managing Editor

Michael D. Brooks, S.E., P.E.

ENERCALC 3D Verifications Manual
Build 20
February 2021

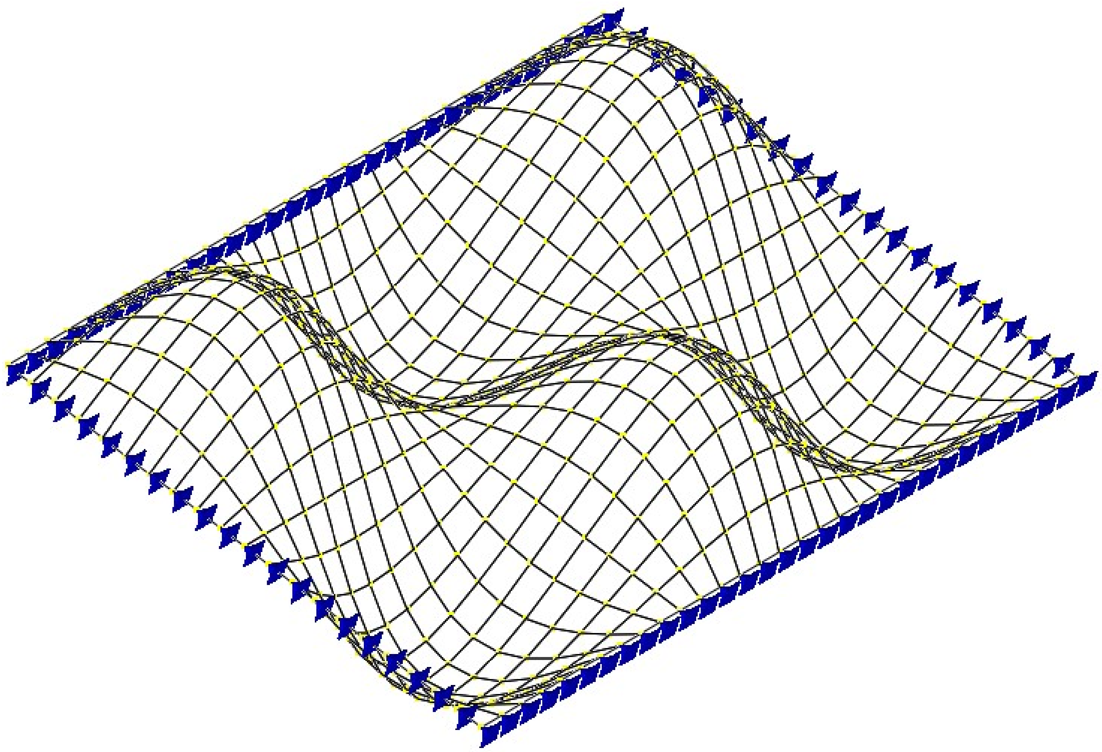
Table of Contents

Part I

1 Introduction

ENERCALC 3D Verifications Manual

Last Revised: 9 February 2021



ENERCALC Inc.

Part II

2 Static - Beam Element

2.1 A-01 (Simple 3d-Truss - Model Type 3D Truss)

Objective

To verify the behavior of the 3d truss element.

Problem Description

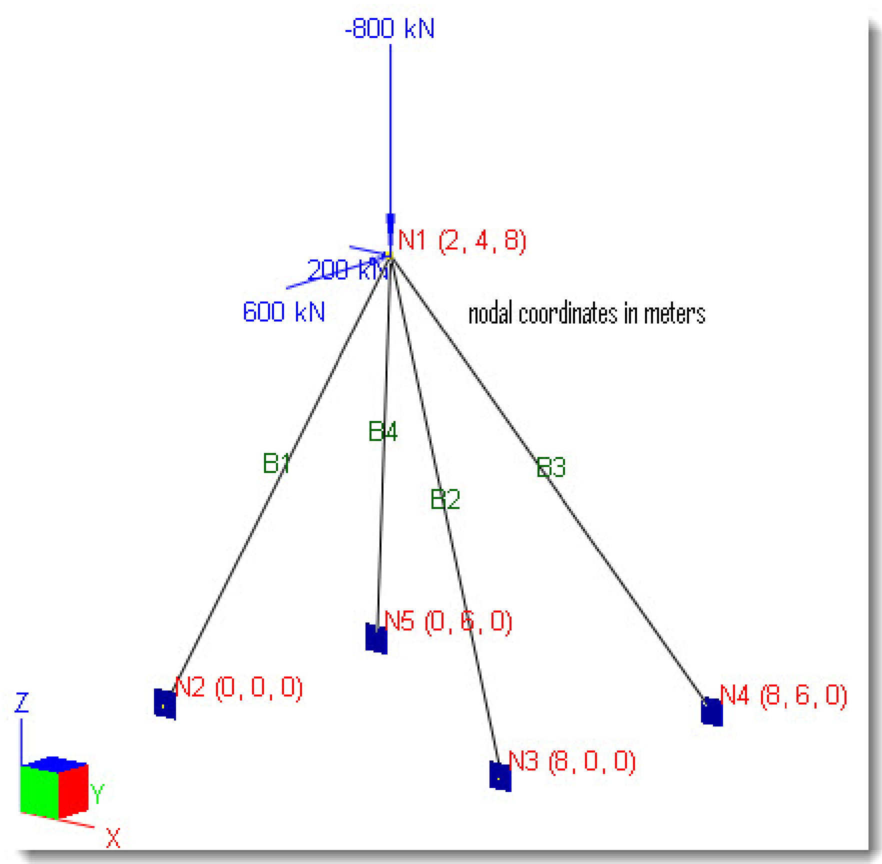
A simple 3d truss is supported and loaded as shown below. Nodal X, Y, and Z coordinates are given in parenthesis.

Material properties: E = 200 KN/mm^2, v = 0.3

Section properties: A_{12} = 2e4 mm^2, A_{13} = 3e4 mm^2, A_{14} = 4e4 mm^2, A_{15} = 3e4 mm^2

All members I_z = 1e10 mm^4, I_y = 1e10 mm^4 , J = 1e10 mm^4

Nodal forces applied at node 1: P_x = 200 KN, P_y = 600 KN, P_z = -800 KN



Finite Element Model

4 beam elements

Model type: 3D Truss

Results

The displacements and support reactions are given in [Ref 1].

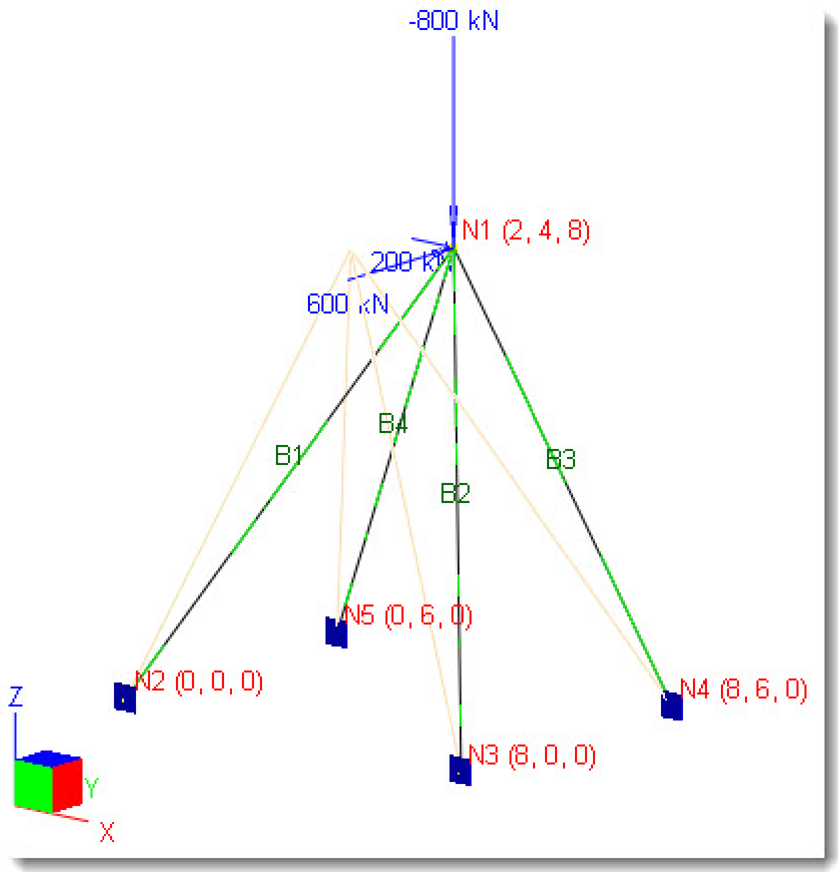
Units: displacement-mm; reaction-KN

	ENERCALC 3D			[Ref 1]		
	X	Y	Z	X	Y	Z
Displacement @ N1	0.1779	2.722	-0.4865	0.1783	2.722	-0.4863
Reactions @ N2	-76.39	-152.78	-305.56	-76.4	-152.8	-305.6
Reactions @ N3	170.83	-113.88	-227.77	170.8	-113.8	-227.7
Reactions @ N4	-470.83	-156.94	627.77	-470.7	-156.9	627.8
Reactions @ N5	176.39	-176.39	705.56	176.3	-176.3	705.5

Comments

The results given by ENERCALC 3D are very close to the referenced values.

The deflection diagram is shown below for illustration purposes.



Deflection Diagram

Reference

[1]. McGuire, Gallagher and Ziemian, “Matrix Structural Analysis” 2nd Edition, pp104, John Wiley & Sons, Inc., 2000

2.2 A-01 (Simple 3d-Truss - Model Type 3D Frame)

Objective

To verify the behavior of the 3d frame element with moment releases

Problem Description

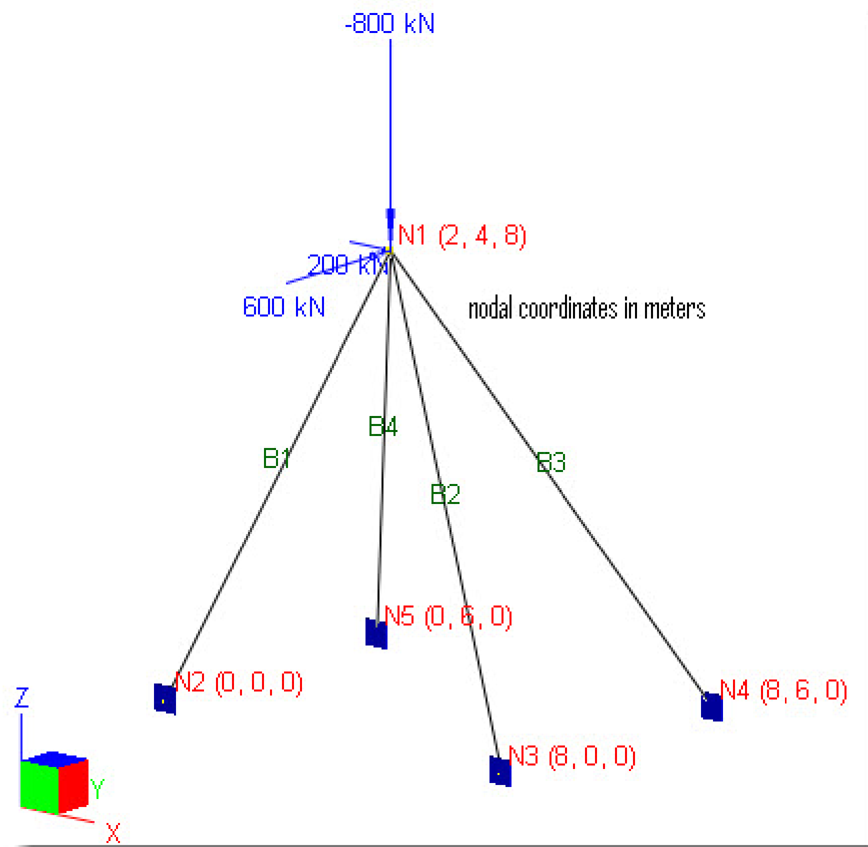
A simple 3d truss is supported and loaded as shown below. Nodal X, Y, and Z coordinates are given in parenthesis.

Material properties: E = 200 KN/mm^2, v = 0.3

Section properties: A_{12} = 2e4 mm^2, A_{13} = 3e4 mm^2, A_{14} = 4e4 mm^2, A_{15} = 3e4 mm^2

All members I_z = 1e10 mm^4, I_y = 1e10 mm^4 , J = 1e10 mm^4

Nodal forces applied at node 1: P_x = 200 KN, P_y = 600 KN, P_z = -800 KN



Finite Element Model

4 beam elements

Model type: 3D Frame & Shell

Moment Releases

The following table shows one way to apply moment releases. Please note that we only apply torsional moment release either (not both) end of a member.

Member Id	Start oz	End oz	Start oy	End oy	Start ox	End ox
1	Released	Released	Released	Released	Released	Not Released
2	Released	Released	Released	Released	Released	Not Released
3	Released	Released	Released	Released	Released	Not Released
4	Released	Released	Released	Released	Not Released	Not Released

Results

The displacements and support reactions are given in [Ref 1].

Units: displacement-mm; reaction-KN

	ENERCALC 3D			[Ref 1]		
	X	Y	Z	X	Y	Z
Displacement @ N1	0.1779	2.722	-0.4865	0.1783	2.722	-0.4863
Reactions @ N2	-76.39	-152.78	-305.56	-76.4	-152.8	-305.6
Reactions @ N3	170.83	-113.88	-227.77	170.8	-113.8	-227.7
Reactions @ N4	-470.83	-156.94	627.77	-470.7	-156.9	627.8
Reactions @ N5	176.39	-176.39	705.56	176.3	-176.3	705.5

Comments

The results given by ENERCALC 3D are very close to the referenced values. The results in this example using 3D Frame & Shell model with moment releases are identical to those in the previous example using 3D Truss model. It is generally easier and more efficient to use 3D Truss model type if your model contains only truss members as the program will automatically suppress global OX, OY and OZ DOFs. On the other hand, 3D Frame and Shell model type (with proper moment releases) should be used if your model contains both truss and frame members.

Reference

[1]. McGuire, Gallagher and Ziemian, “Matrix Structural Analysis” 2nd Edition, pp104, John Wiley & Sons, Inc., 2000

2.3 A-02 (Simple 3d-Beam)

Objective

To verify the behavior of the 3d beam element

Problem Description

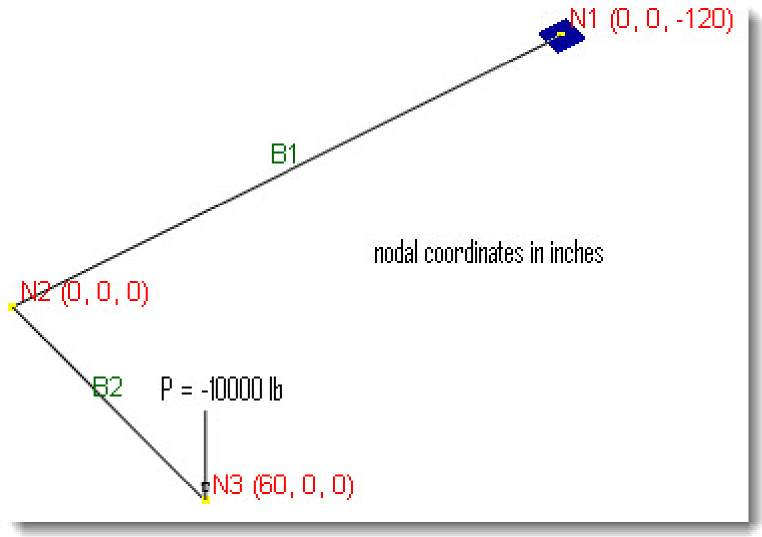
A simple 3d beam of round section is fixed at one end and loaded at the tip of the other end as shown below.

Lengths: L_1 = 120 in, L_2 = 60 in

Material properties: E = 2.9e7 psi, G = 11.15e6, v = 0.3

Section properties: Ix = Iy = 1017.88 in^4, J = 2023.75 in^4, Az = 10 in^2

Tip Force P = 1e4 lb



Finite Element Model

2 beam elements

Model type: 3D Frame & Shell (shear deformation ignored)

Results

The tip vertical displacement Dy at N3 may be calculated as [Ref 1]:

$$D_y = \frac{P}{3EI_x}(L_1^3 + L_2^3) + \frac{P}{GL}(L_1 L_2^2) = -0.4098$$

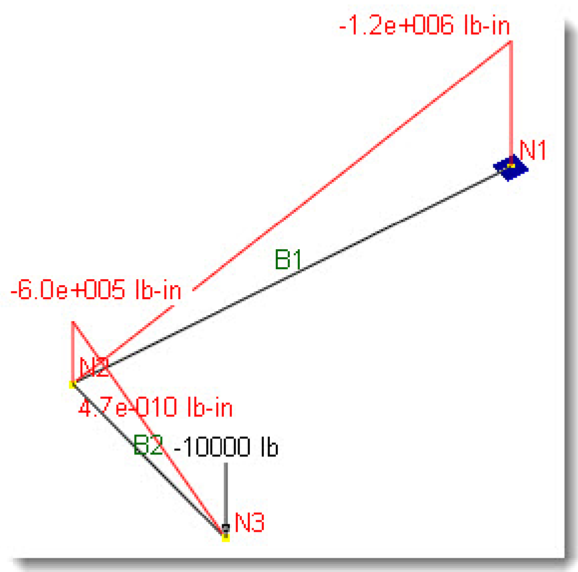
Unit: displacement - in

	ENERCALC 3D	Theoretical
Displacement Dy @ N3	-0.4098	-0.4098

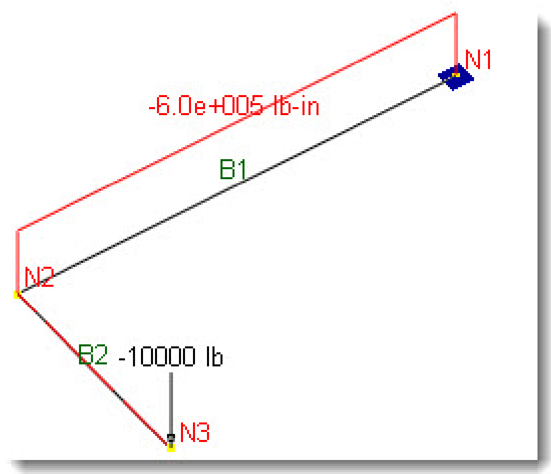
Comments

The results given by ENERCALC 3D are identical to the theoretical values.

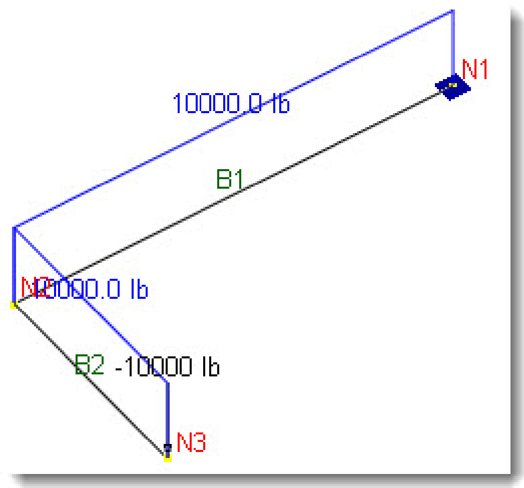
The moment, shear and deflection diagrams are shown below for illustration purposes.



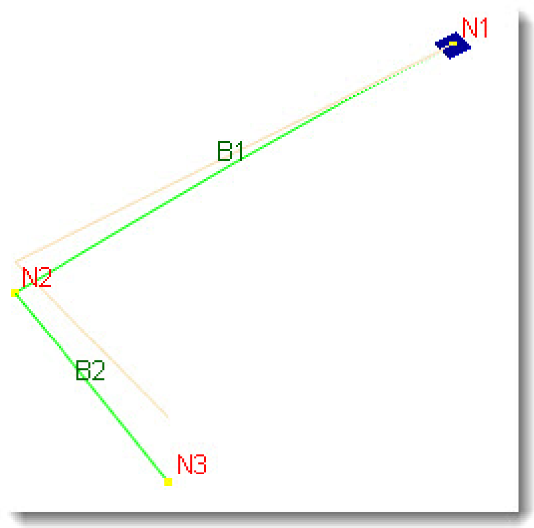
Major Moment Diagram (Mz)



Torsion Diagram (Mx)



Shear Diagram (Vy)



Deflection Diagram

Reference

[1]. Long & Bao, “Structural Mechanics”, pp146, People’s Educational Publishing House, China, 1983.

2.4 A-03 (Beam on Grade)

Objective

To verify the behavior of the line spring

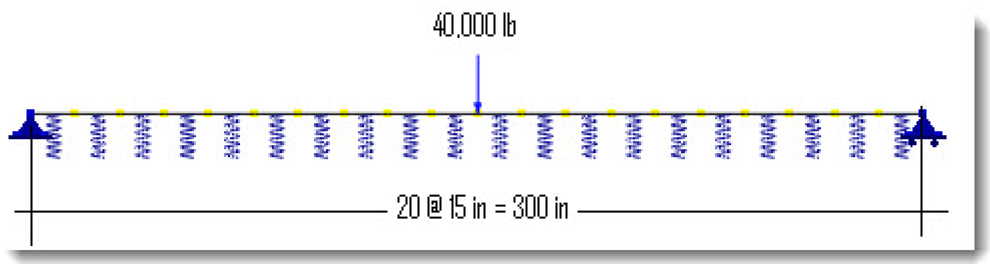
Problem Description

A 300 in beam is supported on an elastic foundation and subjected to a point force of -40,000 lb at the middle as shown below [Ref 1]:

Material properties: E = 29,000 ksi, v = 0.3

Section properties: I_z =125.8 in4, A = 1 in^2

Elastic line spring constant: K_y = 1500 lb / in^2



Finite Element Model

20 beam elements

Model type: 2D Frame

Results

The displacement and moment at the middle of the span are given in [Ref 1].

Units: displacement – in; moment – kip-in

@ middle of the span	ENERCALC 3D	[Ref 1]
Displacement Dy	-0.239	-0.238
Moment Mz	544.44	547

Comments

1. The results given by ENERCALC 3D are very close to the referenced values.

2. Line springs may be replaced by equivalent nodal springs or even truss elements with appropriate section properties as indicated in [Ref 1].

Reference

[1]. McGuire, Gallagher and Ziemian, “Matrix Structural Analysis” 2nd Edition, pp87, John Wiley & Sons, Inc., 2000

2.5 A-04 (P-delta Beam)

Objective

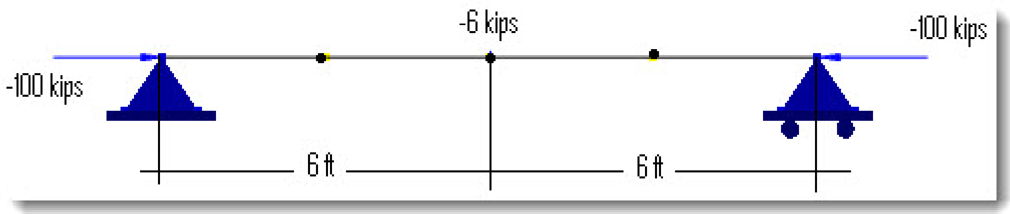
To verify the 2nd-order behavior (P-δ) of beam element

Problem Description

A 12 ft simply supported beam is subjected to a pair of compressive forces of P = -100 kips at the ends and a transverse point force of Q = -6 kips at the middle as shown below [Ref 1].

Material properties: E = 30e6 psi, v = 0.3

Section: 4 x 4 in (I_z = 21.3333 in4, A = 16 in^2)



Finite Element Model

4 beam elements

Model type: 2D Frame (First order and P-Delta)

Results

The displacement and moment at the middle of the beam may be calculated as follows [Ref 1]:

First order: $M_z = \frac{QL}{4} = 18$ kip-ft; $D_y = \frac{QL^3}{48EI} = 0.583$ in

Second order: $u = \frac{L}{2}\sqrt{\frac{P}{EI}} = 0.9$ rad (= 51.57°) $M_z = \frac{QL}{4}\frac{\tan(u)}{u} = 25.2$ kip-ft;

$D_y = \frac{QL}{4P}\left(\frac{\tan(u) - u}{u}\right) = -0.864$ in

Units: displacement – in; moment – kip-in

@ middle of the span	ENERCALC 3D	[Ref 1]
First-order Displacement Dy	-0.5832	-0.583
First-order Moment Mz	18	18
Second-order Displacement Dy	-0.8643	-0.864
Second-order Moment Mz	25.203	25.2

Comments

1. The results given by ENERCALC 3D are very close to the referenced values.

2. In order to capture P- δ behavior that is associated with member curvature, the beam must be split into multiple segments. In this example, we used 4 segments and produced satisfactory results. On the other hand, the splitting is not needed to capture P-Δ behavior that is associated with the lateral translation of the frame members.

Reference

[1]. Leet & Bernal, “Reinforced Concrete Design” 3rd Edition, pp294, McGraw-Hill, 1997

2.6 A-05 (Rotational Spring)

Objective

To verify the behavior of the rotational spring

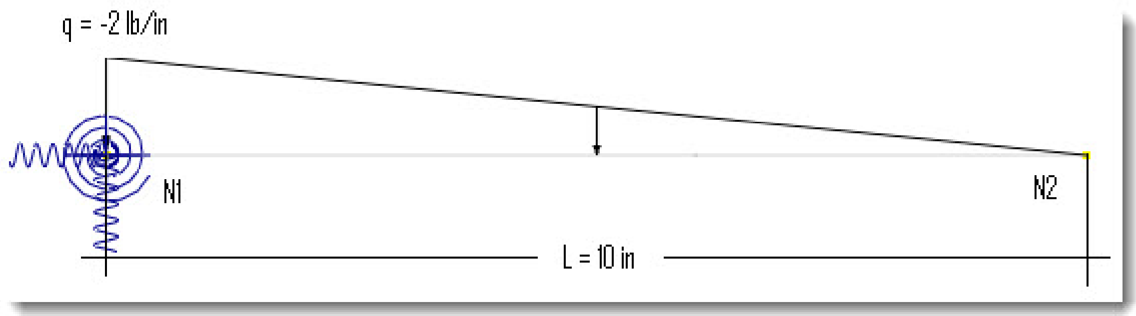
Problem Description

A 10-inch long cantilever beam is subjected to a triangular linear load of q = 2 lb/in.

Material properties: E = 2.9e7 psi, v = 0.3

Section properties: Ix = 1000 in^4, Az = 10 in^2

Boundary condition: rotational spring constant Koz = 1e4 lb-in/rad, Dx and Dy fixed.



Finite Element Model

1 beam element
Assign large spring constants to Kx, Ky to represent fixed DOFs Dx, Dy
Model type: 2D Frame

Results

The rotational displacement Doz at N1 and vertical displacement Dy at N2 may be calculated as:

@N1: $D_{oz} = \dfrac{(0.5qL)*L/3}{K_{oz}} = -3.333e^{-3}$ rad

@N2, $D_z = D_{oz} L = -3.333e^{-2}$ in

Units: displacement – in; rotation - rad

	ENERCALC 3D	Theoretical
Rotation Doz @ N1	-3.333e-3	-3.333e-3
Displacement Dy @ N2	-3.333e-2	-3.333e-2

Comments

The results given by ENERCALC 3D are identical to the theoretical values.

The displacements due to beam strains are negligible.

2.7 A-06 (Non-Prismatic Continuous Beam)

Objective

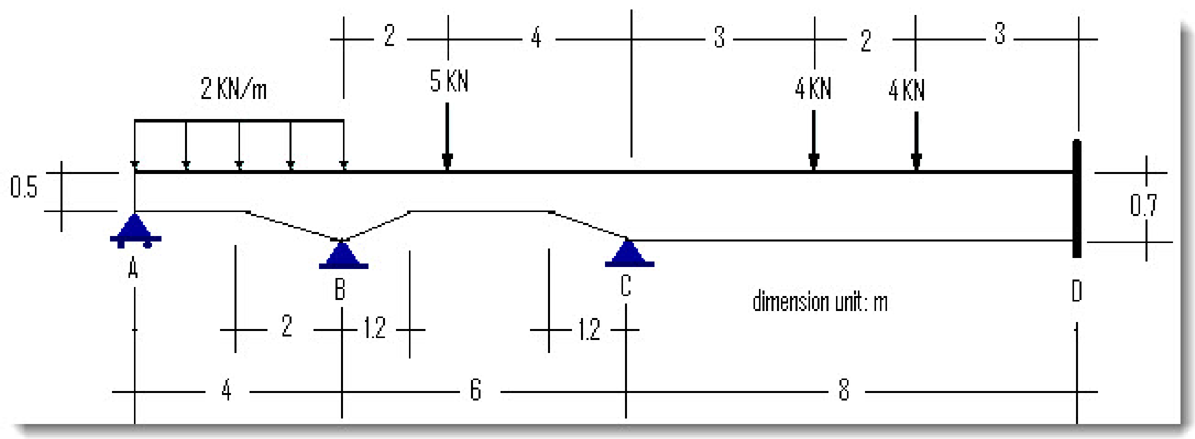
To verify the behavior of a non-prismatic beam

Problem Description

A 3-span non-prismatic continuous beam is fixed at the right end as shown below [Ref 1].

Material properties: E = 1.99948e11 N/m^2, v = 0.3

Section properties: width b = 0.1 m, heights as shown (unit: meter)



Finite Element Model

3 beam elements, then use Create > Templates > Non-Prismatic Members

Model type: 2D Frame (do not consider shear deformation)

Results

The moments at supports given by ENERCALC 3D are compared with those given in [Ref 1].

Unit: moment – KN-m

ENERCALC 3D			[Ref 1]		
Mz @ B	Mz @ C	Mz @ D	Mz @ B	Mz @ C	Mz @ D
-4.39	-4.24	-9.13	-4.28	-4.21	-9.15

Comments

The results given by ENERCALC 3D are close to the referenced values.

In ENERCALC 3D, the non-prismatic beam is approximated by splitting an existing beam into multiple beams (segments) to which different section properties are automatically assigned. The steps to create the model in this problem are as follows:

1. Create 3 (prismatic) beams: AB - 4 m, BC – 6m, CD – 8 m

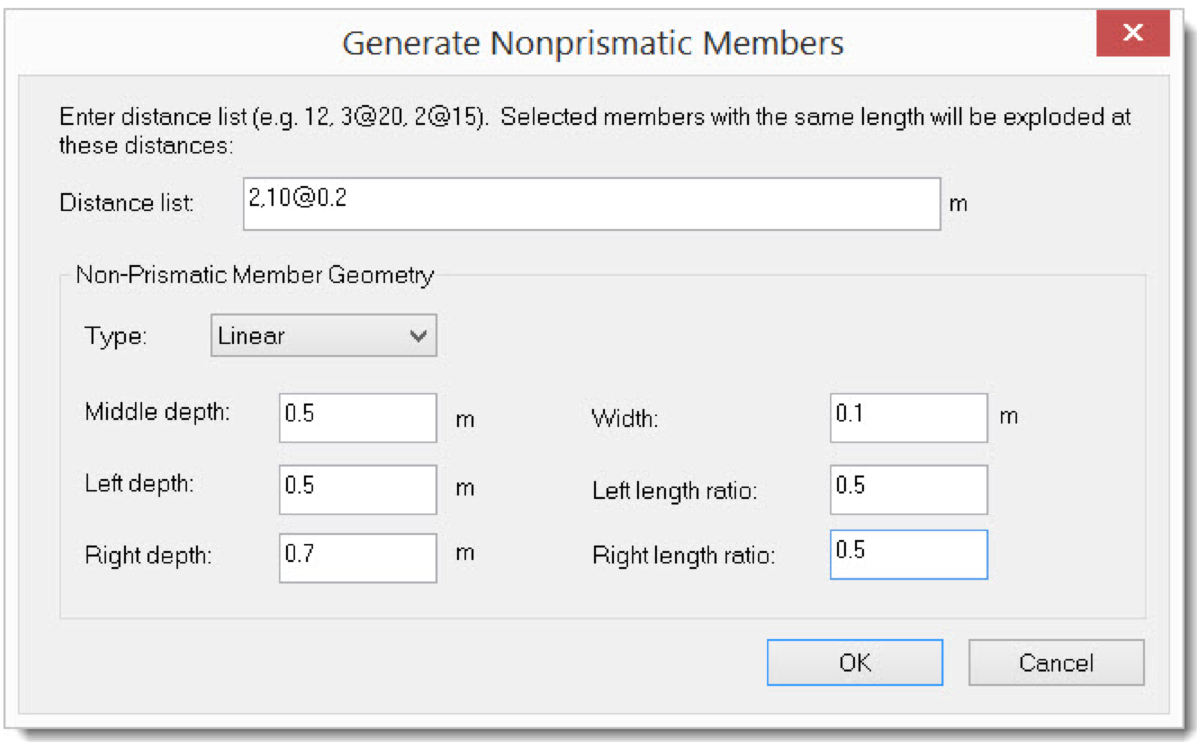
2. Define and assign uniform and point loads on beams

3. Assign supports to A, B, C and D

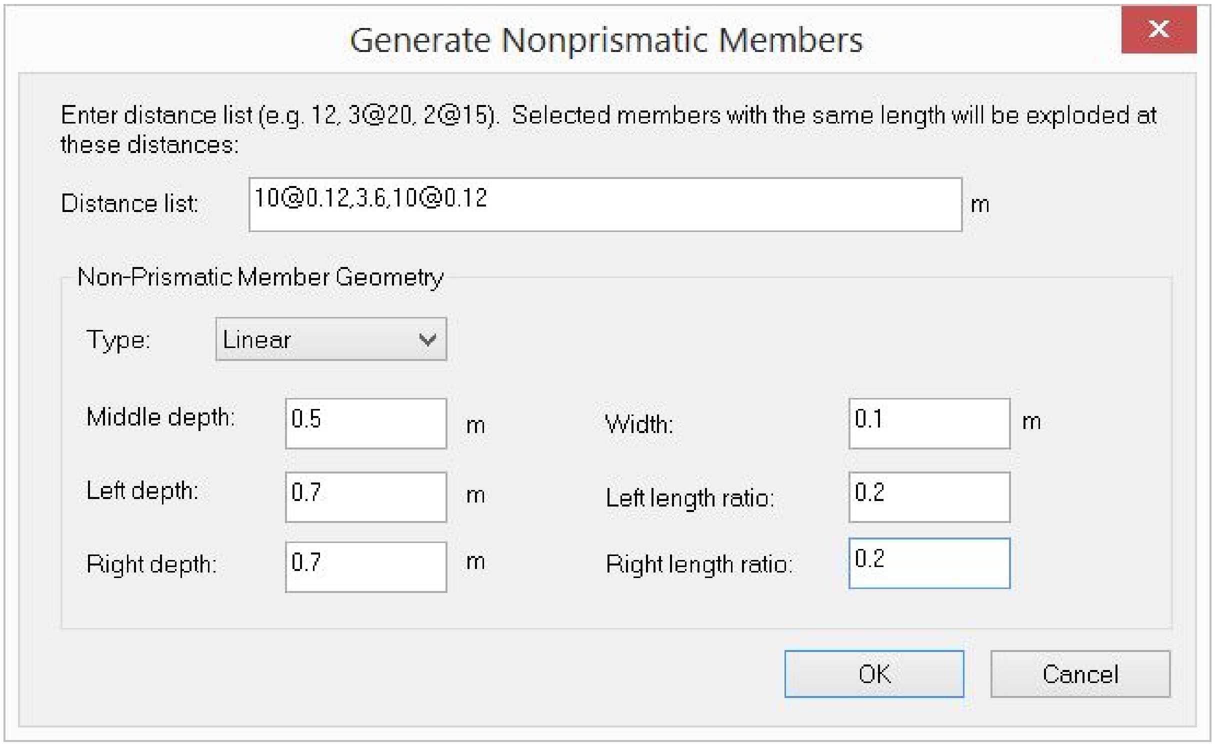
4. Define and assign section to beam CD

5. Define and assign material to beams AB, BC, and CD.

6. Select the beam AB. Run the command Create > Templates > Non-Prismatic Members. Enter the input for "Generate Nonprismatic Members" as follows. The distance list specifies how many beam segments to be used to approximate the non-prismatic beam. In our input, we use one segment for the left 2 m and 10 segments for the right 2 m haunch. More segments could be used to achieve even more accurate result. It should be pointed out that appropriate section properties are assigned to the segmented beams.



7. Select the beam BC. Run the command Create > Templates > Non-Prismatic Members. Enter the input for “Generate Nonprismatic Members” as follows.



Reference

[1]. Lin, Liu, Jiang “Structural Statics Calculation Manual”, pp 232, Building Industry Publishing House of China, 1993

2.8 A-07 (2D Steel Frame)

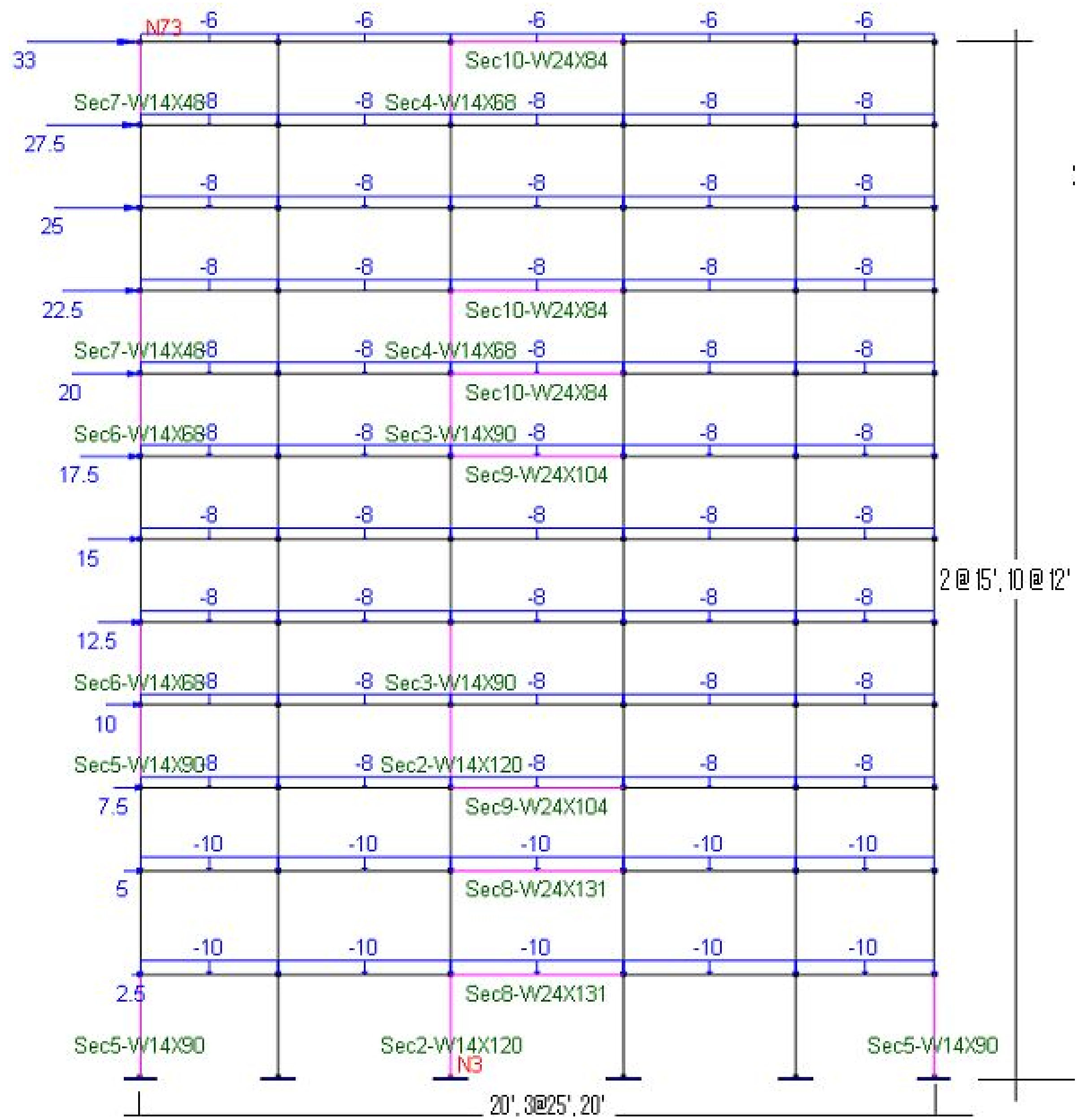
Objective

To verify the behavior of the beam element in a large 2D steel frame

Problem Description

A 5-span, 12-story 2D steel frame is subjected to static lateral and vertical loads as shown below. All beams are W24's and all columns are W14's. The lateral loads are in kips and vertical linear loads are in kip/ft (self weight included).

Material properties: E = 29000 ksi, v = 0.3, density = 483.84 lb/ft^3



Interior columns:

Floor 1 – 4: W14x120
Floor 5 – 8: W14x90
Floor 9 – 12: W14x68

Exterior columns:

Floor 1 – 4: W14x90
Floor 5 – 8: W14x68
Floor 9 – 12: W14x48

Beams:

Floor 1 – 4: W24x131
Floor 5 – 8: W24x104
Floor 9 – 12: W24x84

Units: Iz, Iy and J – in^4, A, Ay and Az – in^2

Section	Iz	Iy	J	A	Ay	Az
W14X120	1380	495	9.37	35.3	8.555	23.03
W14X90	999	362	4.06	26.5	6.16	17.1583
W14X68	722	121	3.01	20	5.81	12
W14X48	484	51.4	1.45	14.1	4.692	7.96308
W24X131	4020	340	9.5	38.5	14.8225	20.64
W24X104	3100	259	4.72	30.6	12.05	16
W24X84	2370	94.4	3.7	24.7	11.327	11.5757

Finite Element Model

132 beam elements

Model type: 2D Frame (shear deformation included)

Results

The displacements and support reactions compared with another program, Frame Analysis & Design (STRAAD) [Ref 1].

Units: displacement-in; reaction force-kips, reaction moment - kip-ft

	ENERCALC 3D		Frame Analysis & Design (STRAAD)	
	First order	Second order	First order	Second order
Dx @ node 73	5.981	7.151	5.9762	7.1347
Rx @ node 3	-36.773	-34.825	-36.7694	-34.8356
Ry @ node 3	2456.514	2457.846	2456.4503	2457.9036
Roz @ node 3	303.299	377.605	303.2664	377.6628

Comments

The results given by ENERCALC 3D are very close to the referenced values.

Reference

[1]. “Frame Analysis & Design”, Digital Canal Corporation, Dubuque, Iowa, USA

2.9 A-08 (A Simple Suspension Bridge)

Objective

To verify the behavior of the beam element with moment releases

Problem Description

A suspension bridge consists of a 60m long beam fixed on both ends and two 25m long truss members which suspend the beam as shown below.

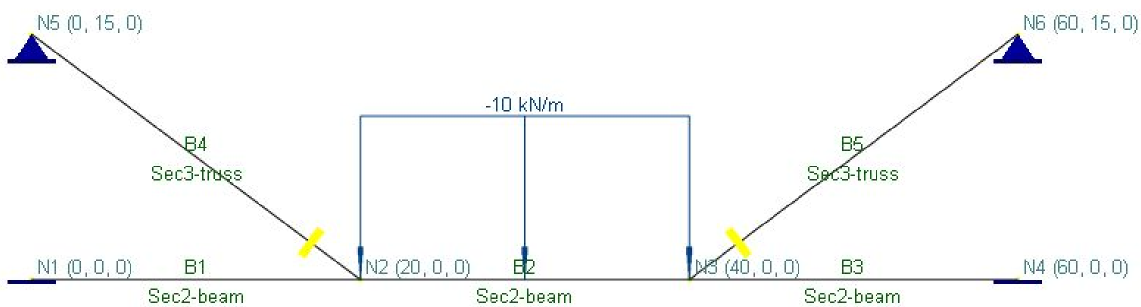
Lengths: shown in parentheses

Material properties: E = 200 kN/mm^2, v = 0.3

Beam Section: Ix = Iy = J= 0.1 m^4, Az = 1.0955 m^2

Truss Section: Ix = Iy = J= 2e-6 m^4, Az = 0.005 m^2

Uniform load on beam: -10 kN/m



Finite Element Model

5 beam elements (moment release at truss ends connecting the beam)

Model type: 2D Frame (shear deformation ignored)

Results

The displacements and internal forces given by ENERCALC 3D are compared with the reference [Ref 1].

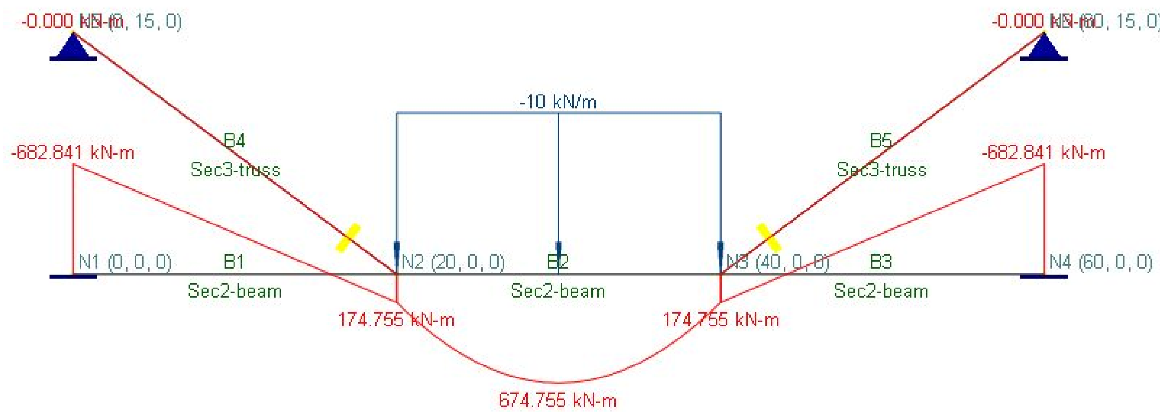
	ENERCALC 3D	Ref 1
Displacement Dy @ N2 (m)	-3.970e-003	79400/(EI) = -3.97e-3
Rotation Doz @ N2 (rad)	-2.540e-004	5080/(EI) = -2.540e-004
Maximum (+) Moment in Beam (kN-m)	674.755	675
Maximum (-) Moment in Beam (kN-m)	-682.840	682
Shear at Beam Ends (kN)	42.88	42.8
Shear at Beam Third Point (kN)	100	100
Maximum Axial Force in Trusses (kN)	95.2	95.2

Comments

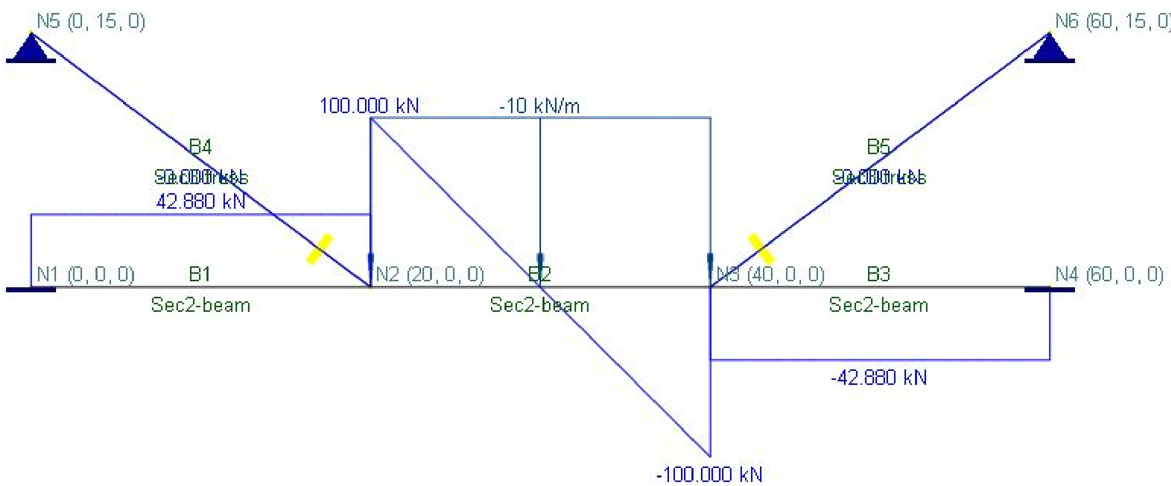
The results given by ENERCALC 3D are very close to the referenced values. Since the model contains both truss and beam members, we used 2D Frame model type and applied moment releases to beam elements for truss members.

The reference gives the relationship of section properties as $(EA)_{truss} = (EI)_{beam}/(20m)$. The properties used in the problem were selected based on this assumption. The beam section area is much greater than the truss section area. Therefore, the axial deformation in the beam is practically ignored.

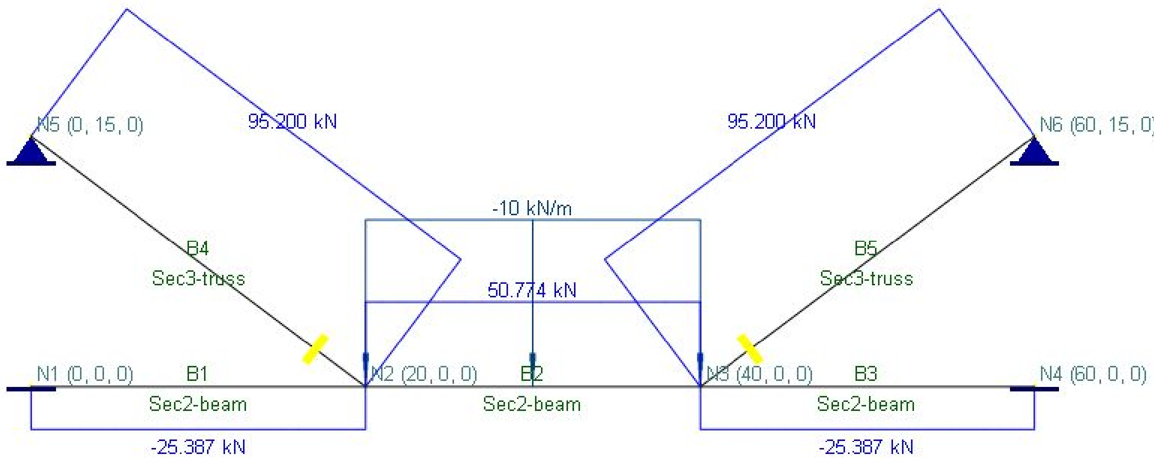
The moment, shear, and axial force diagrams are shown below for illustration purposes.



Major Moment Diagram (Mz)



Shear Diagram (Vy)



Axial Force Diagram (Vx)

Reference

[1]. Long & Bao, “Structural Mechanics”, pp279, People’s Educational Publishing House, China, 1983.

2.10 A-09 (2D Truss with Tension Only Member)

Objective

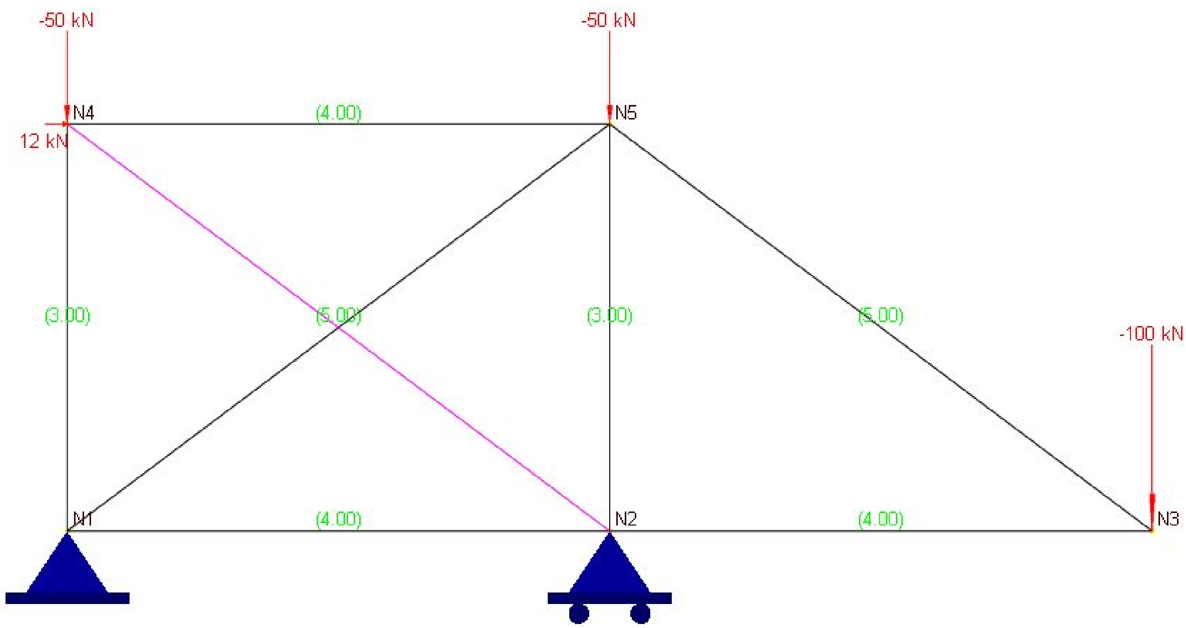
To verify the behavior of the tension only element

Problem Description

A member connecting node 2 and node 4 is tension only in the 2D truss shown below.

Lengths: shown in parentheses in meters
Material properties: E = 205 kN/mm^2, v = 0.3
All Sections: Az = 1500 mm^2

Loads: as shown



Finite Element Model

8 beam elements, with one member connecting N2-N4 being tension only

Model type: 2D Truss

Results

The displacements and internal forces given by ENERCALC 3D are compared with the reference [Ref 1].

	ENERCALC 3D	Ref 1
Displacement Dx @ N3 (mm)	3.469	3.46
Displacement Dy @ N3 (mm)	19.12	19.13
Axial Force in Member connecting N1-N5 (KN)	181.67	181.7

Comments

The results given by ENERCALC 3D are very close to the referenced values. Since the member connecting N2-N4 is tension only but subjected to compression force, its stiffness is ignored in the 2nd iteration during the solution of this nonlinear model. We can also set the member to be inactive to achieve the same effect. The difference between using tension/compression only members and inactive members is that the former requires non-linear solution while the latter does not (unless other nonlinearities such as non-linear springs or P-Delta analysis exist).

Reference

[1]. William M.C. McKenzie, “Examples in Structural Analysis”, pp 125, Taylor & Francis, 2006.

2.11 A-10 (3D Frame with Rigid Diaphragms)

Objective

To verify the behavior of the rigid diaphragms in 3D Frame

Problem Description

A two-story building is subjected two nodal loads in global X direction at the two story level nodes. The X-bay and Z-bay distances are both 18 ft. The story height is 12 ft.

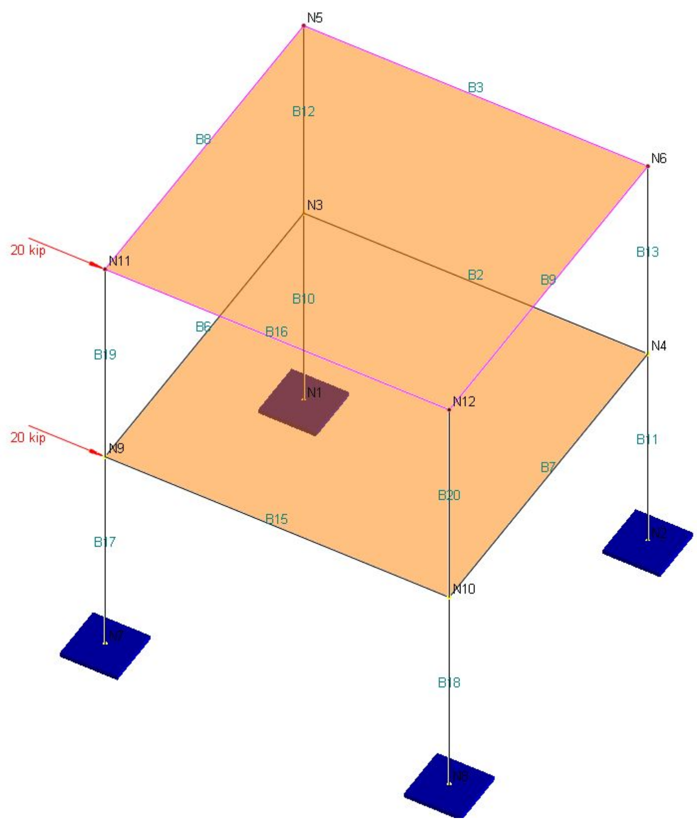
Material properties: E = 3155.92 ksi, v = 0.15

All Sections: rectangular 12x12 in.

$Iyy = Izz = 1728\ in^4$; $J = 2920.32\ in^4$;

$Az = 144\ in^2$; $Ax = Ay = 120\ in^2$

Loads: two 20 kips nodal loads in global X direction as shown.



Finite Element Model

Model type: 3D Frame, with rigid diaphragms defined at two story levels.

Diaphragm stiffness factor: default value (=10000)

Results

The displacements and support reactions given by ENERCALC 3D with and without rigid diaphragm actions are shown below.

Displacements:

Node	Dx (in)	Dy (in)	Dz (in)	Dox	Doy	Doz
Diaphragm actions considered						
5	1.01E+00	4.95E-03	6.70E-01	1.01E-03	6.20E-03	-2.01E-03
6	1.01E+00	-4.95E-03	-6.70E-01	-1.01E-03	6.20E-03	-2.01E-03
11	2.35E+00	4.95E-03	6.70E-01	1.01E-03	6.20E-03	-4.03E-03
12	2.35E+00	-4.95E-03	-6.70E-01	-1.01E-03	6.20E-03	-4.03E-03
Diaphragm actions ignored						
5	8.36E-01	4.17E-03	5.03E-01	8.31E-04	5.99E-03	-1.81E-03
6	8.36E-01	-4.17E-03	-5.03E-01	-8.31E-04	5.97E-03	-1.81E-03
11	2.53E+00	5.72E-03	5.03E-01	8.31E-04	5.99E-03	-4.23E-03
12	2.52E+00	-5.72E-03	-5.03E-01	-8.31E-04	5.97E-03	-4.24E-03

Reactions:

Node	Rx (kip)	Ry (kip)	Rz (kip)	Rox	Roy	Roz
Diaphragm actions considered						
1	-5.42E+00	-1.15E+01	-4.58E+00	-3.37E+01	-7.63E+00	4.30E+01
2	-5.42E+00	1.15E+01	4.58E+00	3.37E+01	-7.63E+00	4.30E+01
7	-1.46E+01	-1.15E+01	-4.58E+00	-3.37E+01	-7.63E+00	1.10E+02
8	-1.46E+01	1.15E+01	4.58E+00	3.37E+01	-7.63E+00	1.10E+02
Diaphragm actions ignored						
1	-3.96E+00	-9.55E+00	-3.12E+00	-2.34E+01	-7.55E+00	3.26E+01
2	-3.96E+00	9.55E+00	3.12E+00	2.34E+01	-7.52E+00	3.26E+01
7	-1.61E+01	-1.34E+01	-3.12E+00	-2.34E+01	-7.55E+00	1.21E+02
8	-1.60E+01	1.34E+01	3.12E+00	2.34E+01	-7.52E+00	1.21E+02

Comments

The diaphragm actions are noticeable in this example. Although the model is subjected to unsymmetrical loads, the nodal rotations about global Y axis are the same for nodes on the diaphragms. This means that the diaphragm stiffness factor, which happens to be the default value 10000, is appropriate for this example. The program is also capable of handle slanting diaphragms. For example, you may rotate the model about Z axis by (-30) degrees. Adjust the local angles for horizontal members as well as the nodal forces accordingly. The Dx for the Node 5, 6, 11 and 12 given by the program are 8.769e-001, 8.720e-001, 2.037e+000 and 2.032e+000 in. The correctness can be verified as the following:

Dx @ node 5: 1.01cos30 + 0.00495sin30 = 0.877

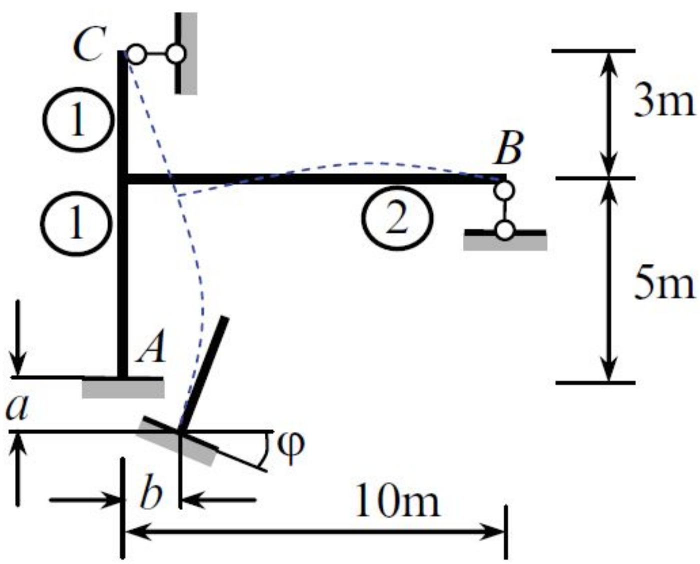
2.12 A-11 (2D Frame with Support Settlements)

Objective

To verify the behavior of the forced translational and rotational displacements

Problem Description

A frame [Ref 1] is clamped at point A and rolled at points B and C as shown below. The relative flexural stiffness of each element is shown in a circle. No external load is applied to the frame, but the frame is subjected to settlement of fixed support A. Assume that the vertical, horizontal, and angular settlements are a = 2 cm, b = 1 cm, and φ = 0.01 rad, respectively.



Finite Element Model

Model type: 2D Frame, without considering frame shear deformation.

The reference does not specify material and section specifically, so we will use steel and rectangular sections (100 mm x 100 mm for vertical members and 200 mm x 100 mm for horizontal member, which satisfy the relative flexural stiffness of members).

E = 200 kN/mm^2, I = 8.33333e+006 mm^4

Results

The support reactions given by ENERCALC 3D are shown below.

	ENERCALC 3D	Ref 1
Reaction Moz @ A	7.57 kN-m	C1 * EI = 4.5418 * 10^{-3} * (1/m) * 200 kN/mm^2 * 8.333 * 10^6 mm^4 = 7.5667 kN-m
Reaction Ry @ B	-0.19 kN	C2 * EI = -1.119 * 10^{-4} * m^2 * 200 kN/mm^2 * 8.333 * 10^6 mm^4 = -0.1865 kN
Reaction Rx @ C	-1.18 kN	C3 * EI = -7.076 * 10^{-4} * m^2 * 200 kN/mm^2 * 8.333 * 10^6 mm^4 = -1.179 kN

Note: From [Ref 1]

C1 = 4.5418 * 10^{-3} (unit: 1/m)

C2 = -1.119 * 10^{-4} (unit: 1/m^2)

C3 = -7.076 * 10^{-4} (unit: 1/m^2)

Comments

The results given by ENERCALC 3D are very close to the referenced values.

Reference

[1]. Igor A. Karnovsky, Olga Lebed, “Advanced Methods of Structural Analysis”, pp 248, Springer Science+Business Media, LLC, 2010.

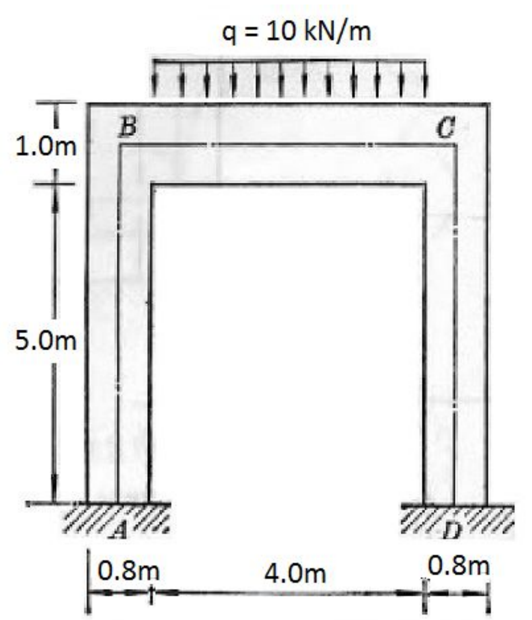
2.13 A-12 (2D Frame with Rigid Offsets)

Objective

To verify the behavior of the rigid offsets

Problem Description

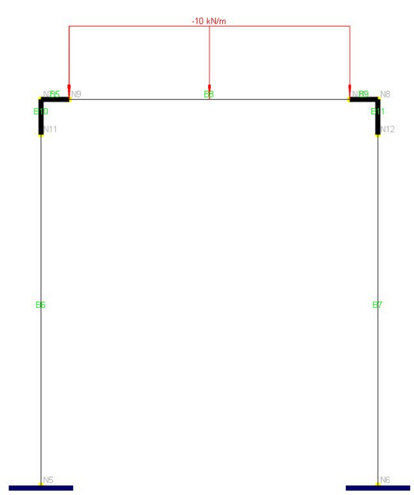
A portal frame [Ref 1] is clamped at point A and D as shown below. The columns are of rectangular size 200mm x 800mm. The beam is of rectangular size 200mm x 1000 mm. The beam is subjected to an uniform load 10 kN/m.



Finite Element Model

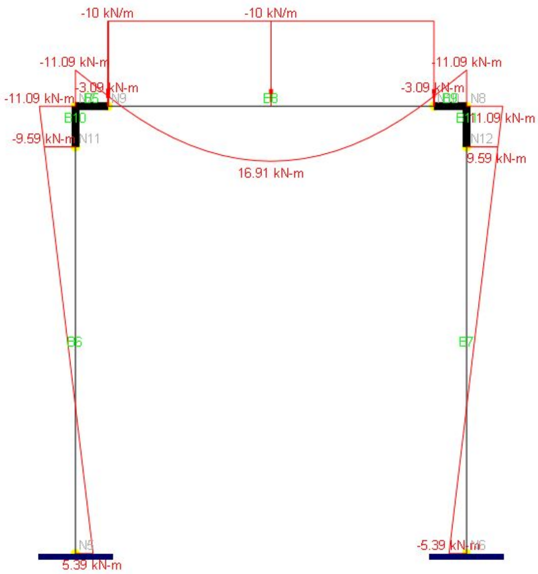
Model type: 2D Frame, without considering frame shear deformation.

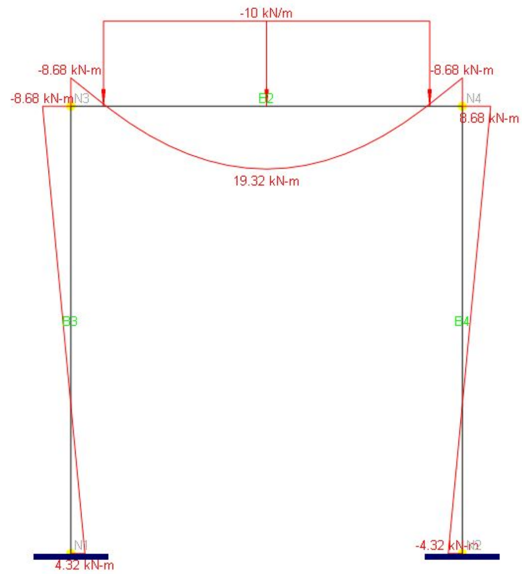
The reference does not specify material specifically, so we will use concrete with fc = 3ksi.



Results

The following are the moment diagrams for modeling the structure with rigid offsets and without rigid offsets.





	ENERCALC 3D With Rigid Offsets	Ref 1
Beam Max Negative Moment	11.09 kN-m	11.23 kN-m
Beam Max Positive Moment	16.91 kN-m	16.9 kN-m
Moment Reactions at Supports	5.39 kN-m	5.43 kN-m

Comments

The results given by ENERCALC 3D are very close to the referenced values. The reference computes moment diagram manually using displacement method. There are noticeable differences between results with rigid offsets and those without.

Reference

[1]. Long & Bao, “Structural Mechanics”, pp296, People’s Educational Publishing House, China, 1983.

2.14 A-13 (2D Truss with an Inclined Roller)

Objective

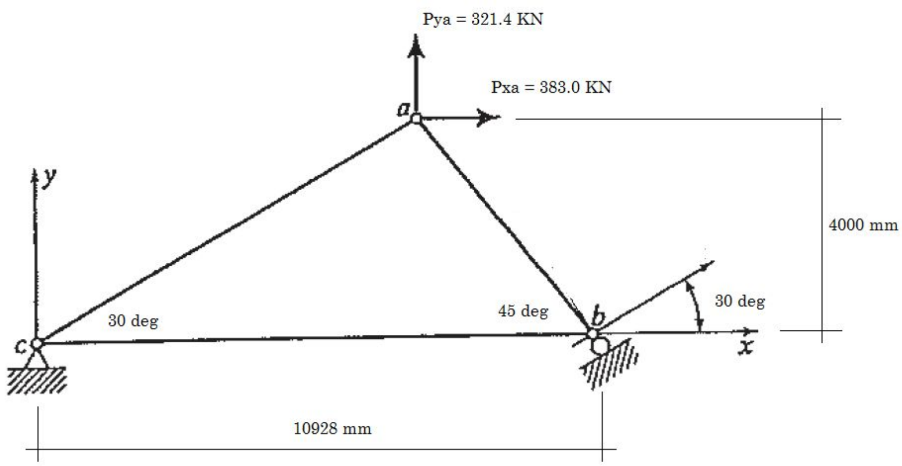
To verify the behavior of inclined roller using multi-DOF constraint

Problem Description

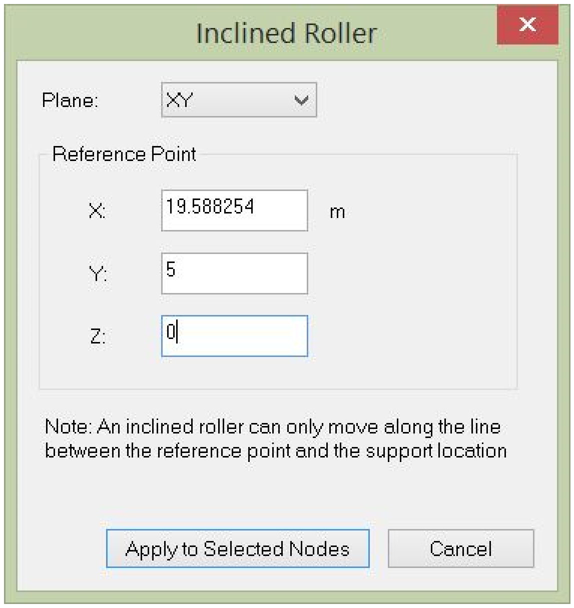
A truss [Ref 1] is supported by a pinned support at point c and a roller (inclined at 30 degrees from horizontal line) at point b as shown below.

Sections: ab = 20,000 mm^2, ac = 15,000 mm^2, bc = 18,000 mm^2

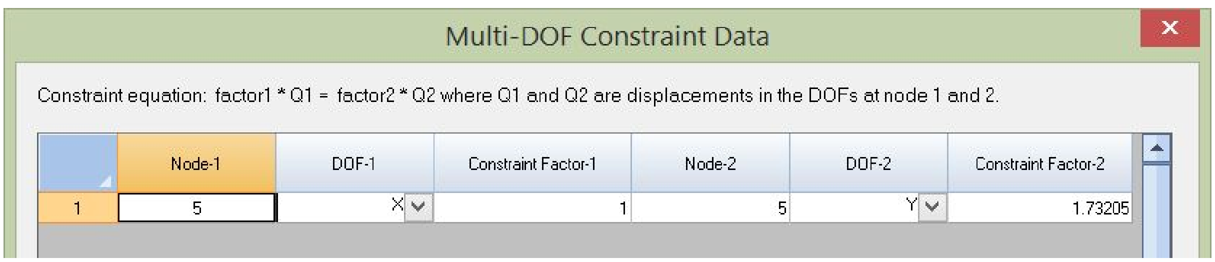
Material: E = 200 MPa



When creating the inclined roller, we can set any point along the roller angle line as the reference point. For example, if the coordinate at point b is (10.928, 0, 0), then we can set the reference point as (10.928 + 10 * cos30, 10 * sin30, 0) = (19.588254, 5, 0).



This effectively creates a multi-DOF constraint as the following:



Results

The following are the displacements and support reactions given by ENERCALC 3D and [Ref 1]. The reaction resultant @ b is calculated by hand as following:

$R = \dfrac{383 \times 4 - 321.4 \times \left(\dfrac{4}{\tan(30)}\right)}{10.928 \times \cos(30)} = -73.4$ kN (pointing to bottom-right). $R_x = 73.4 \times \sin(30) = 36.7$ kN

$R_y = -73.4 \times \cos(30) = -63.57$ kN

	ENERCALC 3D	Ref 1
Displacement Dx @ a	0.9282 mm	0.928 mm
Displacement Dy @ a	1.142 mm	1.143 mm
sqrt(Dx * Dx + Dy * Dy) @ b	0.09416 mm	0.094 mm
Reaction Rx @ c	-419.70 kN	-419.7 kN (by hand)
Reaction Ry @ c	-257.83 kN	-257.83 kN (by hand)
Reaction Rx @ b	36.70 kN (constrained force)	36.70 kN (by hand)
Reaction Ry @ b	-63.57 kN (constrained force)	-63.57 kN (by hand)

Comments

The displacement results given by ENERCALC 3D are very close to the referenced values. The support reactions are not given in Ref 1 but can be easily calculated by hand, which match exactly with those given by ENERCALC 3D.

Reference

[1]. W. McGuire & R.H. Gallagher & R.D. Ziemian, “Matrix Structural Analysis” pp 390, 2nd ed., John Wiley & Sons, Inc., 2000

2.15 A-14 (2D Truss with Thermal Load)

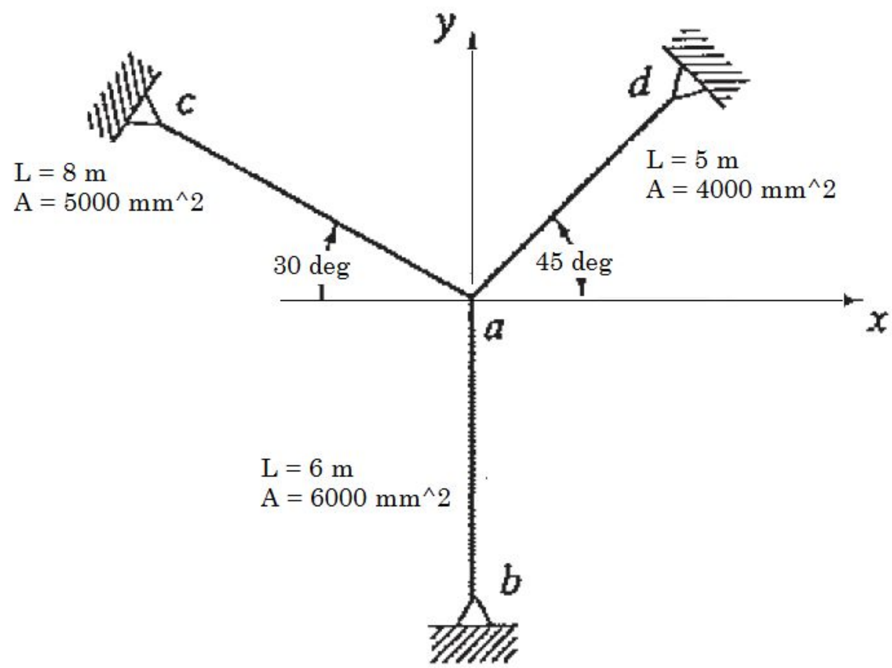
Objective

To verify the behavior of thermal load

Problem Description

In the truss [Ref 1] below, all bars are cooled by 20 degrees Celsius.

Material: E = 200 MPa, thermal coefficient α = 1.2e-5 mm/mm per degree Celsius



Results

The following are the results given by ENERCALC 3D and [Ref 1].

	ENERCALC 3D	Ref 1
Displacement Dx @ a	-4.044e-01 mm	-0.4045 mm
Displacement Dy @ a	-6.995e-02 mm	-0.0698 mm
Reaction Rx @ b	0 kN	0 kN
Reaction Ry @ b	-274.01 kN	-274 kN
Reaction Rx @ c	-173.71 kN	-173.8 kN
Reaction Ry @ c	100.29 kN	100.2 kN
Reaction Rx @ d	173.71 kN	173.8 kN
Reaction Ry @ d	173.71 kN	173.8 kN

Comments

The displacement results given by ENERCALC 3D are very close to the referenced values.

Reference

[1]. W. McGuire & R.H. Gallagher & R.D. Ziemian, “Matrix Structural Analysis” pp 127, 2nd ed., John Wiley & Sons, Inc., 2000

2.16 A-15 (Multi-DOF Constraints - Cyclically Symmetric Frame)

Objective

To verify the multi-DOF constraints to enforce cyclic symmetry

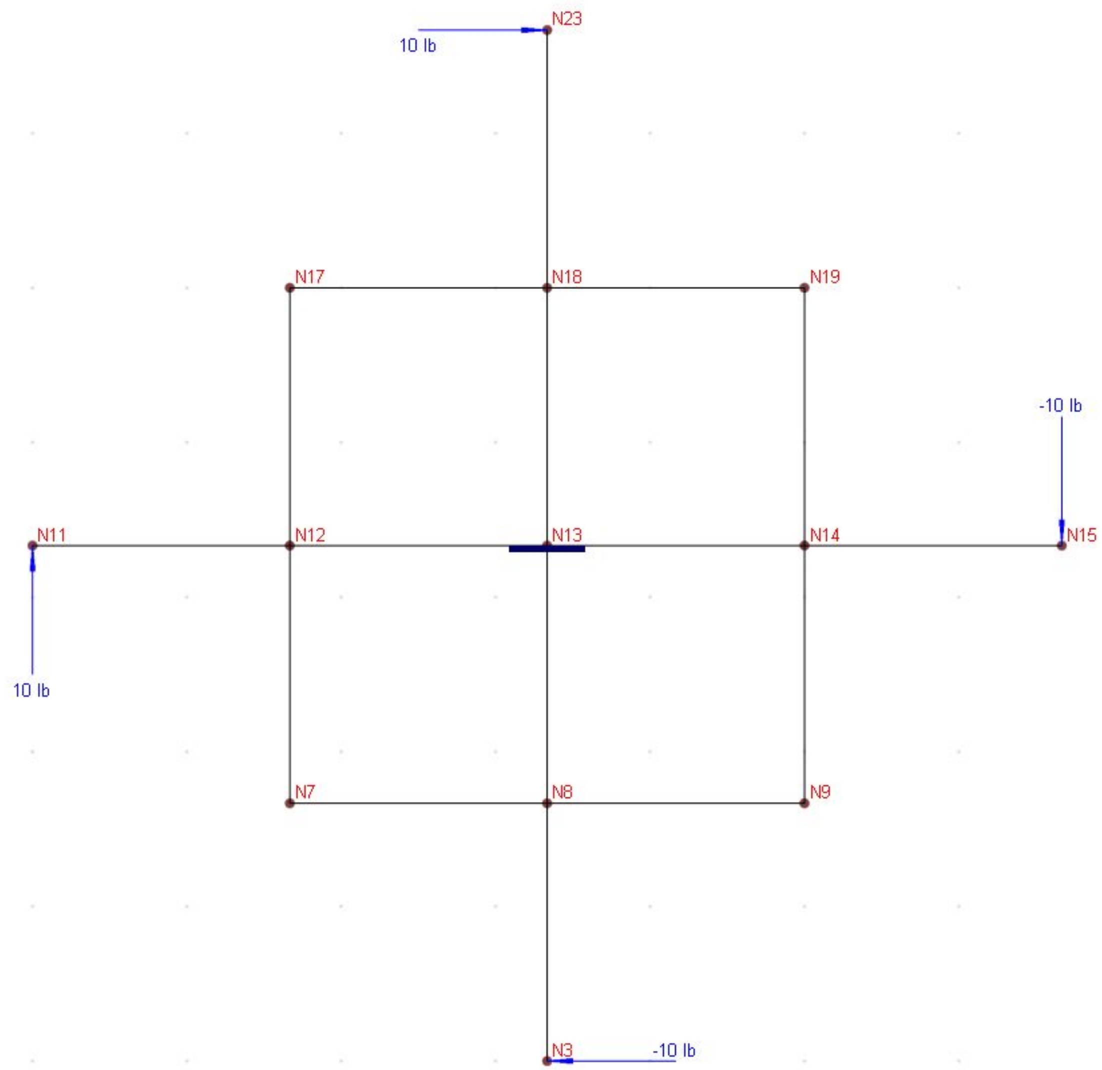
Problem Description

In the frame [Ref 1] below, each of the 16 members is 10 inch long.

Material: E = 1.2e7 psi, v = 0.15.
Sections: A = 1.0 in^2, Iyy = Izz = 8.33e-2 in^4

Four cyclic loads: P = 10 lb

Boundary condition: Fixed at the center node (N13)



Finite Element Model

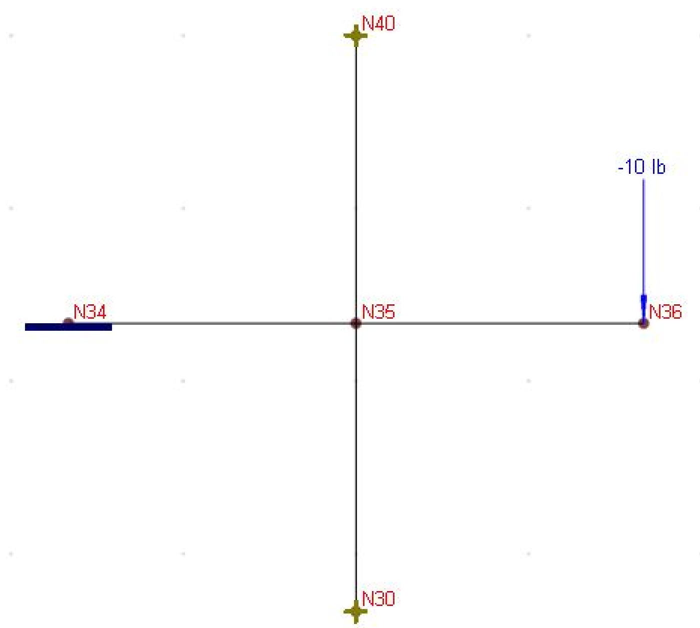
To take advantage of the cyclic symmetry, we are going to model only one quarter of the structure (4-element model) with the following multi-DOF constraints at node 40 and node 30.

$X_{40} = -Y_{30}$

$Y_{40} = X_{30}$

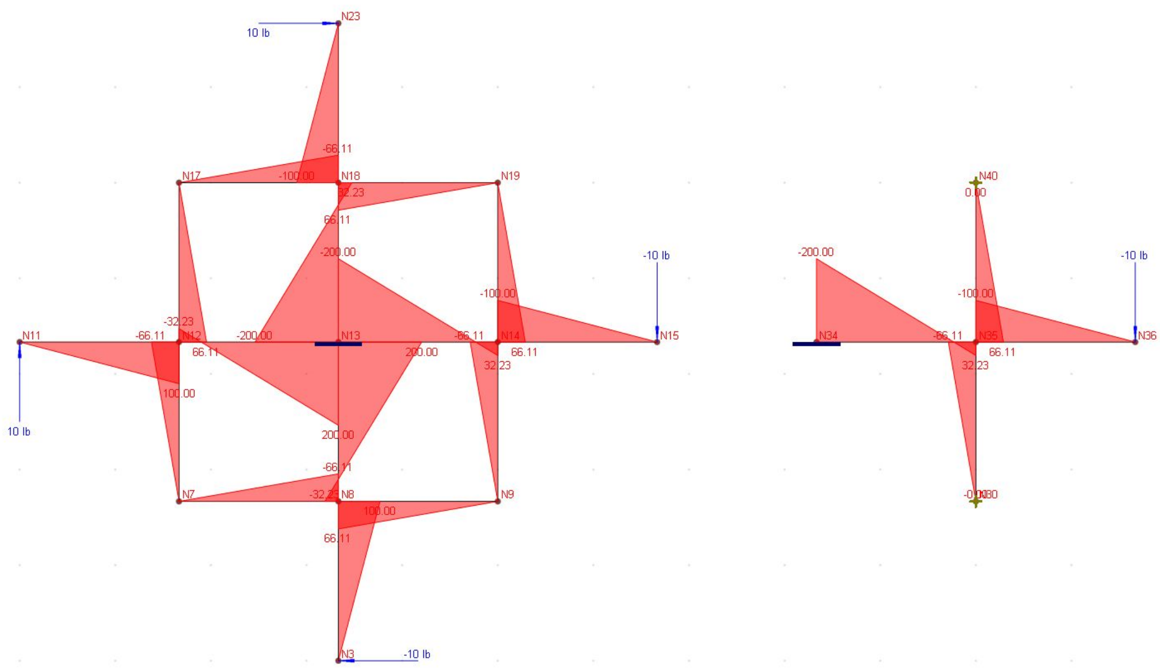
$Oz_{40} = Oz_{30}$

We can use Geometry->Multi-DOF Constraints->Generic Constraints menu to define these three displacement constraints. Alternatively, we can directly enter the constraints in a spreadsheet from Input Data->Multi-DOF Constraints menu.



Results

To illustrate, the following shows the identical Mz moment diagrams for both 16-element model and 4-element model.



Reference

[1]. ADINA Verification Manual, ADINA R & D Inc., Example A.40, June 2001

2.17 A-16 (Coupled Spring)

Objective

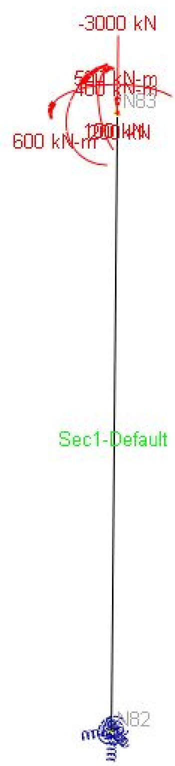
To verify the behavior of coupled spring which is useful in modeling bridge foundations.

Problem Description

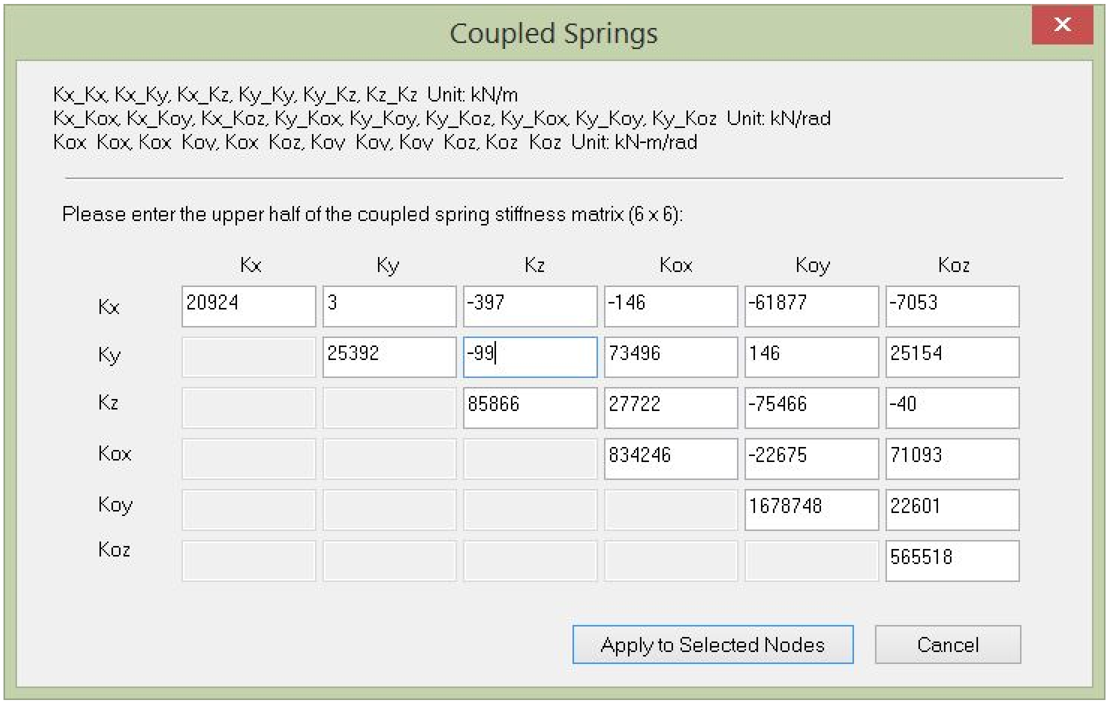
In the 10 meter column [Ref 1] below, the top is subjected to the loads: Fx = 100.00 (kN), Fy = 200.00 (kN), Fz = -3000.00 (kN), Mx = 400.00 (kN-m), My = 500.00 (kN-m) and Mz = 600.00 (kN-m).

Material: E = 3.25e+07 kN/ m^2, v = 0.20.

Sections: Izz = 0.0104 m^4 , Iyy = 0.0417 m^4 , J = 0.0286 m^4 , A= 0.5 m^2 , Ay = 0.4167 m^2 , Az = 0.4167 m^2



The bottom of the column is supported by a coupled spring with the following stiffness matrix terms (see “Calculation of Coupled Spring Stiffness Terms” below)



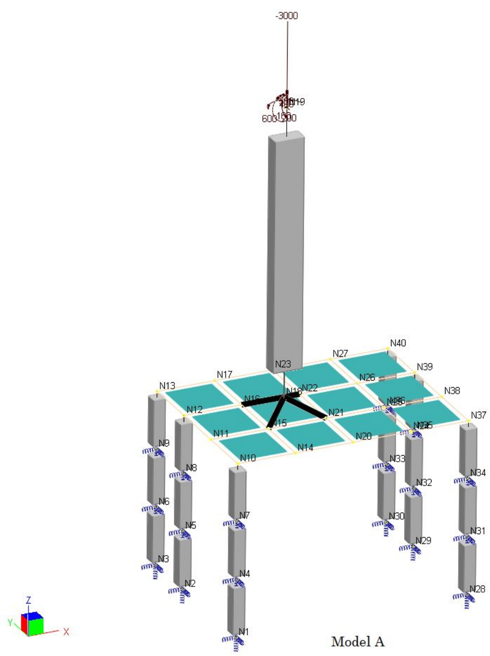
Calculation of Coupled Spring Stiffness Terms

The stiffness matrix terms of the coupled spring used in this example are calculated based on the following simplified bridge piers below. On the left is the full pier (column + foundation) model A while on the right is the foundation-only model B. In order to compute the stiffness matrix of the coupled spring, 6 loads in separate load cases (1000 kN for Px, Py and Pz; 1000 kN-m for Mx, My and Mz) are applied at the bottom of the column in Model B. We first solve the model B to obtain displacement matrix (displacements for each of these load cases).

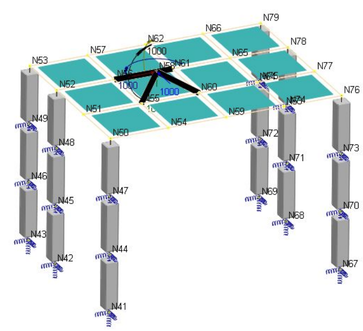
Displacements matrix, m, rad						
Px	0.054200	-0.000621	0.002081	-0.000001	0.002083	0.000621
Py	-0.000621	0.054930	0.001592	-0.004734	0.000005	-0.001856
Pz	0.002081	0.001592	0.012380	-0.000534	0.000626	-0.000002
Mx	-0.000001	-0.004734	-0.000534	0.001633	-0.000002	0.000005
My	0.002083	0.000005	0.000626	-0.000002	0.000701	-0.000002
Mz	0.000621	-0.001856	-0.000002	0.000005	-0.000002	0.001858

We then invert the displacement matrix to obtain the stiffness matrix. Note the stiffness matrix is multiplied by 1000 so the stiffness terms are in the right units as shown below.

Stiffness matrix, kN/m, kN/rad, kN-m/rad						
Px	20924	3	-397	-146	-61877	-7053
Py	3	25392	-99	73496	146	25154
Pz	-397	-99	85866	27722	-75466	-40
Mx	-146	73496	27722	834246	-22675	71093
My	-61877	146	-75466	-22675	1678748	22601
Mz	-7053	25154	-40	71093	22601	565518



Model A



Model B

Results

The nodal displacements in the coupled spring model are very close to those obtained from the full model A. This illustrates that a coupled spring can be used to simplify the modeling of a bridge sub-structures effectively.

	Model with a Coupled Spring			Full Model A		
	X	Y	Z	X	Y	Z
Displacement @ Top (m)	3.961e-02	1.706e-01	-3.667e-02	3.961e-02	1.706e-01	-3.668e-02
Rotation @ Top (rad)	-1.971e-02	6.763e-03	1.629e-02	-1.971e-02	6.763e-03	1.629e-02
Displacement @ Bottom (m)	2.551e-03	1.261e-02	-3.482e-02	2.554e-03	1.261e-02	-3.483e-02
Rotation @ Bottom (rad)	-1.957e-03	-6.161e-04	7.998e-04	-1.957e-03	-6.159e-04	7.997e-04

Part III

3 Static - Shell Element (Bending)

3.1 B-01 (Plate Patch Test)

Objective

To verify the plate (MITC4 thick plate formulation) passes the patch test

Problem Description

A plate of size 0.12 x 0.24 in is subjected to forced displacements at the four corners as shown below. The boundary conditions are:

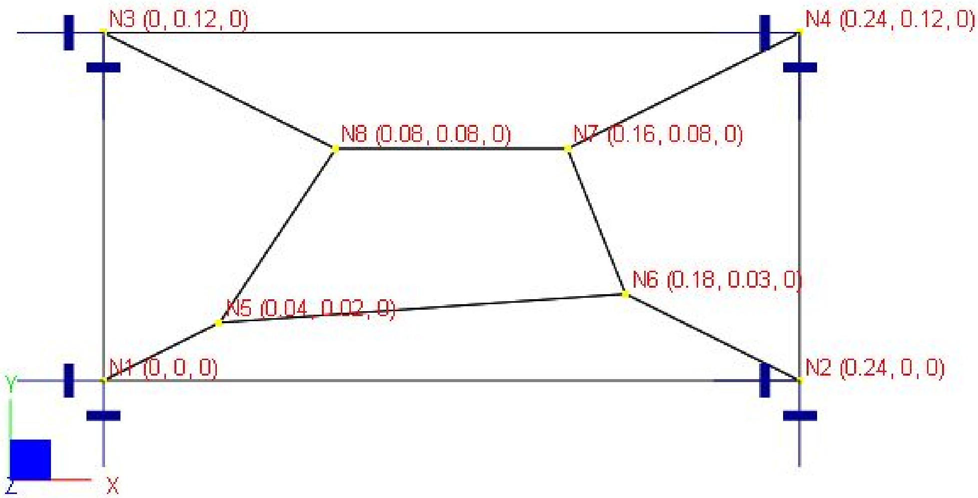
$w = 1.0e^{-3}(x^2 + xy + y^2) / 2$

$$\theta_x = \frac{\partial w}{\partial y} = 1.0e^{-3}(y + x/2)$$

$$\theta_y = -\frac{\partial w}{\partial x} = 1.0e^{-3}(-x - y/2)$$

Material properties: E = 1.0e6 psi, v = 0.25

Geometry: nodal coordinates are shown in the parenthesis below, thickness t = 0.001 in



Finite Element Model

5 shell elements

Model type: 2D Plate Bending (MITC4 thick plate formulation)

Forced displacements on boundary nodes:

Units: displacement – in; rotation - rad

Boundary Nodes	Displacement Dz	Rotation Dox	Rotation Doy
1	0	0	0
2	2.88e-5	1.20e-4	-2.40e-4
3	7.20e-6	1.20e-4	-6.00e-5
4	5.04e-5	2.40e-4	-3.00e-4

Results

The displacements of internal nodes can be calculated based on the boundary conditions. The generalized strains and stresses may be calculated as follows:

$$\varphi_x = \frac{\partial^2 w}{\partial x^2} = 1.0e^{-3}$$;

$$\varphi_y = \frac{\partial^2 w}{\partial y^2} = 1.0e^{-3}$$;

$$\varphi_{xy} = 2\frac{\partial w}{\partial x}\frac{\partial w}{\partial y} = 1.0e^{-3}$$;

$$\begin{pmatrix} M_{xx} \\ M_{yy} \\ M_{xy} \end{pmatrix} = \frac{Et^3}{12(1-\nu^2)} \begin{pmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{pmatrix} \begin{pmatrix} \phi_x \\ \phi_y \\ \phi_{xy} \end{pmatrix} = \begin{pmatrix} 1.11e-7 \\ 1.11e-7 \\ 3.33e-8 \end{pmatrix}$$

The constant stresses are also given by [Ref 1].

Units: displacement – in; rotation - rad

Nodes	ENERCALC 3D			Theoretical		
	Dz	Dox	Doy	Dz	Dox	Doy
5	1.40e-6	4.00e-5	-5.00e-5	1.40e-6	4.00e-5	-5.00e-5
6	1.935e-5	1.20e-4	-1.95e-4	1.935e-5	1.20e-4	-1.95e-4
7	2.24e-5	1.60e-4	-2.00e-4	2.24e-5	1.60e-4	-2.00e-4
8	9.60e-6	1.20e-4	-1.20e-4	9.60e-6	1.20e-4	-1.20e-4

Units: moment – lb-in/in

ENERCALC 3D			[Ref 1]		
Mxx	Myy	Mxy	Mxx	Myy	Mxy
1.11e-7	1.11e-7	3.33e-8	1.11e-7	1.11e-7	3.33e-8

Comments

The results given by ENERCALC 3D are identical to the theoretical and referenced values.

A patch test consists of creating a small "patch" of elements and then imposing an assumed displacement field at the boundary nodes. The assumed displacement field is chosen such that it causes a constant stress in the mesh. To pass the patch test, computed displacements at the interior nodes must be consistent with the assumed displacement field and the computed stresses must be constant. Patch tests are important because they ensure solution convergence—so that increasing mesh fineness results in more accurate results.

The MITC4 plate formulation passes the patch test. The Kirchhoff plate formulation passes the patch test if the elements are rectangular. The Kirchhoff plate formulation is not applicable here.

Reference

[1]. MacNeal & Harder, "A Proposed Standard Set of Problems to Test Finite Element Accuracy", Finite Elements in Analysis and Design, 1 (1985) 3-20

[2]. Cook, Malkus, Plesha, Witt, "Concept and Applications of Finite Element Analysis" 4th Edition, pp238, John Wiley & Sons, Inc., 2002

3.2 B-02 (Parapet)

Objective

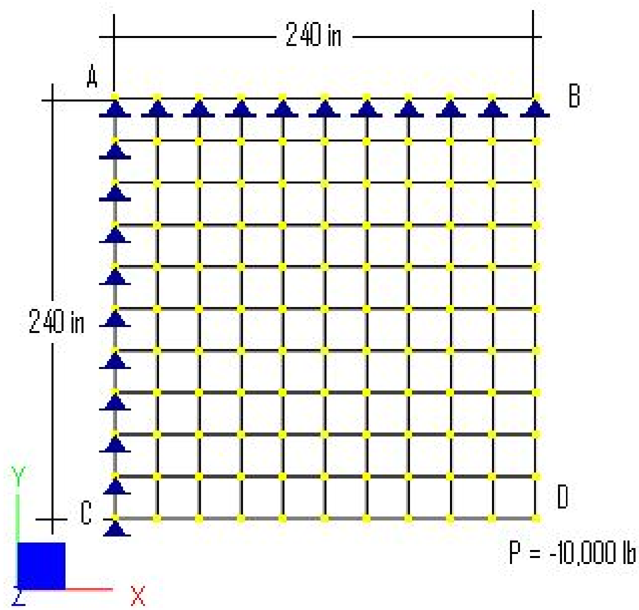
To verify the plate (Kirchhoff thin plate formulation) element under constant twist.

Problem Description

A plate of size 240 x 240 in is subjected to a transverse point load of -10,000 lb at a corner D as shown below. The boundary lines AB and AC are simply supported.

Material properties: E = 2.9e+007 psi, v = 0.30

Thickness t = 10 in



Finite Element Model

100 shell elements

Model type: 2D Plate Bending (Kirchhoff thin plate formulation)

Results

The displacements, internal forces, and moments may be calculated as follows [Ref 1]:

$M_{xx} = M_{yy} = 0$;

$M_{xy} = -P/2 = -5{,}000$ lb-in/in

$V_{xx} = V_{yy} = 0$

$w_D = \dfrac{Pxy}{2(1-\nu)}\dfrac{12(1-\nu^2)}{Et^3} = -0.1549$ in

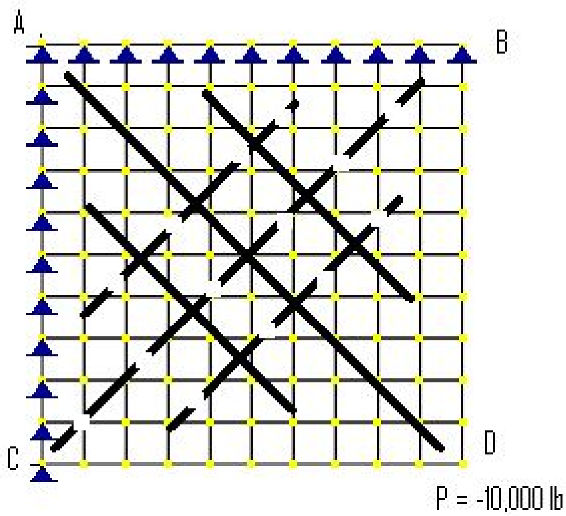
Units: displacement – in; moment – lb-in/in

ENERCALC 3D		[Ref 1]	
Moment Mxy	Displacement Dz @ point D	Moment Mxy	Displacement Dz @ point D
5,000	-0.1549	5,000	-0.1549

Comments

The results given by ENERCALC 3D are identical to the theoretical values.

This is an interesting problem which has practical applications (such as parapet at the corner of a building). It shows that a plate structure may have pure twist Mxy (Mxx = Myy = 0). Generally, for a homogeneous material such as steel, the strength should be checked based on principal stresses. For a non-homogeneous material such as concrete, the strength should be checked based on principal moments (not just Mxx and Myy). In this example, reinforcement should be placed as shown below. The solid lines represent the top reinforcement while the dashed lines do the bottom reinforcement.



In practical applications for concrete slabs, reinforcement placed based on principal moments will be difficult. Alternative methods are available. One of these methods is the so-called Wood-Armer method [Ref 2]. It takes into account Mxy as well as Mxx and Myy for calculating top and bottom reinforcement in two orthogonal directions x and y.

Reference

[1]. Z.L Xu, “Elastic Mechanics” 3rd Ed., pp58, High Education Publishing House, China 1994

ISBN 7-04-002893-X/TB.159

[2]. Park & Gamble “Reinforced Concrete Slab”, pp202, John Wiley & Sons, Inc., 1980

3.3 B-03 (Morley Skew Plate)

Objective

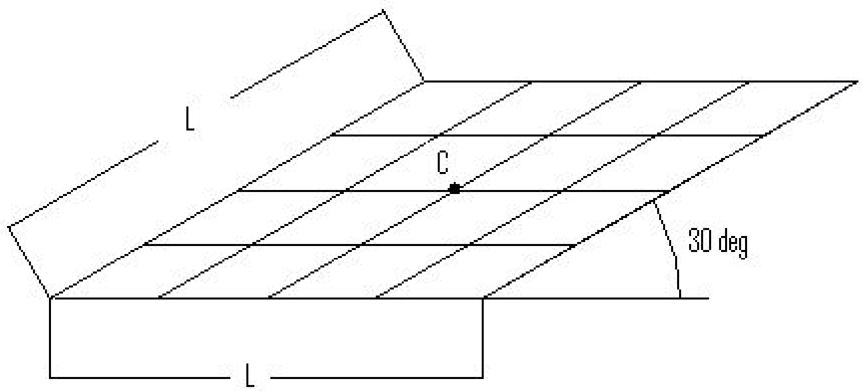
To verify the behavior of the MTC4 thick plate bending element in a skew shape

Problem Description

A skewed, simply supported plate is loaded with a uniform pressure load of 1 psi.

Material properties: E = 1e5 psi, v = 0.3

Geometric properties: L = 100 in, h = 1 in



Finite Element Model

16, 64, 256, 1024 shell elements

Model type: 2D Plate Bending (MITC4 thick plate formulation)

Results

The displacement at the plate center (C) is given by [Ref 1].

Unit: displacement - in

Displacement Dz	ENERCALC 3D	[Ref 1]
4 x 4 mesh	3.9182	3.9182
8 x 8 mesh	3.8991	3.8991
16 x 16 mesh	4.1875	4.1875
32 x 32 mesh	4.4098	4.4098

Comments

The displacements given by ENERCALC 3D are identical to the referenced values. The correct theoretical displacement is given as 4.640 in.

Reference

[1]. Sa, Jorge, Valente and Areias “Development of shear locking-free shell elements using an enhanced assumed strain formulation”, International Journal of Numerical Methods in Engineering, 2002; 53: 1721-1750

3.4 B-04 (Fixed Rectangle)

Objective

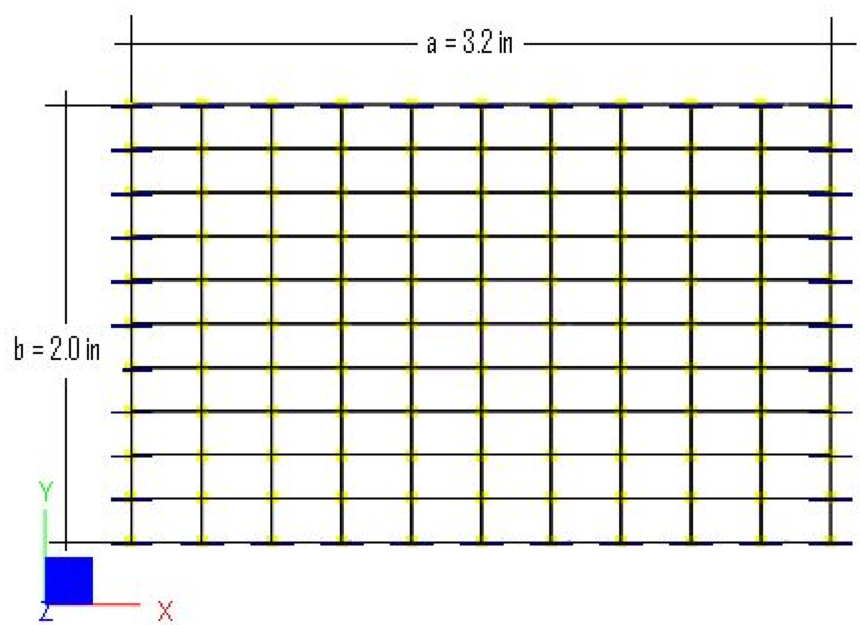
To verify the behavior of the MTC4 thick plate and the Kirchhoff thin plate bending elements

Problem Description

A 3.2 x 2 in rectangular plate is fixed on all edges and subjected to a uniform pressure of q = -1e-4 psi as shown below.

Material properties: E = 1.7472e7 psi, v = 0.3

Thickness: t = 1e-4 in



Finite Element Model

100 shell elements

Model type: 2D Plate Bending (MITC4 thick plate and Kirchhoff thin plate)

Results

The displacements and stresses are compared with those produced by another program, ADINA. Theoretical results are calculated as follows [Ref 1]:

Displacement @ center: $D_z = \dfrac{0.0251 * qb^4}{Et^3} = 2.299$ in

Stress @ center of long edge: $\sigma_y = \dfrac{0.4680 * qb^2}{t^2} = 1.872e4$ psi

Stress @ center: $\sigma_y = \frac{0.2286 * qb^2}{t^2} = 9.144\,e3$ psi

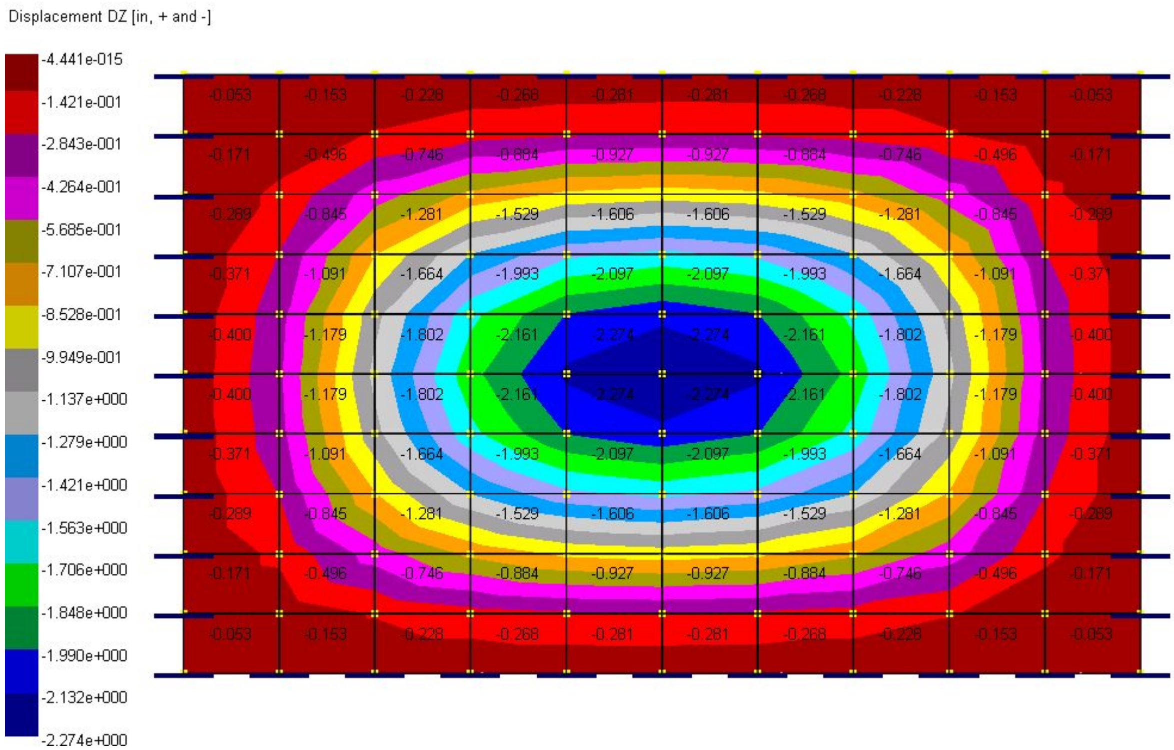
Units: displacement – in; rotation – rad; stress - psi

	ENERCALC 3D		ADINA	Theoretical
	MITC4 (thick)	Kirchhoff (thin)		
Displacement Dz @ center	-2.274	-2.342	-2.274	2.299
Max Rotation Dox	3.653	3.608	3.653	-
Max Rotation Doy	2.502	2.373	2.502	-
Stress Sxx @ center of short edge	7507	12927	7507	-
Stress Sxx @ center	-4880	-4763	-4880	-
Stress Syy @ center of long edge	13478	18743	13478	18720
Stress Syy @ center	-9143	-9483	-9143	-9144
Max Stress Sxy	2556	2459	2556	-

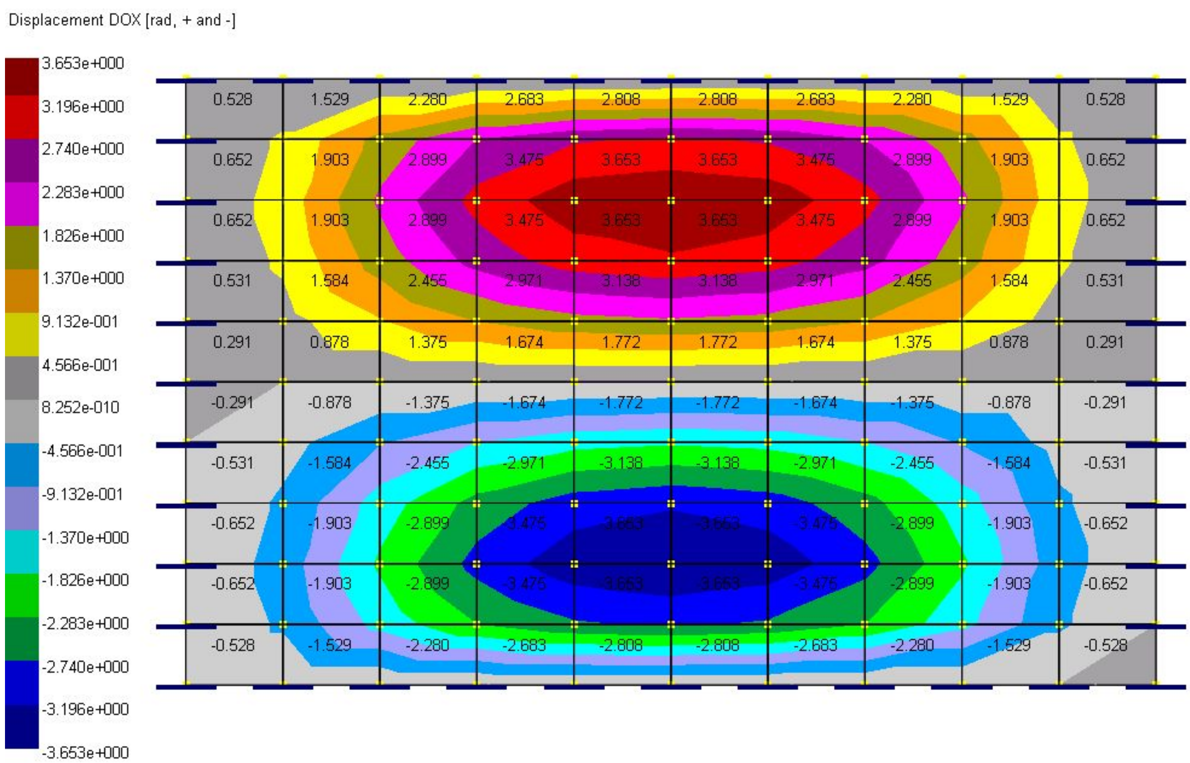
Comments

The results given by ENERCALC 3D using the MITC4 are identical to those given by another reputable finite element program, ADINA. The results also compare well with the theoretical results based on the thin plate theory. The stress prediction of the MITC4 thick plate at the boundary seems to be less accurate than that of the Kirchhoff thin plate.

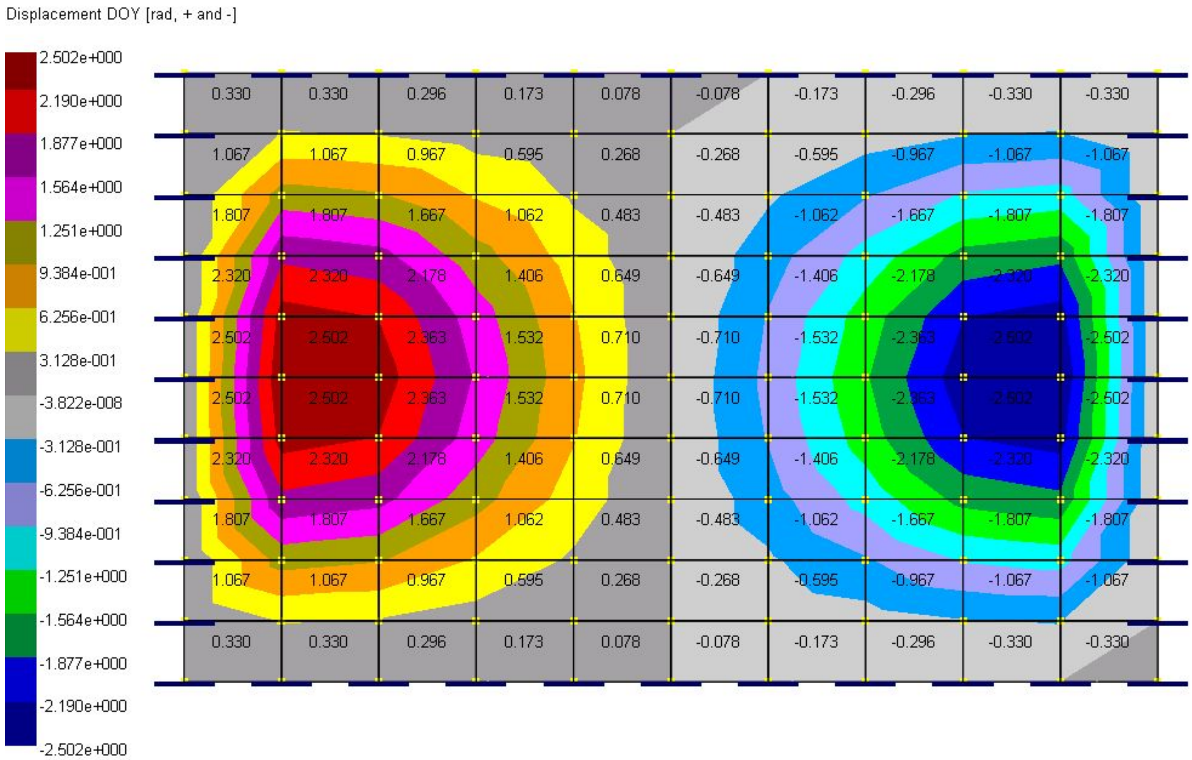
The following illustrates displacement and stress contours (not smoothed) based on the MITC4 thick plate.



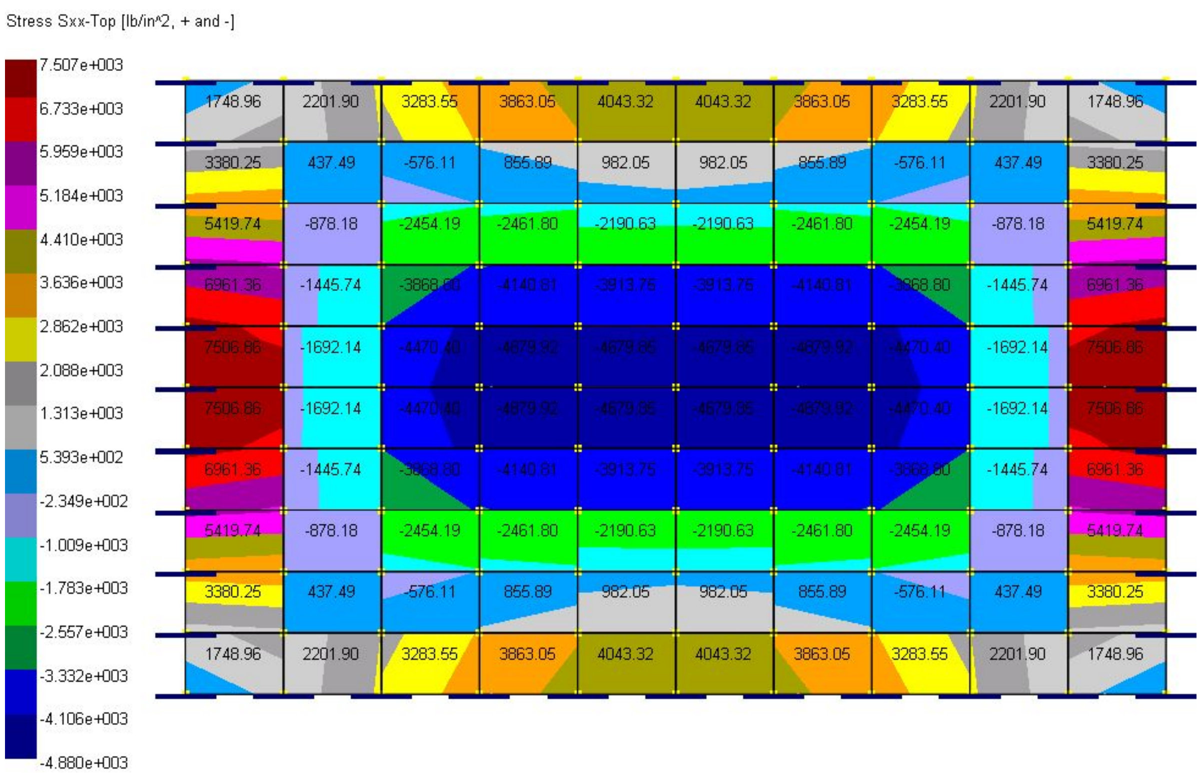
Dz Displacement Contour



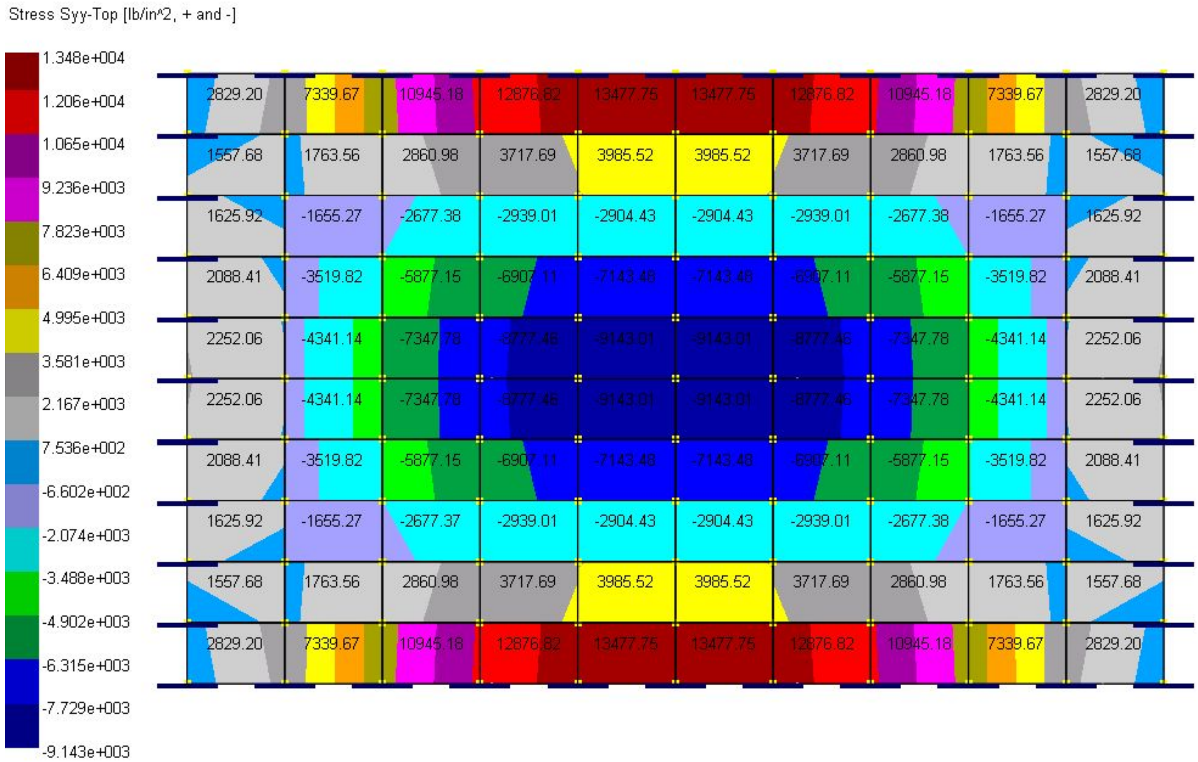
Dox Displacement Contour



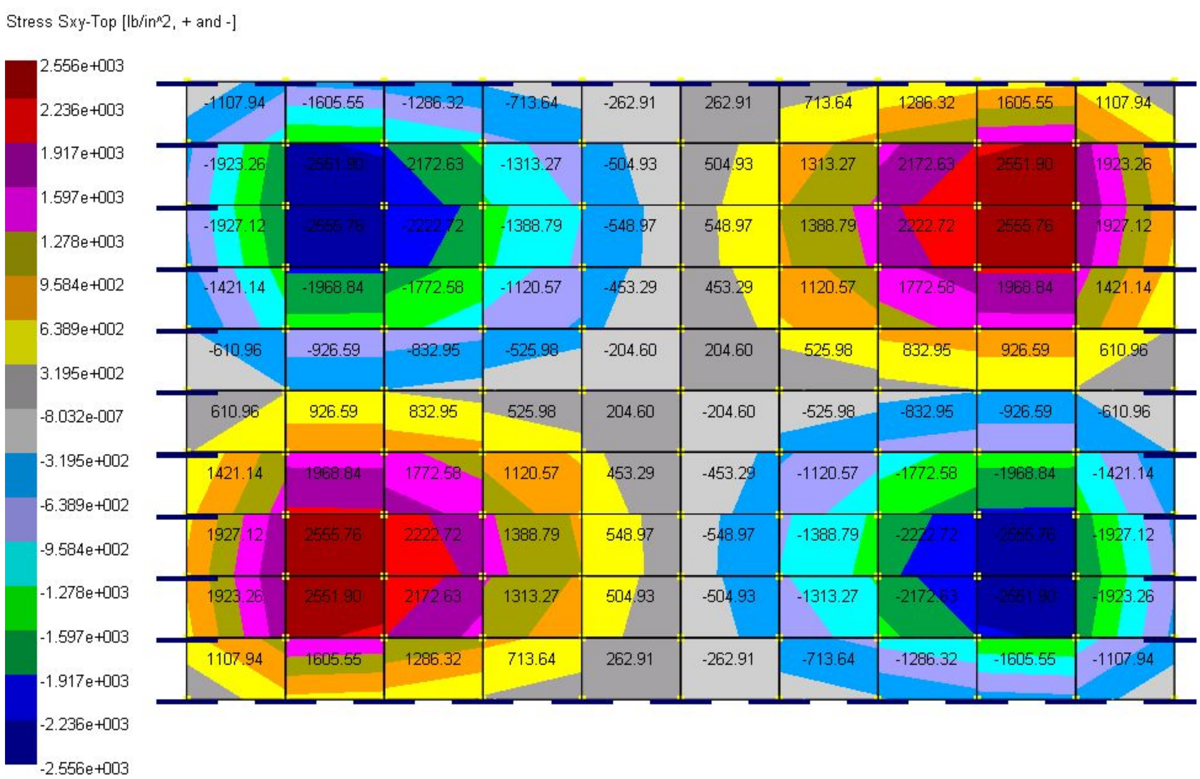
Doy Displacement Contour



Sxx Stress Contour



Syy Stress Contour



Sxy Stress Contour

Reference

[1]. Roark & Yong, “Formulas for Stress and Strain” 5th Ed, pp392, McGraw-Hill Inc., 1975

Part IV

4 Static - Shell Element (Membrane)

4.1 C-01 (Membrane Patch Test)

Objective

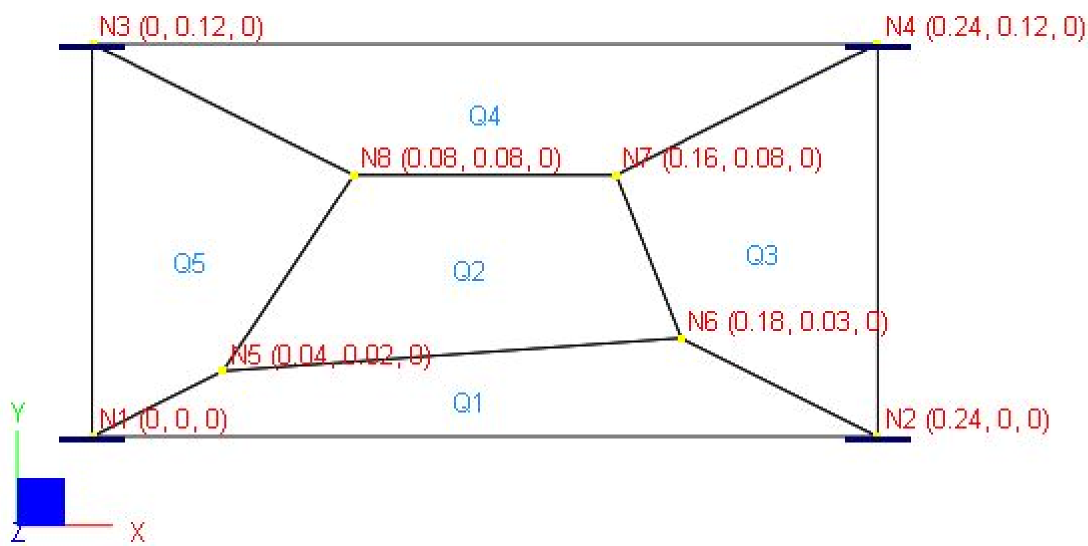
To verify membrane formulations pass the patch test

Problem Description

A plate of size 0.24 x 0.12 in is subjected to forced displacements at the four corners as shown below. The boundary conditions are: $u = 10^{-3}(x + y / 2)$; $v = 10^{-3}(y + x / 2)$

Material properties: E = 1.0e6 psi, v = 0.25

Geometry: nodal coordinates are shown in the parenthesis below, thickness t = 0.001 in



Finite Element Model

5 shell elements

Model type: 2D Plane Stress

Forced displacements on boundary nodes:

Unit: displacement - in

Boundary Nodes	Displacement Dx	Displacement Dy
1	0	0
2	2.4e-4	1.2e-4
3	6.0e-5	1.2e-4
4	3.0e-4	2.4e-4

Results

The displacements of internal nodes can be calculated based on the boundary conditions.

The constant strains may be calculated as follows: $\varepsilon_{xx} = \frac{\partial u}{\partial x} = 1.0e^{-3}$; $\varepsilon_{yy} = \frac{\partial v}{\partial y} = 1.0e^{-3}$;
$\varepsilon_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = 1.0e^{-3}$

Constant stresses may be calculated accordingly and are given in [Ref 1].

Unit: displacement - in

Internal Node	ENERCALC 3D		Theoretical	
	Displacement Dx	Displacement Dy	Displacement Dx	Displacement Dy
5	5.00e-5	4.00e-5	5.00e-5	4.00e-5
6	1.95e-4	1.20e-4	1.95e-4	1.20e-4
7	2.00e-4	1.60e-4	2.00e-4	1.60e-4
8	1.20e-4	1.20e-4	1.20e-4	1.20e-4

Unit: stress - psi

ENERCALC 3D			[Ref 1]		
Stress Sxx	Stress Syy	Stress Sxy	Stress Sxx	Stress Syy	Stress Sxy
1333	1333	400	1333	1333	400

Comments

The results given by ENERCALC 3D are identical to the theoretical and referenced values.

A patch test consists of creating a small “patch” of elements and then imposing an assumed displacement field at the boundary nodes. The assumed displacement field is chosen such that it causes a constant stress in the mesh. To pass the patch test, computed displacements at the interior nodes must be consistent with the assumed displacement field and the computed stresses must be constant. Patch tests are important because they ensure solution convergence—so that increasing mesh fineness results in more accurate results.

Both compatible and incompatible membrane formulations pass the patch test.

Reference

[1]. MacNeal & Harder, “A Proposed Standard Set of Problems to Test Finite Element Accuracy”, Finite Elements in Analysis and Design, 1 (1985) 3-20

[2]. Cook, Malkus, Plesha, Witt, “Concept and Applications of Finite Element Analysis” 4th Edition, pp238, John Wiley & Sons, Inc., 2002

4.2 C-02 (Slender Cantilever)

Objective

To verify membrane formulation of the shell element using regular and irregular element shapes

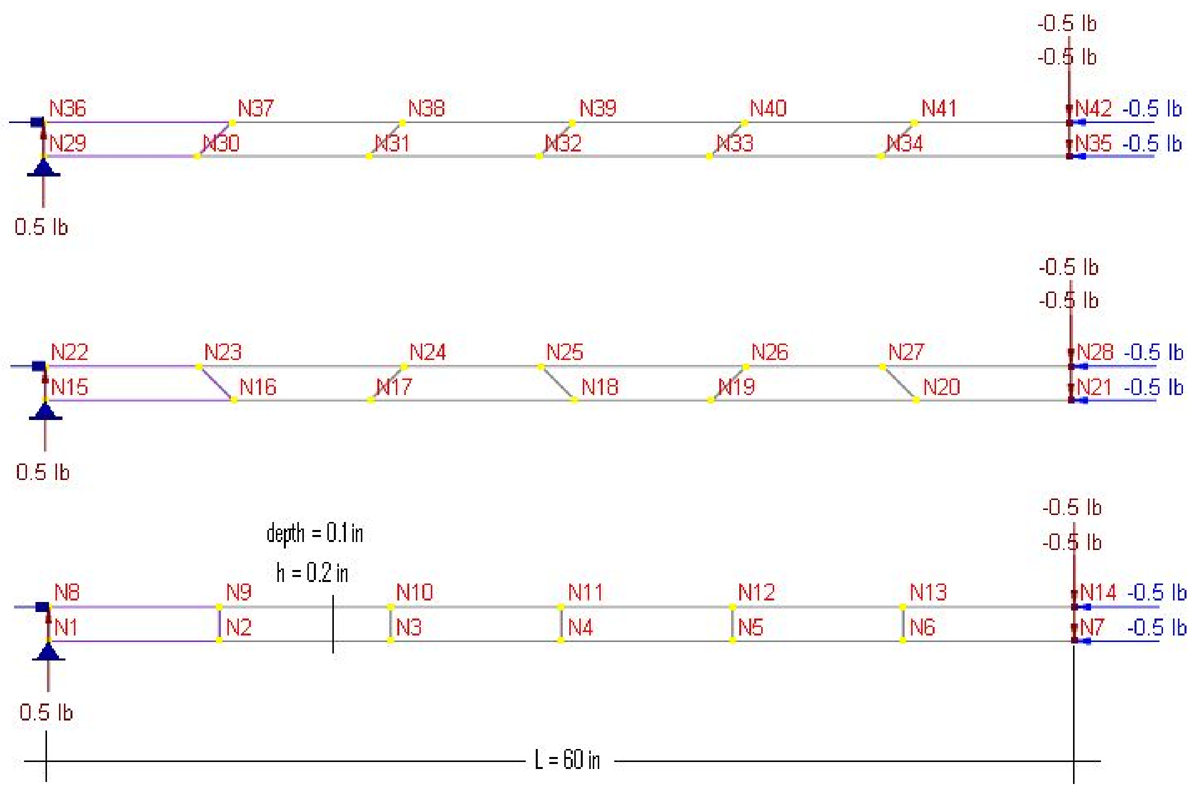
Problem Description

The slender cantilever beam shown below is modeled with a). regular shape elements; b). trapezoidal shape elements; c). parallelogram shape elements. Trapezoidal and parallelogram shapes take 45^0 angle. All elements have equal volume.

Material properties: E = 1.0e7 psi, v = 0.3

Section properties: Length = 60 in, height = 0.2 in, thickness t = 0.1 in

Loads: a). unit axial force; b). unit in-plane shear



Finite Element Model

6 shell elements

Model type: 2D Plane Stress

Results

The tip displacements are given by [Ref 1]. Theoretical stresses at the root are also given here for comparison.

Units: displacement – in; stress - psi

Element	Load type	ENERCALC 3D		[Ref 1]	
		Displacements @ tip	Stresses @ root	Displacements @ tip	Stresses @ root
Compatible Regular	Axial force	3.0e-5	-50	3.0e-5	-50
	In-plane shear	-0.01009	-846.2	0.1081	-9000
Incompatible Regular	Axial force	3.0e-5	-50	3.0e-5	-50
	In-plane shear	-0.1073	-8250.0	0.1081	-9000
Incompatible Trapezoidal	Axial force	3.0e-5	-50	3.0e-5	-50
	In-plane shear	-0.02385	-7071.6	0.1081	-9000
Incompatible Parallelogram	Axial force	3.0e-5	-50	3.0e-5	-50
	In-plane shear	-0.08608	-6510.1	0.1081	-9000

Comments

The results given by ENERCALC 3D are mixed in comparison with the referenced values.

All meshes behave correctly in the axial force loading. For in-plane shear, the regular mesh using incompatible membrane formulation behaves the best. The behavior of the regular mesh using compatible formulation and the irregular mesh using compatible or incompatible formulation can be improved by using more elements. In practice, a rectangular element shape with small aspect ratio should be used whenever possible.

Reference

[1]. MacNeal & Harder, “A Proposed Standard Set of Problems to Test Finite Element Accuracy”, Finite Elements in Analysis and Design, 1 (1985) 3-20

4.3 C-03 (Bathe Membrane Nodal Resultants)

Objective

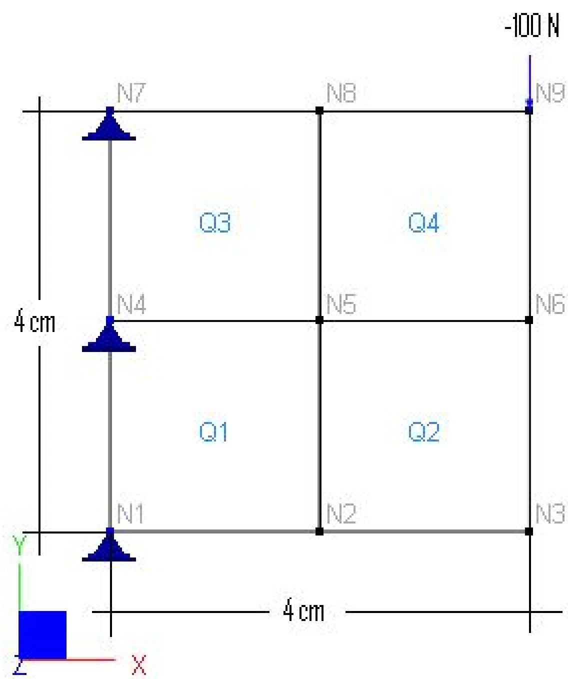
To verify the calculation of nodal resultants for compatible membrane formulation

Problem Description

The cantilever plate shown below is modeled with 2 x 2 mesh using compatible membrane formulation.

Material properties: E = 2.7e6 psi, v = 0.3

Thickness t = 0.1 in



Finite Element Model

4 shell elements

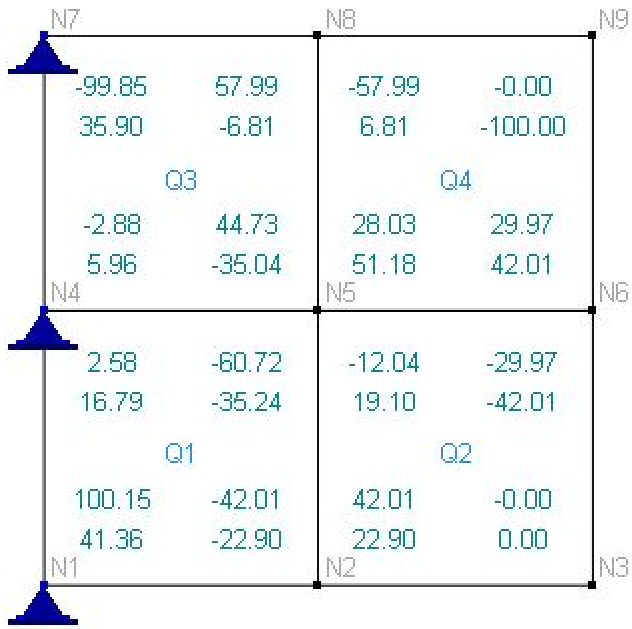
Model type: 2D Plane Stress (using compatible formulation)

Results

The nodal resultants given by ENERCALC 3D are identical to those given by [Ref 1].

As shown below, the nodal resultants are displayed in two lines at each node of each element. The first line denotes the local x component and the second line does the local y component.

The unit is N.



Comments

The results given by ENERCALC 3D are identical to the referenced values.

The nodal resultants represent forces that hold each element in equilibrium. Finite element solutions must always satisfy nodal point equilibrium and element equilibrium. This is true whether a coarse or fine mesh is employed.

Reference

[1]. Bathe, “Finite Element Procedures”, pp 179, Prentice-Hall, Inc., 1996

4.4 C-04 (Cook Membrane Problem)

Objective

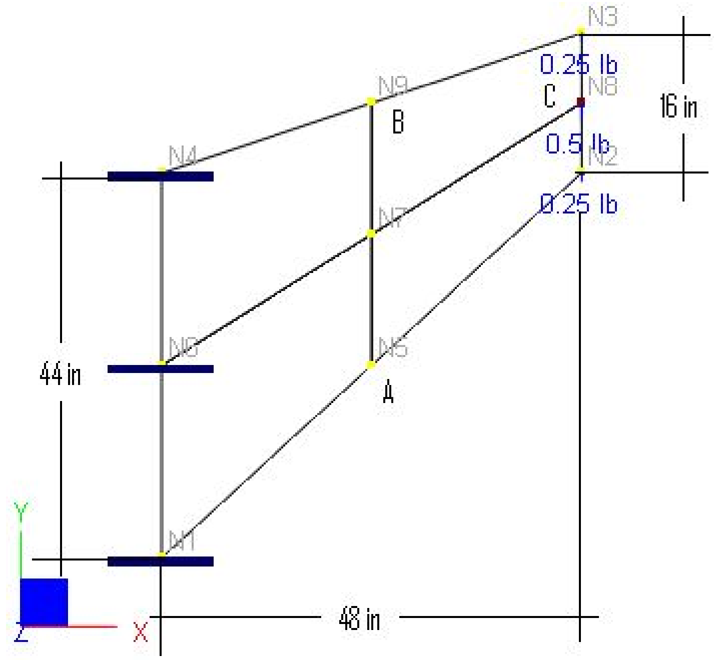
To verify compatible and incompatible membrane formulations

Problem Description

The skewed cantilever plate shown below is subjected to a distributed shear of 1 lb at the end.

Material properties: E = 1.0 psi, v = 0.333

Thickness t = 1 in



Finite Element Model

4 shell elements

Model type: 2D Plane Stress (using compatible and incompatible formulations)

Results

The best results are given by [Ref 1] as follows:

Displacement Dy @ C: 23.9 in

Principal stress S1 @ A: 0.236 psi

Principal stress S2 @ B: -0.201 psi

Units: displacement – in; stress - psi

	Compatible formulation			Incompatible formulation		
	Displacement Dy @ C	Principal Stress S1 @ A	Principal Stress S2 @ B	Displacement Dy @ C	Principal Stress S1 @ A	Principal Stress S2 @ B
2 x 2 mesh	11.85	0.1078	-0.07762	21.05	0.1789	-0.1694
64x64	23.92	0.2376	-0.2038	23.96	0.2368	-0.2035
[Ref 1]	23.9	0.236	-0.201	23.9	0.236	-0.201

Comments

The results given by ENERCALC 3D are compared with the referenced values. For the 2 x 2 coarse mesh, the incompatible formulation is superior to the compatible one. For the 64 x 64 fine mesh, both compatible and incompatible formulations give satisfactory results.

Reference

[1]. Bergan & Filippa, “Triangular membrane element with rotational degrees of freedom”, Comput. Meth. Appl. Mech. Engng., 50: 25-69, 1985

Part V

5 Static - Shell Element

5.1 D-01 (Bathe Membrane + Beam)

Objective

To verify the combined behavior of compatible membrane and beam elements

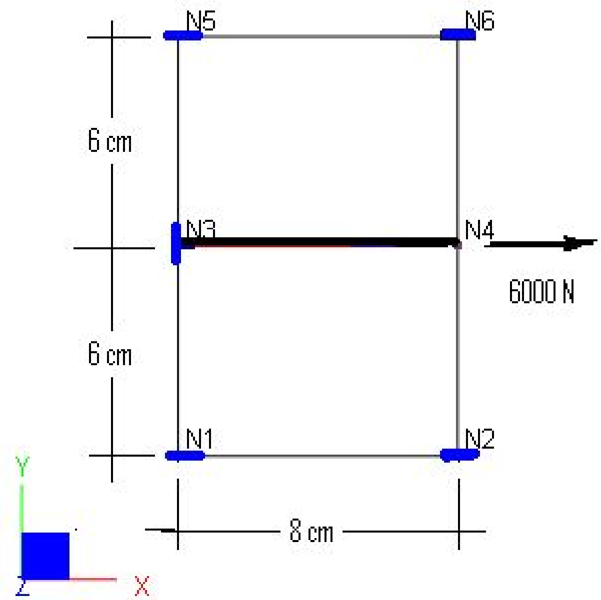
Problem Description

An 8 x 12 cm plate is fixed on three sides. It is reinforced with a bar element in the middle as shown below. The free end of the bar is subjected to a horizontal force of 6000 N.

Material properties: E = 30e6 N/cm^2, v = 0.30

Plate thickness t = 0.1 cm

Bar cross sectional area = 1 cm^2



Finite Element Model

2 shell elements + 1 beam element

Model type: 3D Frame & Shell (use compatible membrane formulation)

Results

The tip displacement of the bar given by ENERCALC 3D is compared with that given by [Ref 1] as follows:

Unit: displacement - cm

	ENERCALC 3D	[Ref 1]
Tip displacement Dx @ N4	9.34e-4	9.34e-4

Comments

The result given by ENERCALC 3D is identical to the referenced value.

Reference

[1]. Bathe, “Finite Element Procedures”, pp361, Prentice-Hall, Inc., 1996

5.2 D-02 (Curved Beam)

Objective

To verify the shell element using incompatible membrane and the MITC4 thick plate formulations

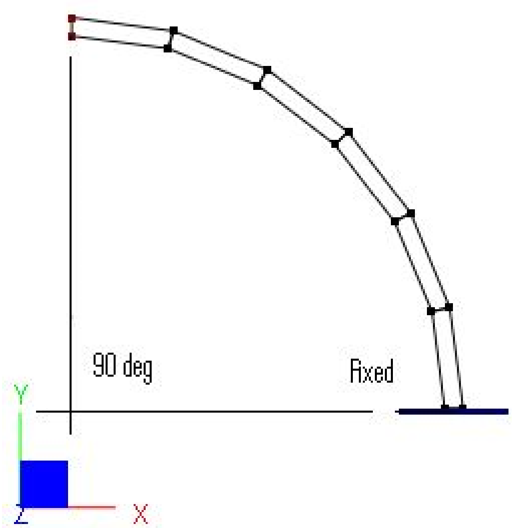
Problem Description

The curved beam shown below [Ref 1] is fixed at the bottom and loaded with two sets of loads at the tip: 1.0 lbf in-plane shear and 1.0 lbf unit out-of-plane shear.

Material properties: E = 1.0e7 psi, v = 0.25

Plate thickness t = 0.1 in

Curved beam inner radius = 4.12 in, outer radius = 4.32 in, arc = 90^{o}



Finite Element Model

6 shell elements

Model type: 3D Frame & Shell (use incompatible membrane and MITC4 thick plate formulations)

Results

The tip displacements in the direction of loads given by ENERCALC 3D are compared with that given by [Ref 1] as follows:

Unit: displacement - in

Displacement in load direction	ENERCALC 3D	[Ref 1]
In-plane shear (in)	0.07751	0.08734 (see Note)
Out-of-plane shear (in)	0.4798	0.5022

Note: The displacement given by [Ref 1] is smaller than the theoretical calculation based on the following [Ref 2]:

$R_{avg} = \frac{4.32+4.12}{2} = 4.22$ in

$I = \frac{0.1*0.2^3}{12} = 6.66667\,e-5$ in^4

$D_y = \frac{\pi/4*P*R_{avg}^3}{EI} = 0.08853$ in

Comments

The results given by ENERCALC 3D are very good considering the very coarse mesh employed. We would obtain better results if more elements were used along the beam length.

Reference

[1]. MacNeal & Harder, “A Proposed Standard Set of Problems to Test Finite Element Accuracy”, Finite Elements in Analysis and Design, 1 (1985) 3-20

[2]. Roark & Yong, “Formulas for Stress and Strain” 5th Ed, pp215, McGraw-Hill Inc., 1975

5.3 D-03 (Pinched Cylinder)

Objective

To verify the membrane and bending behavior of the shell element in a curved structure

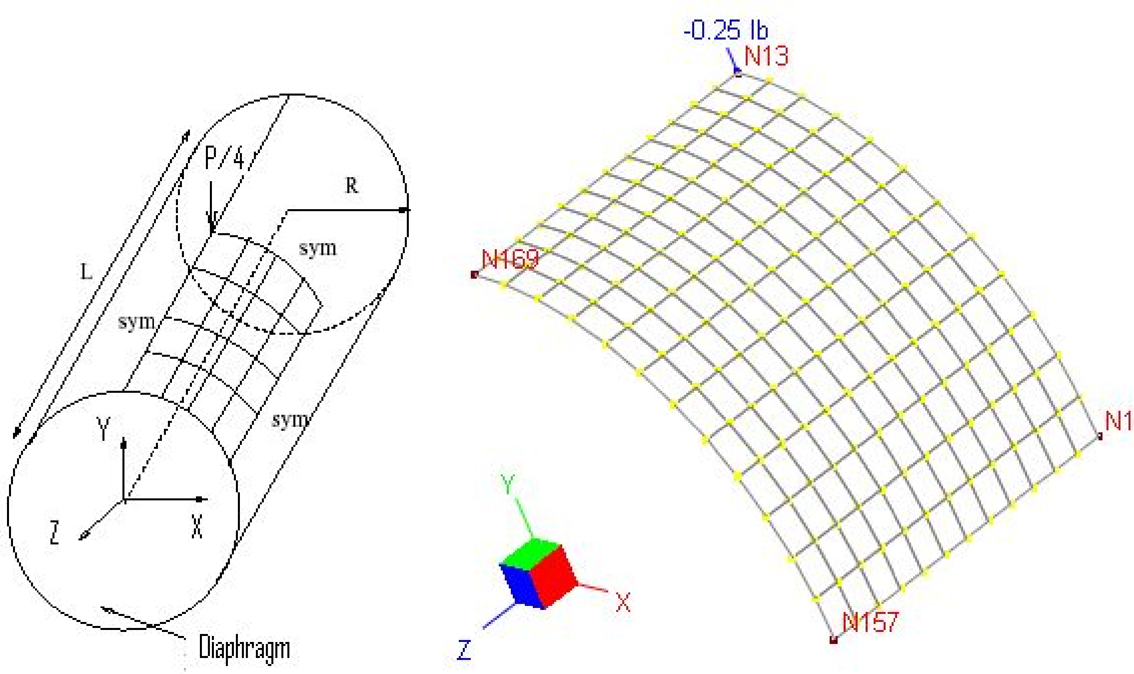
Problem Description

A thin cylindrical shell with diaphragm boundary conditions at both circular ends is loaded with two opposed point loads at the center of the surface.

Material properties: E = 3.0e6 psi, v = 0.3

Geometric properties: L = 600 in, R = 300 in, t = 3 in

Load: P = 1.0 lb



Finite Element Model

144 shell elements
Due to symmetry, one eighth of the cylinder is modeled with a12x12 mesh
Boundary conditions:

Edge N1-N13: Dz, Dox, Doy fixed
Edge N1-N157: Dy, Dox, Doz fixed
Edge N13-N169: Dx, Doy, Doz fixed
Edge N157-N169: Dx, Dy, Doz fixed

Note: N13 is restrained in Dx, Dz, Dox, Doy, Doz.

Model type: 3D Frame and Shell

Results

The deflection under load is given by [Ref 1] as Dy = -1.825e-5 in.

Unit: displacement - in

ENERCALC 3D				[Ref 1]
Displacement under load using different shell formulations				-1.825e-005
Compatible membrane		Incompatible membrane		
Kirchhoff	MITC4	Kirchhoff	MITC4	
-1.819e-005	-1.595e-005	-1.833e-005	-1.605e-005	

Comments

The results given by ENERCALC 3D are comparable to the referenced values.

It appears that the Kirchhoff thin plate bending formulation yields results close to the referenced values. This is especially true when plate/shell thickness is very thin. Of course, we have to remember that the Kirchhoff plate only applies to rectangular shell elements.

Reference

[1]. Cook, Malkus, Plesha, Witt, “Concept and Applications of Finite Element Analysis” 4th Edition, pp583, John Wiley & Sons, Inc., 2002

5.4 D-04 (Scordelis-Lo Roof)

Objective

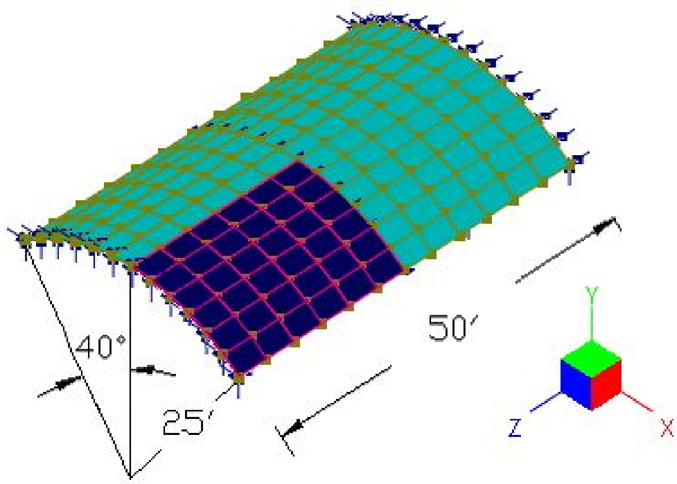
To verify the membrane and bending behavior of the shell element in a curved structure

Problem Description

The Scordelis-Lo barrel roof below [Ref 1, Ref 2] has a length of 50 ft, a radius of 25 ft, and a sweeping angle of 80 degrees. The roof is supported on rigid diaphragms along its two curved edges (D_x and D_y fixed, but not D_z). The two straight edges are free. A surface load of -90 lb/ft^2 in the global Y direction (self weight) is applied to the entire roof.

Material: E = 4.32e8 lb/ft^2 (3e6 psi); v = 0.0;

Thickness: t = 0.25 ft.

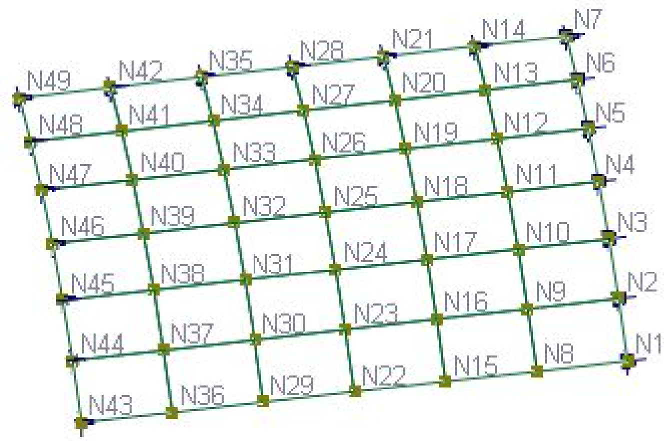


Finite Element Model

36 shell elements
Due to symmetry, one quarter of the roof is modeled with a 6x6 mesh. The boundary conditions are specified in the following table.

Nodes	Fixed DOFs
N1 to N6	Z, OX, OY
N7	X, Z, OX, OY, OZ
N14, N21, N26, N35, N42	X, OY, OZ
N43 to N48	X, Y, OZ
N49	X, Y, OY, OZ

Model type: 3D Frame and Shell



Results

The results given by ENERCALC 3D compare well with benchmark values.

Units: displacement – ft; stress - ksf

	Displacement Dy @ N1	Displacement Dx @ N1	Top Principal Stress S1 @ N7	Bottom Principal Stress S2 @ N7	Top Principal Stress S1 @ N1	Bottom Principal Stress S1 @ N1
MITC4 Compatible	-0.291	-0.153	171.74	-197.69	242.55	349.35
MITC4 Incompatible	-0.307	-0.162	183.97	-210.78	225.09	352.77
Kirchhoff Compatible	-0.290	-0.153	174.88	-200.62	238.73	351.68
Kirchhoff Incompatible	-0.306	-0.161	187.55	-214.41	224.46	352.69
Benchmark Value	-0.302	-0.159	191.23	-218.74	215.57	340.70

Comments

The results given by ENERCALC 3D are comparable to the referenced values.

Reference

[1]. MacNeal & Harder, "A Proposed Standard Set of Problems to Test Finite Element Accuracy", Finite Elements in Analysis and Design, 1 (1985) 3-20

[2]. Scordelis & Lo, "Computer Analysis of Cylindrical Shells", Journal of the American Concrete Institute, Volume 61, May, 1964

5.5 D-05 (Hemispherical Shell with Point Loads)

Objective

To verify the membrane and bending behavior of the MITC4 shell element in a doubly-curved, very thin shell structure

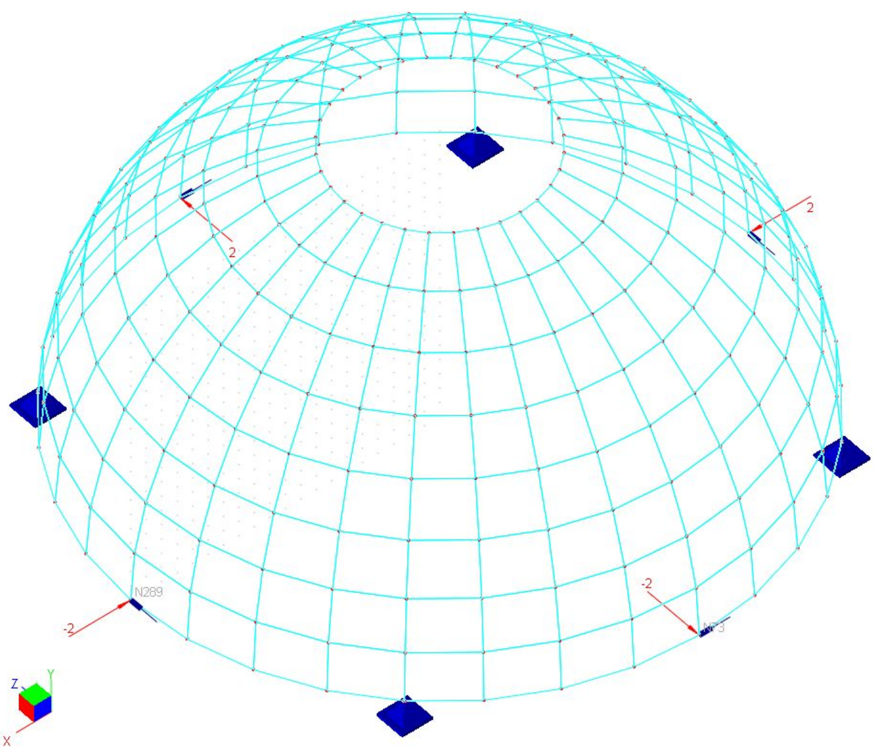
Problem Description

The hemispherical shell below [Ref 1] has a radius of 10 ft and a thickness of 0.04 ft. The equator is a free edge and is loaded with four 2-kip point loads alternating in sign at 90 degrees intervals. The edge of the hole at the top (72 degrees from the axis of revolution) is free.

Material: E = 6.825e7 kip/ft^2; v = 0.3;

Thickness: t = 0.04 ft;

Radius R = 10 ft.



Finite Element Model

8 x 32, 16 x 64 and 32 x 128 shell elements

For simplicity of boundary conditions, symmetry of the structure is not considered. The boundary restraints are applied to prevent instability of the structure.

Model type: 3D Frame and Shell

Results

The results given by ENERCALC 3D compare well with benchmark values.

Units: displacement – ft

Radial displacement at load point

	8 x 32 mesh	16 x 64 mesh	32 x 128 mesh
MITC4 Compatible	9.272e-2	9.289e-2	9.334e-2
MITC4 Incompatible	9.292e-2	9.313e-2	9.346e-2
Benchmark Value	9.400e-2	9.400e-2	9.400e-2

Comments

The results given by ENERCALC 3D are comparable to the benchmark values.

This problem is one of the more challenging benchmark tests for shell elements. The reason is that the shell is doubly curved and shell thickness is very small in comparison with its span (radius). Both membrane and bending strains contribute significantly to the radial displacement at the load point. This example shows the superiority of the MITC4 shell element.

We could have taken advantage of the symmetry and only model one quadrant of the structure. The boundary condition requires a little more thinking but is still straightforward in this case. An example is included with the program to illustrate this approach.

Modeling Tips

The most efficient way to construct this model in the program is as follows. First generate arc members using the command Create > Templates > Arc Members. Then use Modify > Revolve > Revolve Members to Shells command to generate doubly curved shell elements.

Reference

[1]. MacNeal & Harder, “A Proposed Standard Set of Problems to Test Finite Element Accuracy”, Finite Elements in Analysis and Design, 1 (1985) 3-20

Part VI

6 Static - Brick Element

6.1 E-01 (Slender Brick Beam)

Objective

To verify compatible and incompatible brick formulations using regular element shapes

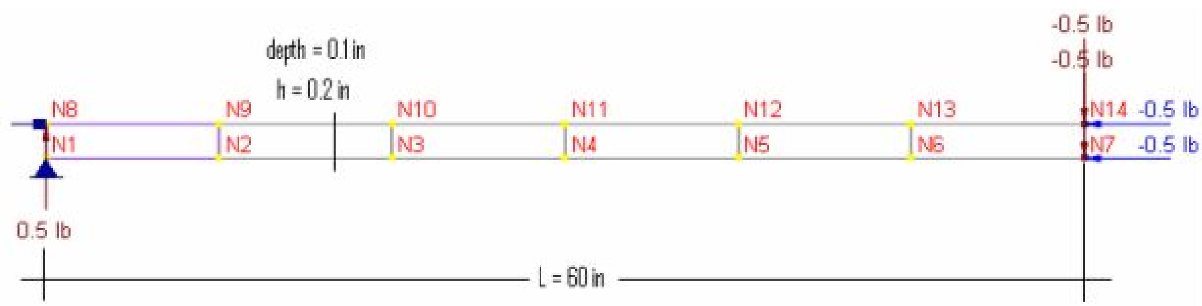
Problem Description

The slender cantilever beam shown below is modeled with 6 rectangular brick elements.

Material properties: E = 1.0e7 psi, v = 0.3

Section properties: Length = 60 in, height = 0.2 in, thickness t = 0.1 in

Loads: a). unit axial force; b). unit in-plane shear



Finite Element Model

6 brick elements

Model type: 3D Brick

Results

The tip displacements are given by [Ref 1]. Theoretical stresses at the root are also given here for comparison.

Units: displacement – in; stress - psi

Element	Load type	ENERCALC 3D		[Ref 1]	
		Displacements @ tip	Stresses @ root	Displacements @ tip	Stresses @ root
Compatible	Axial force	3.0e-5	-50	3.0e-5	-50
	In-plane shear	-0.01007	-854.0	0.1081	-9000
Incompatible	Axial force	3.0e-5	-50	3.0e-5	-50
	In-plane shear	-0.1072	-8173	0.1081	-9000

Comments

The results given by ENERCALC 3D are mixed in comparison with the referenced values.

Compatible and incompatible formulations behave correctly in the axial force loading. For in-plane shear, the incompatible brick formulation yields much better results than the compatible one. In practices, finer meshes should be used to achieve satisfactory results, especially for compatible brick elements.

Reference

[1]. MacNeal & Harder, “A Proposed Standard Set of Problems to Test Finite Element Accuracy”, Finite Elements in Analysis and Design, 1 (1985) 3-20

6.2 E-02 (Curved Brick Beam)

Objective

To verify the incompatible brick element in a curved structure

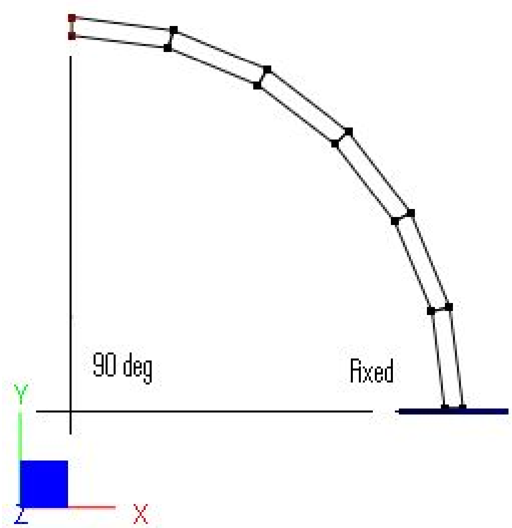
Problem Description

A curved beam as shown below [Ref 1] is fixed at the bottom and loaded with two sets of loads at the tip: 1.0 lbf in-plane shear and 1.0 lbf out-of-plane shear.

Material properties: E = 1.0e7 psi, v = 0.25

Plate thickness t = 0.1 in

Curved beam inner radius = 4.12 in, outer radius = 4.32 in, arc = 90^{o}



Finite Element Model

6 brick elements

Model type: 3D Brick (use incompatible formulations)

Results

The tip displacements in the direction of loads given by ENERCALC 3D are compared with that given by [Ref 1] as follows:

Unit: displacement - in

Displacement in load direction	ENERCALC 3D		[Ref 1]
	6 x 1 mesh	20 x 1 mesh	
In-plane shear (in)	0.07682	0.08814	0.08734 (see Note)
Out-of-plane shear (in)	0.4116	0.4797	0.5022

Note: The displacement given by [Ref 1] is smaller than the theoretical calculation based on the following [Ref 2]:

$$R_{avg} = \frac{4.32 + 4.12}{2} = 4.22 \text{ in}$$

$$I = \frac{0.1 * 0.2^3}{12} = 6.66667e-5 \text{ in}^4$$

$$D_y = \frac{\pi / 4 * P * R_{avg}^3}{EI} = 0.08853 \text{ in}$$

Comments

The results given by ENERCALC 3D are very good considering the relatively coarse meshes employed. We would obtain better results if more elements were used along the beam length.

Reference

[1]. MacNeal & Harder, “A Proposed Standard Set of Problems to Test Finite Element Accuracy”, Finite Elements in Analysis and Design, 1 (1985) 3-20

[2]. Roark & Yong, “Formulas for Stress and Strain” 5^{th} Ed, pp215, McGraw-Hill Inc., 1975

6.3 E-03 (Incompatible Brick)

Objective

To verify the behavior of incompatible brick formulations using irregular meshes

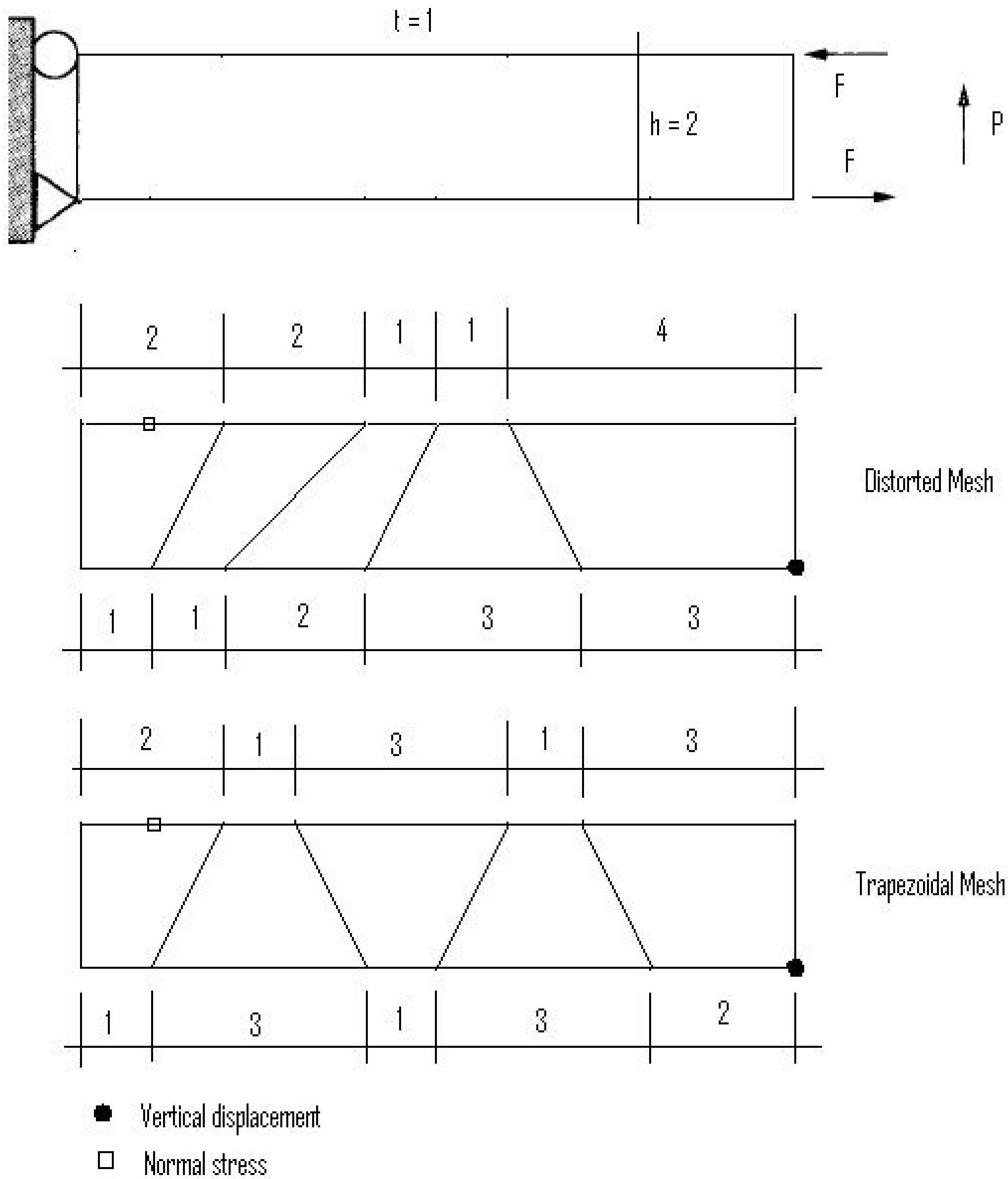
Problem Description

A straight beam with distorted and trapezoidal elements is subjected to two sets of loading: a). end moments; b). end shear.

Material properties: E = 1500 psi, v = 0.25

Geometric properties: L = 10 in, h = 2 in, t = 1 in

Loads: a). F = 1000 lb; b). P = 300 lb



Finite Element Model

5 brick (incompatible) elements

Model type: 3D Brick

Results

The displacements and stresses are given by [Ref 1]. The stresses given for ENERCALC 3D below are the average values at the top four nodes of each of the elements at the supports.

Unit: displacement – in; stress - psi

Mesh	Loading	ENERCALC 3D		Ref 1 (Theoretical)	
		Displacements @ tip	Stresses @ root	Displacements @ tip	Stresses @ root
Distorted	Moment	95.80	-2471	95.8 (100)	-3015 (3000)
	Shear	97.90	-3223	97.9 (102.6)	-4138.5 (-4050)
Trapezoidal	Moment	76.27	-2503	76.252 (100)	-2883.5 (3000)
	Shear	80.16	-3309	80.115 (102.6)	-3860 (-4050)

Comments

The displacements given by ENERCALC 3D are almost identical to the referenced values. The stresses are calculated by averaging the top four nodes of each element at the root. The stresses given by ENERCALC 3D are different from the referenced values due to different methods used in stress calculation. The correct theoretical displacements and stresses are given in parenthesis in the table.

Reference

[1]. Wilson, Ibrahimbegovic, “Use of incompatible displacement modes for the calculation of element stiffness or stresses”, Finite Elements in Analysis and Design 7 (1990) 229-241

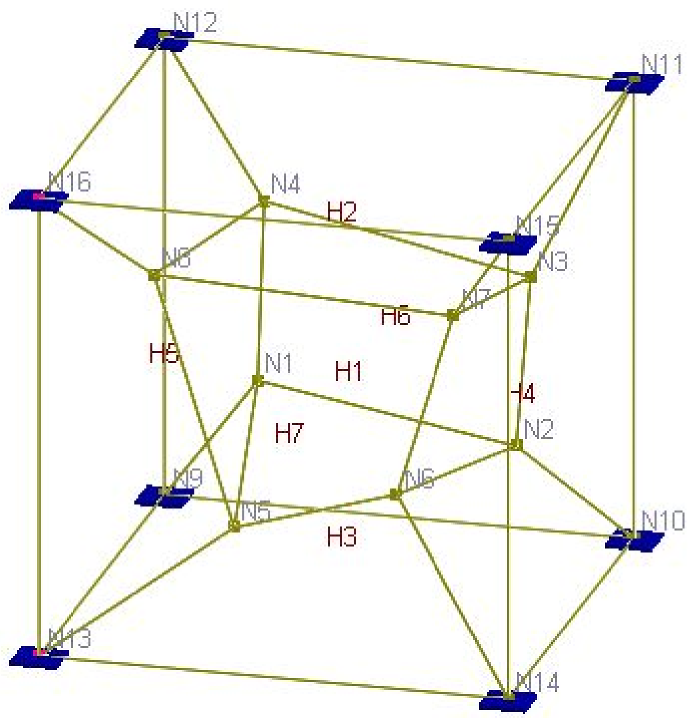
6.4 E-04 (Brick Patch Test)

Problem Description

This is a patch test for a unit cube [Ref 1]. The cube is modeled with 7 eight-node brick elements. Nodal coordinates, element connectivity, and boundary conditions are given in the following tables. Boundary conditions are given as forced displacements. No additional loads are prescribed.

Material: E = 1.e6 psi; v = 0.25

Find stresses for each element.



Nodal coordinates (inch)

Node	X	Y	Z
1	0.249	0.342	0.192
2	0.826	0.288	0.288
3	0.85	0.649	0.263
4	0.273	0.75	0.23
5	0.32	0.186	0.643
6	0.677	0.305	0.683
7	0.788	0.693	0.644
8	0.165	0.745	0.702
9	0	0	0
10	1	0	0
11	1	1	0
12	0	1	0
13	0	0	1
14	1	0	1
15	1	1	1
16	0	1	1

Displacement field
u = 0.001 * (2x + y + z) / 2
v = 0.001 * (x + 2y + z) / 2
w = 0.001 * (x + y + 2z) / 2
Forced displacements (inch) on boundary

Node	Dx	Dy	Dz
9	0	0	0
10	0.001	0.0005	0.0005
11	0.0015	0.0015	0.001
12	0.0005	0.001	0.0005
13	0.0005	0.0005	0.001
14	0.0015	0.001	0.0015
15	0.002	0.002	0.002
16	0.001	0.0015	0.0015

All strains are constant. For example

$$\varepsilon_x = \frac{\partial u}{\partial x} = 0.001$$

$$\varepsilon_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = 0.001$$

Element Connectivity

Element	Node1	Node2	Node3	Node4	Node5	Node6	Node7	Node8
1	1	2	3	4	5	6	7	8
2	4	3	11	12	8	7	15	16
3	9	10	2	1	13	14	6	5
4	2	10	11	3	6	14	15	7
5	9	1	4	12	13	5	8	16
6	9	10	11	12	1	2	3	4
7	5	6	7	8	13	14	15	16

Results

The displacements of internal nodes can be calculated based on the boundary conditions. The constant stresses are also given by [Ref 1].

Units: displacement – in

Nodes	ENERCALC 3D (compatible and incompatible)			Theoretical		
	Dx	Dy	Dz	Dx	Dy	Dz
1	5.16E-04	5.63E-04	4.88E-04	5.16E-04	5.63E-04	4.88E-04
2	1.11E-03	8.45E-04	8.45E-04	1.11E-03	8.45E-04	8.45E-04
3	1.31E-03	1.21E-03	1.01E-03	1.31E-03	1.21E-03	1.01E-03
4	7.63E-04	1.00E-03	7.42E-04	7.63E-04	1.00E-03	7.42E-04
5	7.35E-04	6.68E-04	8.96E-04	7.35E-04	6.68E-04	8.96E-04
6	1.17E-03	9.85E-04	1.17E-03	1.17E-03	9.85E-04	1.17E-03
7	1.46E-03	1.41E-03	1.38E-03	1.46E-03	1.41E-03	1.38E-03
8	8.89E-04	1.18E-03	1.16E-03	8.89E-04	1.18E-03	1.16E-03

Units: stress - psi

	Sxx	Syy	Szz	Sxy	Syz	Sxz
ENERCALC 3D (compatible)	1999.982	1999.982	1999.982	399.999	399.999	399.999
ENERCALC 3D (incompatible)	1999.978	1999.978	1999.978	399.998	399.998	399.998
[Ref. 1]	2000	2000	2000	400	400	400

Comments

Both compatible and incompatible brick elements pass the patch test. Therefore, “the results for any problem solved with the element will converge toward the correct solution as the elements are subdivided.” [Ref. 1] The tiny differences in stresses are due to the penalty approach employed in support enforcement during solution.

Reference

[1]. MacNeal & Harder, “A Proposed Standard Set of Problems to Test Finite Element Accuracy”, Finite Elements in Analysis and Design, 1 (1985) 3-20

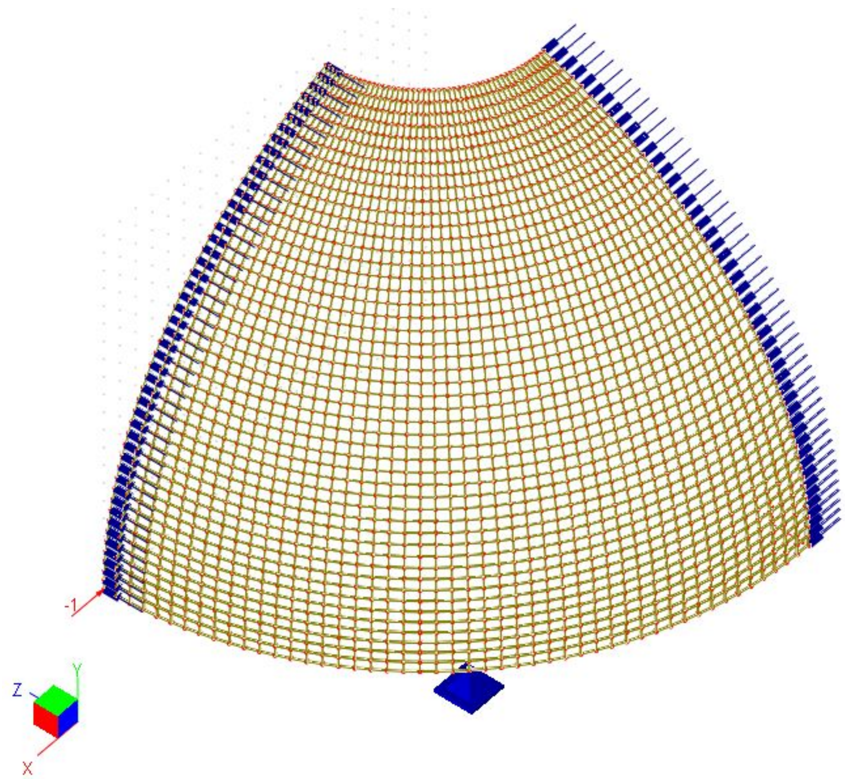
6.5 E-05 (Hemispherical Shell with Point Loads)

Objective

To verify the behavior of the incompatible brick element in a doubly-curved, very thin shell structure

Problem Description

This problem is the same as problem D-05. Only this time we are using the 3D brick element instead of the MITC4 shell element to model the structure.



Finite Element Model

48 x 48 x 1 incompatible brick elements

Due to symmetry of the structure, we model only a quadrant of the structure. Restraints in the direction of global X and Z are applied to the quadrant lines respectively. A single vertical restraint is applied at the center of the quadrant equator. This is to prevent instability of the structure.

Model type: 3D Brick

Results

The result given by ENERCALC 3D compares well with benchmark values.

Units: displacement – ft

Radial displacement at load point

	48 x 48 x 1 mesh
Incompatible brick element	9.262e-2
Benchmark Value	9.400e-2

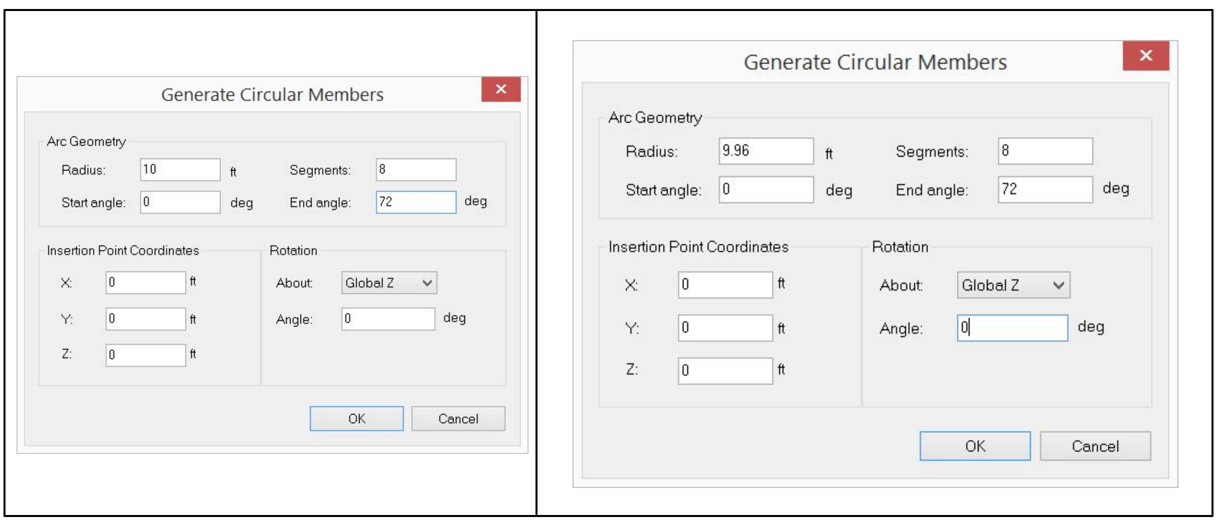
Comments

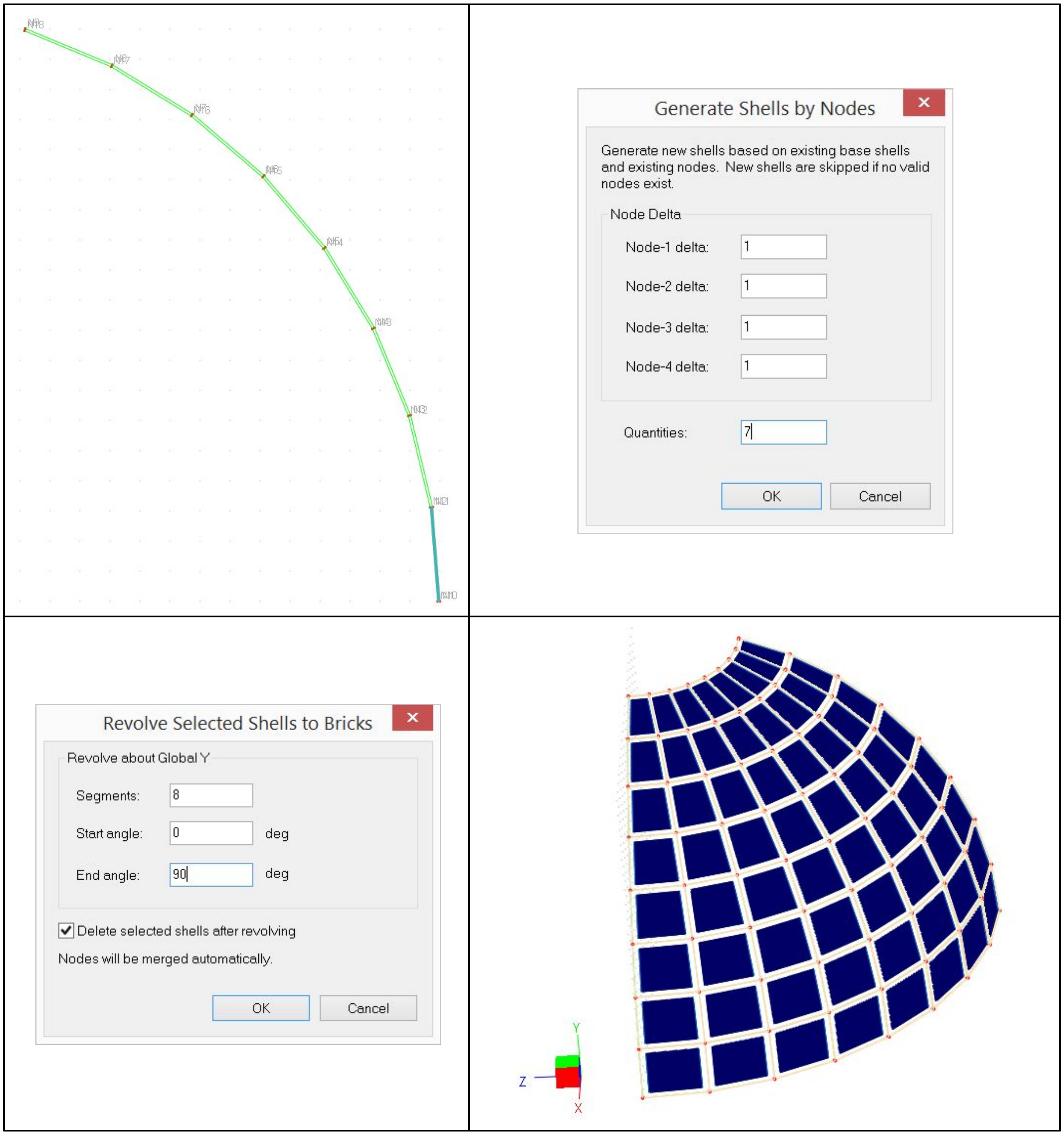
The result given by ENERCALC 3D is comparable to the benchmark value.

This problem is one of the more challenging benchmark tests for solid elements. The reason is that the shell is doubly curved and shell thickness is very small in comparison with its span (radius). We used a relatively fine mesh so the element aspect ratio (8:7:1) would not be too large. Also, we used incompatible brick element formulation. Compatible brick element formulation would be too stiff for this mesh model.

Modeling Tips

The most efficient way to construct this model in the program is as follows (see the figures below). First generate two sets of side arc members using the command Create > Templates > Arc Members. Then create one shell element at the top using the nodes on the arc members. Delete all generated members. Now use Create > Entities from other Entities > Shells by Nodes to generate 7 more shell elements using the existing nodes on the arcs. Lastly, use Modify > Revolve > Revolve Shells to Bricks command to generate brick elements. This method simplifies the generation procedure.





Reference

[1]. MacNeal & Harder, “A Proposed Standard Set of Problems to Test Finite Element Accuracy”, Finite Elements in Analysis and Design, 1 (1985) 3-20

Part VII

7 Dynamic

7.1 F-01 (Simple 2D Frame Vibration)

Objective

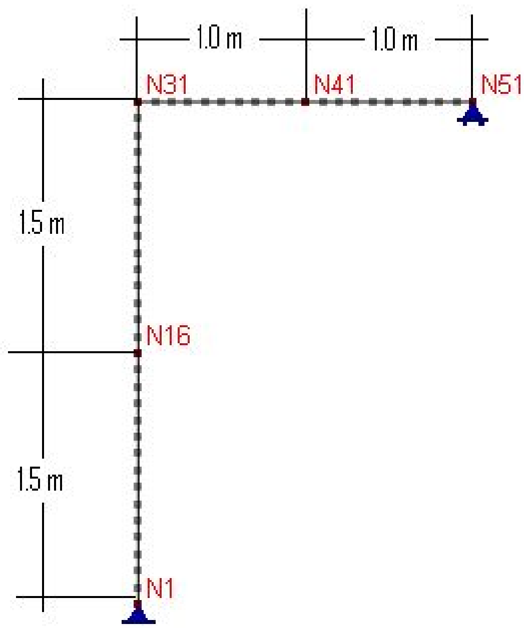
To verify the behavior of beam element vibration

Problem Description

A right-angle frame [Ref 1] vibrates under its own weight as shown below.

Material properties: E = 2e11 Pa, v = 0.29, ρ = 7860 Kg/m^3

Section properties: square section 100 x 100 mm



Finite Element Model

50 beam elements

Model type: 2D Frame (shear deformation considered)

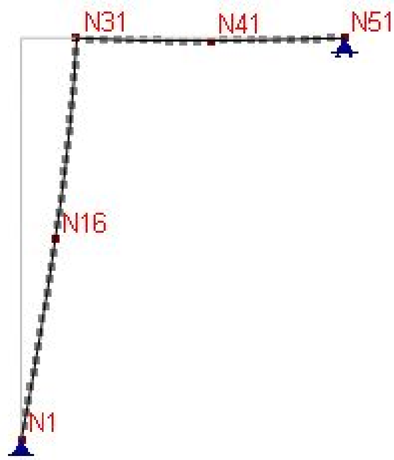
Results

The mode frequencies are given by [Ref 1]

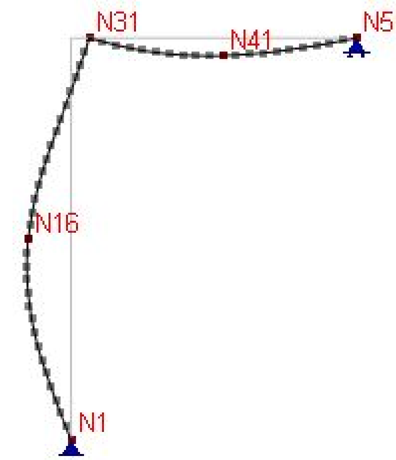
Unit: mode frequency - Hz

Mode Frequency	ENERCALC 3D	[Ref 1]
Mode 1	3.331	3.315
Mode 2	35.07	35.08
Mode 3	70.60	70.77
Mode 4	122.6	122.7
Mode 5	225.7	226.0
Mode 6	269.0	269.4
Mode 7	395.7	396.6
Mode 8	420.7	420.8
Mode 9	552.2	552.3
Mode 10	650.1	649.6

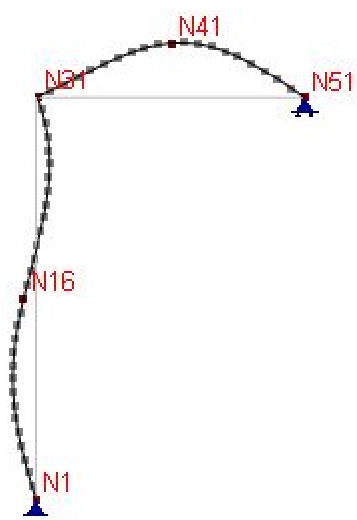
First Four Mode Shapes:



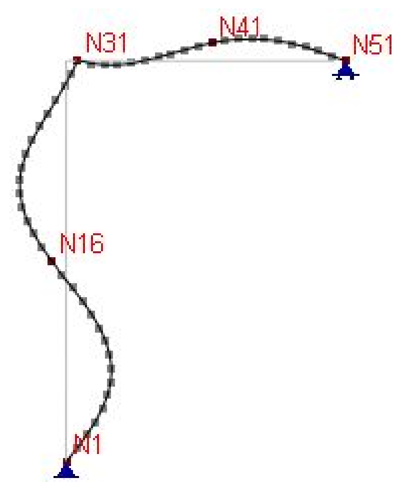
Mode Shape 1



Mode Shape 2



Mode Shape 3



Mode Shape 4

Comments

The vibration frequencies given by ENERCALC 3D are very close to the referenced values.

Reference

[1]. Cook, Malkus, Plesha, Witt, “Concept and Applications of Finite Element Analysis” 4th Edition, pp436, John Wiley & Sons, Inc., 2002

7.2 F-02 (2D Truss Vibration)

Objective

To verify the behavior of truss element vibration

Problem Description

The 2D truss structure [Ref 1] shown below vibrates under its own weight. Nodal coordinates in meters are shown in parenthesis.

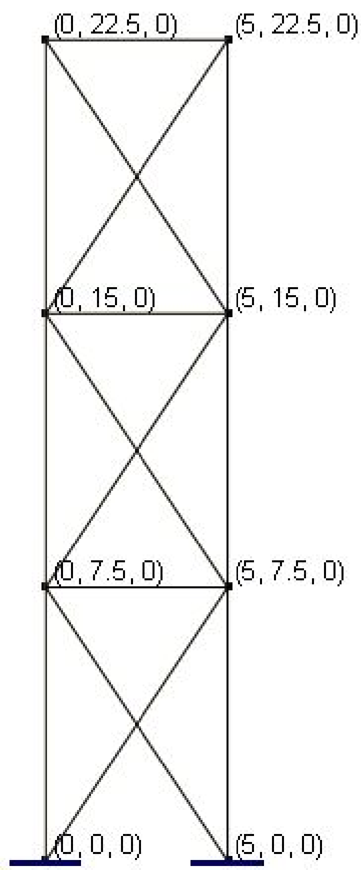
Material properties: E = 7.17e10 N/m^2, v = 0.30, ρ = 2768 Kg/m^3

Section cross-sectional areas

Vertical trusses: 8.0e-5 m^2

Horizontal trusses: 6.0e-5 m^2

Diagonal trusses: 4.0e-5 m^2



Finite Element Model

15 beam elements

Model type: 2D Truss

Results

The mode frequencies are given by [Ref 1]

Unit: mode frequency – Hz

Mode Frequency	ENERCALC 3D	[Ref 1]
Mode 1	7.9822	7.9832
Mode 2	27.9952	28.0012
Mode 3	44.8770	44.8815
Mode 4	49.5731	49.5859
Mode 5	94.9018	94.925
Mode 6	116.3799	116.3882
Mode 7	125.6432	125.6551
Mode 8	126.1574	126.1727
Mode 9	132.1162	132.1308
Mode 10	152.2912	152.3021

Comments

The vibration frequencies given by ENERCALC 3D are very close to the referenced values.

Reference

[1]. Stejskal, Dehombreux, Eiber, Gupta, Okrouhlik, “Mechanics with Matlab” April 2001
Web: http://www.geniemeca.fpms.ac.be/mechmatHTML/

7.3 F-03 (Cantilevered Tapered Membrane Vibration)

Objective

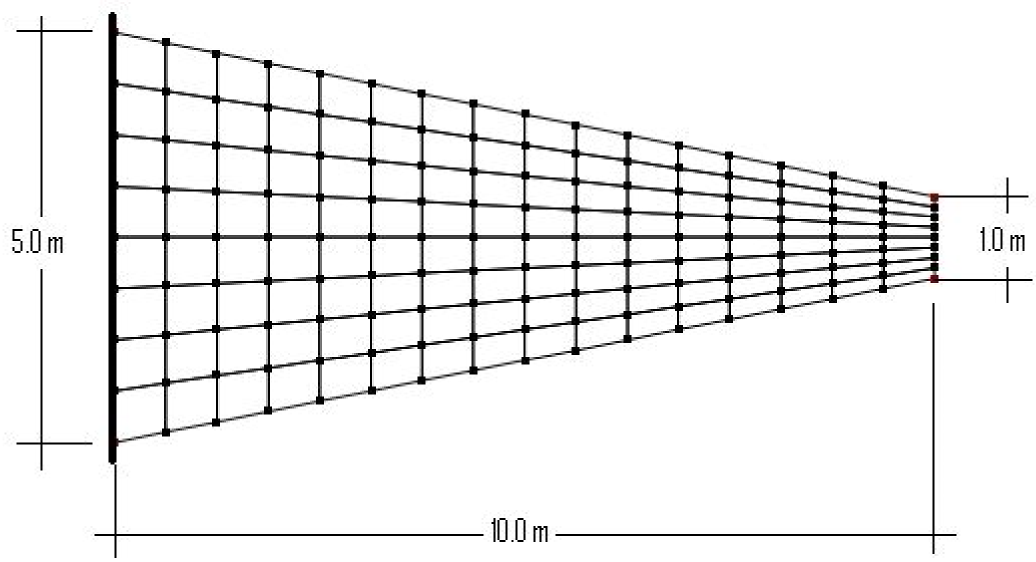
To verify the behavior of membrane plate vibration

Problem Description

The cantilevered tapered membrane plate [Ref 1] shown below vibrates under its own weight.

Material properties: E = 2.0e11 Pa, v = 0.30, ρ = 8000 Kg/m^3

Plate thickness:, t = 0.05 m



Finite Element Model

128 shell elements

Model type: 2D Plane Stress

Results

The mode frequencies are given by [Ref 1].

Unit: mode frequency – Hz

Mode Frequency	ENERCALC 3D		[Ref 1]
	Compatible Membrane	Incompatible Membrane	
Mode 1	44.7076	44.4487	44.623
Mode 2	130.3669	129.2843	130.03
Mode 3	162.4766	162.4449	162.70
Mode 4	246.2847	243.6222	246.05
Mode 5	378.4229	373.7379	379.90
Mode 6	389.4256	389.2006	391.44

Comments

The vibration frequencies given by ENERCALC 3D are very close to the referenced values.

Reference

[1]. Abbassian, Dawswell, Knowles “Selected Benchmarks for Natural Frequency Analysis”, Test No. 32, NAFEMS Finite Element Methods & Standards, Nov. 1987

7.4 F-04 (Cantilever Plate Vibration)

Objective

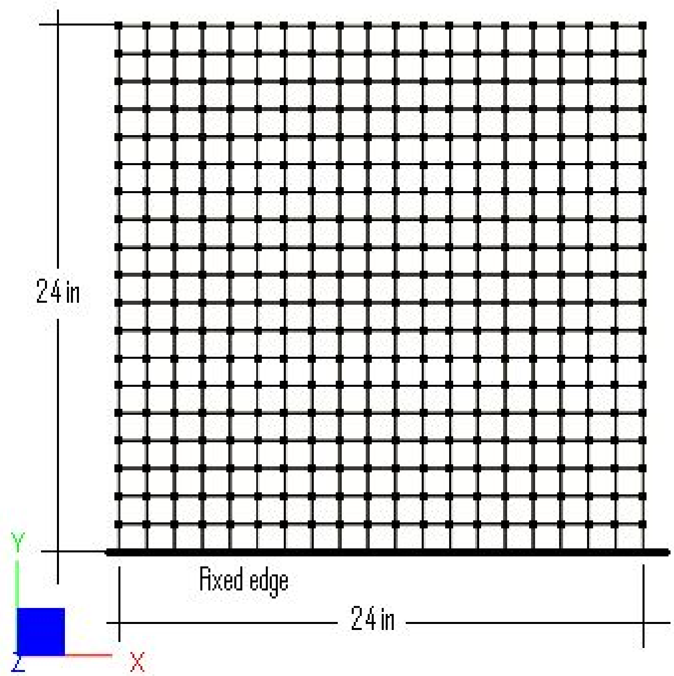
To verify the behavior of plate bending vibration

Problem Description

The 24 x 24 in cantilever plate [Ref 1] shown below vibrates under its own weight.

Material properties: E = 2.95e+007 psi, v = 0.20, density = 0.28356 lb/in^3

Plate thickness: t =1 in



Finite Element Model

361 shell elements (19 x 19 mesh)

Model type: 2D Plate Bending

Results

The mode frequencies are given by [Ref 1].

Unit: mode frequency – Hz

Mode Frequency	ENERCALC 3D		[Ref 1]
	MITC4 Thick Plate	Kirchhoff Thin Plate	
Mode 1	0.0176	0.0175	0.01790
Mode 2	0.0070	0.0069	0.00732
Mode 3	0.0028	0.0028	0.00292
Mode 4	0.0023	0.0022	0.00228
Mode 5	0.0019	0.0019	0.00201

Comments

The vibration frequencies given by ENERCALC 3D are very close to the referenced values.

Reference

[1]. Harris, Crede “Shock and Vibration Handbook”, McGraw-Hill, Inc, 1976

7.5 F-05 (Cantilever Brick Vibration)

Objective

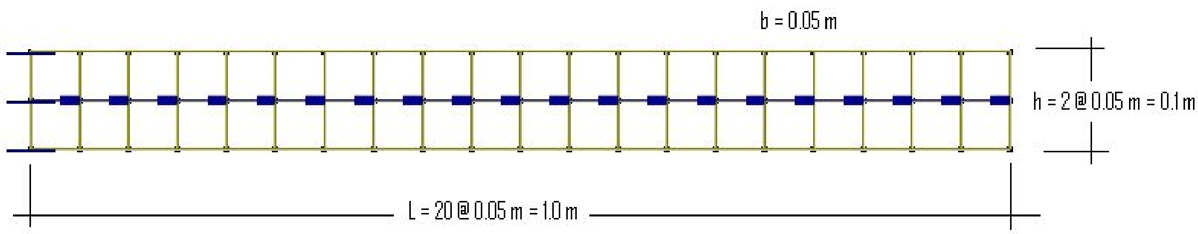
To verify the behavior of brick element vibration

Problem Description

A 1.0 m long cantilever beam fixed at the left end as shown below vibrates under its own weight.

Material properties: E = 2.0e11 N/m^2, v = 0, density = 7800 kg/m^3

Beam section: b x h = 0.05 x 0.1 m



Finite Element Model

40 brick elements (20 x 2 x 1 mesh)

Model type: 3D Brick

Boundary conditions
- Fixed Dx, Dy and Dz for nodes at left end
- Fixed Dx for nodes along the middle line
- Fixed Dz for all nodes

Results

The theoretical mode frequencies may be calculated as follows [Ref 1]:

$$f_n = \frac{K_n}{2\pi L^2}\sqrt{\frac{EI}{m}} = \frac{K_n}{2\pi(1.0)^2}\sqrt{\frac{2.0e^{11}*\frac{1}{12}*0.05*0.1^3}{7800*0.05*0.1}} = 23.26468652*K_n$$

Where K_1 = 3.51602; K_2 = 22.0345; K_3 = 61.6972

Unit: mode frequency – Hz

Mode Frequency	ENERCALC 3D		Theoretical
	Compatible Brick	Incompatible Brick	
Mode 1	86.0831	81.1984	81.80
Mode 2	517.9047	489.7797	512.6
Mode 3	1370.6341	1300.4777	1435.4

Comments

The first and second vibration frequencies given by ENERCALC 3D are close to the theoretical ones. More elements need to be used to get accurate third and higher frequencies.

The boundary conditions are chosen such that out-of-plane and axial directions are suppressed so we can concentrate on the behavior of in-plane vibration.

Reference

[1]. Chopra, “Dynamics of Structures” 2nd Edition, pp 679, Prentice Hall, Inc., 2001

7.6 F-06 (2D Steel Frame Vibration)

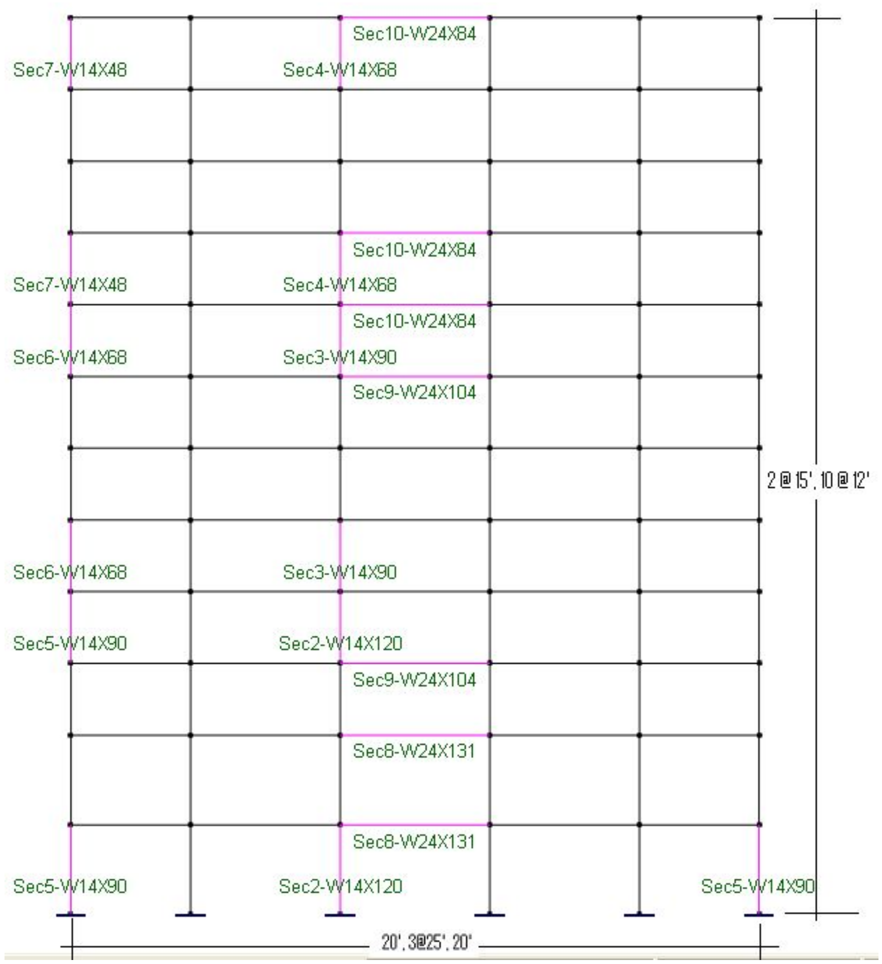
Objective

To verify the behavior of the beam element in large 2D steel frame vibration

Problem Description

A 5-span, 12-story 2D steel frame vibrates under its own weight as shown below. All beams are W24's and all columns are W14's

Material properties: E = 29000 ksi, v = 0.3, density = 483.84 lb/ft^3



Interior columns:
- Floor 1 – 4: W14x120
- Floor 5 – 8: W14x90
- Floor 9 – 12: W14x68

Exterior columns:
- Floor 1 – 4: W14x90
- Floor 5 – 8: W14x68
- Floor 9 – 12: W14x48

Beams:
- Floor 1 – 4: W24x131
- Floor 5 – 8: W24x104
- Floor 9 – 12: W24x84

Units: Iz, Iy and J – in^4, A, Ay and Az – in^2

Section	Iz	Iy	J	A	Ay	Az
W14X120	1380	495	9.37	35.3	8.555	23.03
W14X90	999	362	4.06	26.5	6.16	17.1583
W14X68	722	121	3.01	20	5.81	12
W14X48	484	51.4	1.45	14.1	4.692	7.96308
W24X131	4020	340	9.5	38.5	14.8225	20.64
W24X104	3100	259	4.72	30.6	12.05	16
W24X84	2370	94.4	3.7	24.7	11.327	11.5757

Finite Element Model

132 beam elements

Model type: 2D Frame (shear deformation included)

Results

The first three natural frequencies are compared with another program, Frame Analysis & Design (STRAAD) [Ref. 1].

Units: Hz

	ENERCALC 3D	Frame Analysis & Design (STRAAD)
Mode 1	1.7508	1.7402386
Mode 2	4.6904	4.6629050
Mode 3	7.9692	7.9228372

Comments

The results given by ENERCALC 3D are very close to the referenced values.

Reference

[1]. “Frame Analysis & Design”, Digital Canal Corporation, Dubuque, Iowa, USA

7.7 F-07 (3D Frame Vibration)

Objective

To verify the behavior of the beam element in large 3D frame vibration

Problem Description

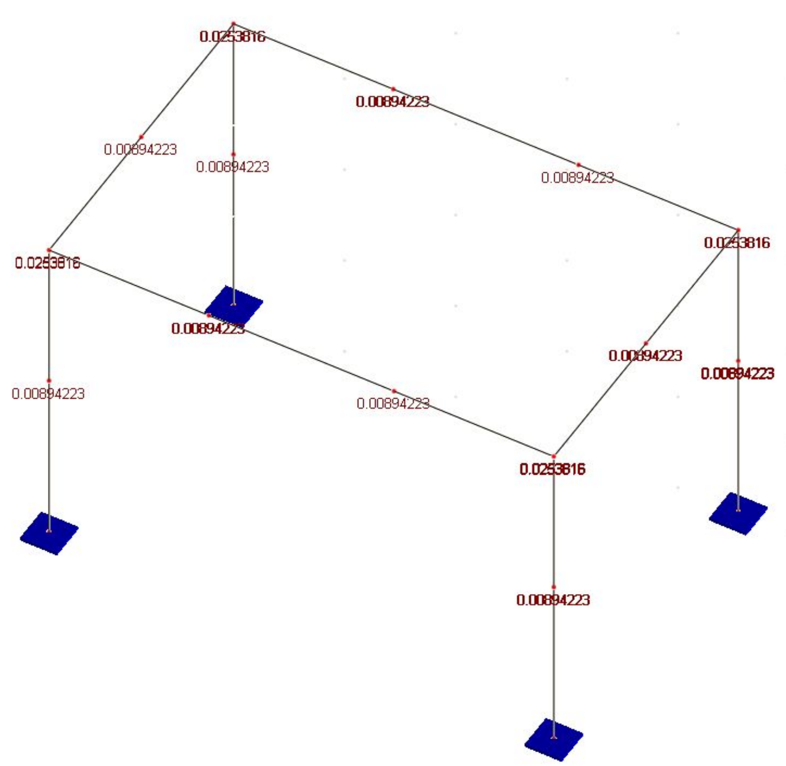
A 3D single story frame structure with a length = 27.25 in, width = 17.25 in and height = 18.625 in, is fixed at the bottom. Nodes are inserted at 8.625 in from the top corner nodes along the length, width and height.

Material: E = 2.79e+007 lb/in^2, v = 0.3

Sections: A = 1.07453 in^2, A_y = A_z = 0.537266 in^2, I_z = I_y = 0.665747 in^4, J = 1.33149 in^4

Masses: Corner nodes = 0.0253816 lb-sec^2/in (X, Y and Z directions)

All other nodes except supports: 0.00894223 lb-sec^2/in (X, Y and Z directions)



Finite Element Model

18 beam elements

Model type: 2D Frame (shear deformation included)

Results

The first 10 natural frequencies are compared with another independent program Larsa [Ref. 1].

Units: Hz

	ENERCALC 3D	Larsa
Mode 1	111.2088	111.21
Mode 2	115.7695	115.77
Mode 3	137.1354	137.13
Mode 4	215.7477	215.74
Mode 5	404.1712	404.16
Mode 6	422.5145	422.50
Mode 7	451.4604	451.45
Mode 8	548.8147	548.80
Mode 9	733.3148	733.29
Mode 10	758.2787	758.26

Comments

The results given by ENERCALC 3D are very close to the referenced values.

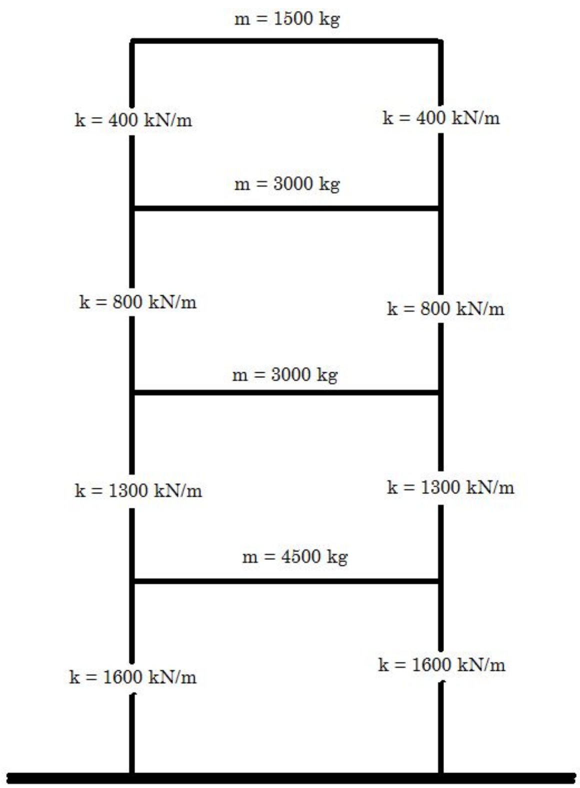
7.8 F-08 (Response Spectrum Analysis of 4 Story Shear Building)

Objective

To verify the results of response spectrum analysis of a shear building using beam elements

Problem Description

A 4-story shear building [Ref 1] with corresponding mass and stiffness info shown below.

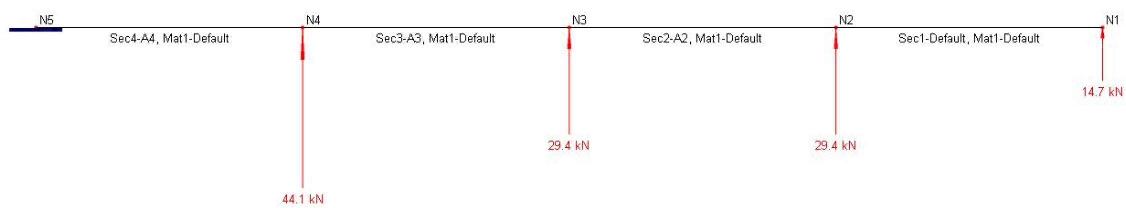


The response spectrum is defined below (from Create > Generate Loads > Response Spectra Library menu).

Period (sec)	Spectral Acceleration (g)
0.0	0.15
0.1	0.18
0.2	0.25
0.3	0.38
0.4	0.50
0.5	0.50
0.6	0.40
0.8	0.32
1.0	0.25
1.2	0.19

We will use four 1m steel beam elements, with sectional area of A1 = 4 mm^2, A2 = 8 mm^2, A3 = 12 mm^2 and A4 = 16 mm^2. E = 199.948 KN/mm^2, v = 0.3. The axial stiffness EA/L will match the shear building column stiffness. We will use very large values for other section properties such as Iz, Iy, J, Ay, Az. This effectively allows us to focus beam element behavior in axial direction only.

We will apply vertical loads F1 = 14.7 KN, F2 = 29.4 KN, F3 = 29.4 KN and F4 = 44.1 at the four free nodes. These forces will be converted to equivalent masses by the program during frequency/response spectrum analysis.



Results

The following lists different results by ENERCALC 3D against the reference [Ref. 1].

Vibration Periods (sec)

	ENERCALC 3D	Reference
Mode 1	0.5788	0.5789
Mode 2	0.2594	0.2595
Mode 3	0.1873	0.1873

Modal Displacements (cm)

Node	ENERCALC 3D			Reference		
	Mode 1	Mode 2	Mode 3	Mode 1	Mode 2	Mode 3
1	5.1930E+00	-3.9987E-01	5.8228E-02	5.19545e+00	-4.00257e-01	5.82355e-02
2	4.0459E+00	3.9837E-02	-6.4594E-02	4.04779e+00	3.98752e-02	-6.46441e-02
3	2.5786E+00	2.1588E-01	1.0244E-02	2.57982e+00	2.16095e-01	1.03938e-02
4	1.2207E+00	1.7499E-01	4.5731E-02	1.22125e+00	1.75157e-01	4.56209e-02

Inertia Forces (N)

Node	ENERCALC 3D			Reference		
	Mode 1	Mode 2	Mode 3	Mode 1	Mode 2	Mode 3
1	9.1746E+03	-3.5167E+03	9.8232E+02	9.18127e+03	-3.52106e+03	9.83037e+02
2	1.4296E+04	7.0070E+02	-2.1794E+03	1.43063e+04	7.01546e+02	-2.18364e+03
3	9.1113E+03	3.7973E+03	3.4565E+02	9.11798e+03	3.80201e+03	3.55155e+02
4	6.4698E+03	4.6169E+03	2.3145E+03	6.47450e+03	4.62254e+03	2.30532e+03

Base Shear Forces (N)

	ENERCALC 3D	Reference
Maximum Likely (SRSS)	3.9478e+004	3.951e+04
Maximum Possible (ABSSUM)	4.6113e+004	4.614e+4

Comments

The results given by ENERCALC 3D are very close to the referenced values. We did not enter nodal masses directly. Therefore, we need to make sure nodal forces are converted to masses before frequency analysis (Analysis > Frequency Analysis).

Reference

[1]. “Earthquake Response Spectrum Analysis of 4 Story Shear Building”, 1996, Mark Austin, Department of Civil Engineering, University of Maryland

7.9 F-09 (Response Spectrum Analysis of 2D Frame)

Objective

To verify the results of response spectrum analysis on a 2D frame.

Problem Description

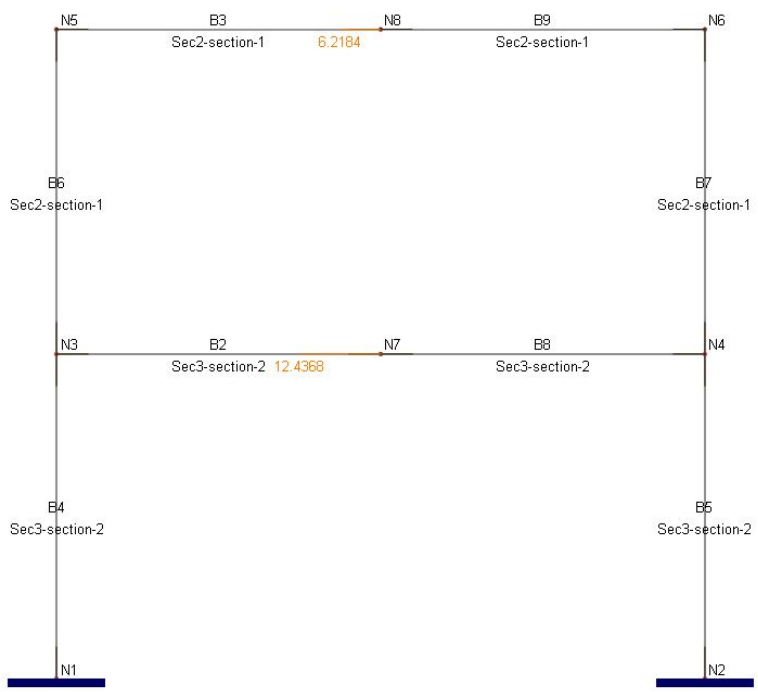
A 2-story concrete frame shown below [Ref 1] fixed at the bottom is subjected to ground motion characterized by the design spectrum specified.

Geometry: bay distance = 20 ft, each story height = 10 ft.

Material: E = 3000 ksi.

Section-1: Iz = 1000 in^4; Section-2: Iz = 2000 in^4. Other section properties are set to very large values to simulate bending only actions.

Masses: first floor center = 12.4368 kip-sec^2/ft in X direction, second floor center = 6.2184 kip-sec^2/ft in X direction.



The design response spectrum is defined below (from Create > Generate Loads > Response Spectra Library menu).

Period (sec)	Spectral Acceleration (g)
0.000	0.500
0.030	0.500
0.125	1.355
0.587	1.355
0.660	1.355
1.562	0.576
4.120	0.218
10.000	0.037

Results

The following lists different results by ENERCALC 3D against the reference [Ref. 1].

Time Periods (sec)

	ENERCALC 3D	Reference
Mode 1	1.5621	1.562
Mode 2	0.5868	0.5868

Modal Displacements SRSS combination (in)

	ENERCALC 3D	Reference
First story	7.576e+000	7.566
Second story	1.884e+001	18.81

Bending Moment (kip-ft)

Element	Location	ENERCALC 3D		Reference	
		Mode 1	Mode 2	Mode 1	Mode 2
First Floor Beam	Left End	-815.6	-56.54	-814	-57
Second Floor Beam	Left End	-396.9	178.5	-396	179
Bottom Column	Top End	425.9	372.9	425	374
	Bottom End	969.6	410.6	968	412
Top Column	Top End	396.9	-178.5	396	-179
	Bottom End	389.7	-316.4	389	-317

Comments

The results given by ENERCALC 3D are very close to the referenced values. The bending moments are from load combinations INERTIA_LOADCOMB_X_MODE_1 and INERTIA_LOADCOMB_X_MODE_2 which are generated automatically during the response spectrum analysis process.

Reference

[1]. pp 562, “Dynamics of Structures – Theory and Applications To Earthquake Engineering”, 2001, Second Edition, by Anil K. Chopra, Prentice Hall.

7.10 F-10 (Response Spectrum Analysis of 3D Frame)

Objective

To verify the results of response spectrum analysis on a 3D frame

Problem Description

A 2-story 3D frame shown below [Ref 1] fixed at the bottom is subjected to ground motion characterized by constant 0.4g for all modes, with 5% damping.

Geometry: X direction = 2 x 35 ft; Y direction = 2 x 13 ft; Z direction = 2 x 25 ft.

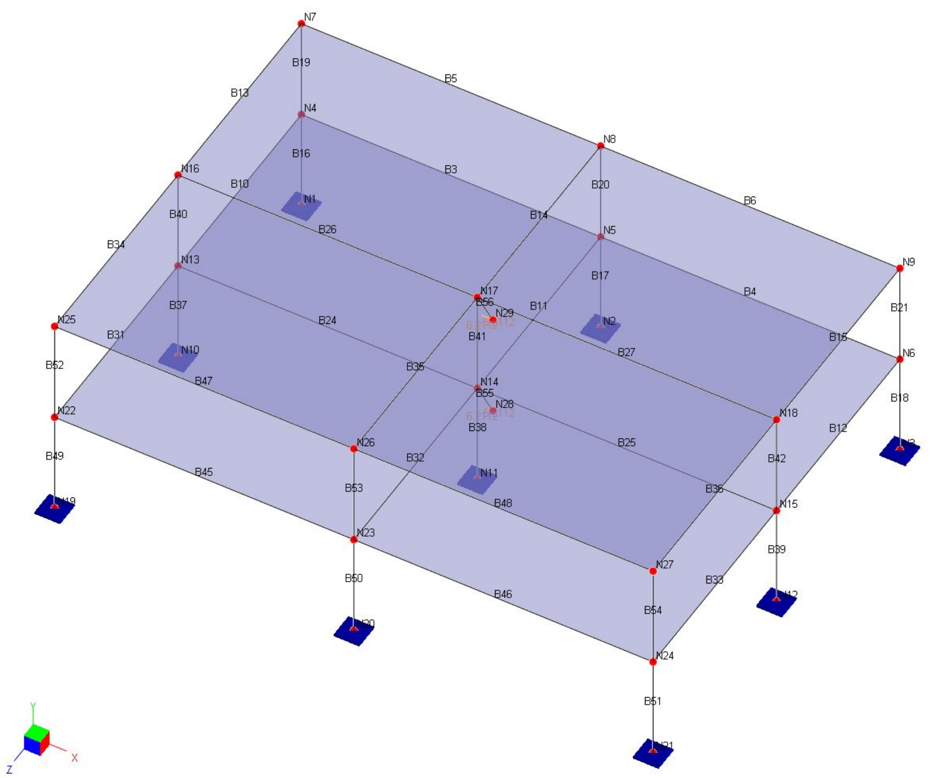
Columns: E = 350,000 k/ft^2. A = 4 ft^2, Iz = 1.25 ft^4, Iy = 1.25 ft^4, J = 1.25 ft^4, Ay =Az = 0 ft^2

Beams: E = 500,000 k/ft^2. A = 5 ft^2, Iz = 2.61 ft^4, Iy = 1.67 ft^4, J = 1.25 ft^4, Ay =Az = 0 ft^2

Two additional nodes 28 (38, 13, 27) and 29 (38, 26, 27) are placed on the first and second floors as they are the center of masses for the respective floors.

Masses: 6.2112 k-sec^2/ft at nodes 28 and 29 (X and Z directions).

To prevent nodes 28 and 29 from being orphaned nodes, we will add two additional beams with very small section properties (Iz = Iy = J = 1e-5 ft^4, A = 1e-5 ft^2) to connect node 28 with node 14 (or any other node on the first floor) and nodes 29 with node 17 (or any other node on the second floor).



Results

The following lists different results by ENERCALC 3D against the reference [Ref. 1].

	ENERCALC 3D	Reference
Mode 1 period (sec)	0.2269	0.2271
Mode 2 period (sec)	0.2152	0.2156
Mode 3 period (sec)	0.0733	0.0733
Mode 4 period (sec)	0.0719	0.0720
X displacement at node 29 ABSSUM modal combination (ft)	0.02045	0.02050
X displacement at node 29 SRSS modal combination (ft)	0.02010	0.02012
X displacement at node 29 CQC modal combination (ft)	0.02011	0.02014

Comments

The results given by ENERCALC 3D are very close to the referenced values. This verification problem also confirms the robustness of rigid diaphragm implementation. Due to program limitation, we have to add couple of weak beams on the floors to prevent center-of-mass nodes (node 28 and 29) from being orphaned.

Reference

[1]. Example 1-024, Sap2000 Software Verification Manual, 2007, Computers and Structures, Inc., Berkeley, California.

Part VIII

8 Concrete Design

8.1 G-01 (Flexural Design of Concrete Beams)

Objective

To verify the design of the rectangular and Tee concrete beams

Problem Description

The following concrete beams are to be designed according to ACI 318-02 code. The flange width and thickness are given in parenthesis for Tee beams.

Beam	B x H [B_f x T_f] (in)	f_c (ksi)	f_y (ksi)	d_t (in)	d' (in)	M_u (ft-kips)
1	10 x 16	4	60	13.5	2.5	123.2
2	14 x 23	4	60	20.5	2.5	516
3	10 x 21.5 [30 x 2.5]	4	60	19.0	2.5	227
4	10 x 21.5 [30 x 2.5]	4	60	19.0	2.5	400
5	10 x 22.5	3	40	20.0	2.5	129
6	11 x 25	3	60	22.5	2.5	403
7	10 x 20	4	60	16.0	2.5	211

Finite Element Model

7 beam elements with appropriate material and design criteria assigned

Model type: 2D Frame

Results

The design results of these beams are compared with the references.

Beam	ENERCALC 3D		References			
	As	As'	As	As'	Reference	Page
1	2.41	0	2.40	0	Ref [1]	pp. 7-23
2	6.59	1.44	6.58	1.43	Ref [1]	pp. 6-30
3	2.77	0	2.77	0	Ref [1]	pp. 7-33
4	5.10	0	5.10	0	Ref [1]	pp. 7-35
5	2.37	0	2.37	0	Ref [2]	pp. 133
6	4.66	1.31	4.74	1.20	Ref [2]	pp. 191
7	3.48	0.70	3.48	0.70	Ref [3]	pp. 102

Comments

The results given by ENERCALC 3D are very close to the referenced values. The model consists of multiple simply supported beams. Nodal moments of opposite signs are applied to nodes to achieve uniform moments in each member. The program is very versatile to design multiple isolated beams as well as to design members in integrated frames.

Reference

[1]. “Notes on ACI 318-02 Building Code Requirements for Structural Concrete”, 8th Edition, Portland Cement Association, 2002

[2]. James G. MacGregor & James K. Wight, “Reinforced Concrete – Mechanics and Design”, 4th Edition, Pearson Prentice Hall, 2005

[3]. Arthur H. Nilson, David Darwin, Charles W. Dolan, “Design of Concrete Structures”, 13th Edition, McGraw-Hill Higher Education, 2004

8.2 G-02 (Shear Design of Concrete Beams)

Objective

To verify the shear design the rectangular and circular concrete beams (columns)

Problem Description

The following concrete beams (columns) are to be designed according to ACI 318-02 code. The concrete cover to stirrup is 1.5 inches.

Beam	Dimension (in)	f_c (ksi)	f_{ys} (ksi)	Longitudinal Bar Size	Stirrup Size	P_u (kips)	V_u (kips)
1	Rectangular 12 x 16	4	40	#6	#3	160 (compression)	20
2	Circular Diameter 14	4	40	#6	#3	10 (compression)	30

Finite Element Model

2 beam elements with appropriate material and design criteria assigned

Model type: 2D Frame

Results

The design result of the first beam element is compared with the [Ref 1]. The second beam element is a round column subjected to compression and is designed as follows:

$$V_c = 2\left(1+\frac{P_u}{2000A_g}\right)\sqrt{f_c}\,b_w d = 2\left(1+\frac{10000}{2000*\pi*7^2}\right)\sqrt{4000}(14)(0.8)(14) = 20{,}478$$ lbs

$$\phi V_c = 0.75*20.478 = 15.358$$ kips

$$s = \frac{\phi A_v f_{ys} d}{(V_u - \phi V_c)} = \frac{0.75*(0.22)(40000)(0.8*14)}{(30-15.358)*1000} = 5.05$$ in.

Note: For circular section, b_w = 2R, d = 0.8(2R) are used to compute V_c and V_s, according to ACI 318-02 11.3.3 and 11.5.7.3

The following table shows ϕV_c and required stirrup spacing for the two beam elements. The program does not round the required stirrup spacing to the practical dimension.

Beam	ENERCALC 3D		Reference / Theoretical	
	ϕV_c (kips)	s (in)	ϕV_c (kips)	s (in)
1	22.175	6.88	22.2	6.9
2	15.359	5.05	15.358	5.05

Comments

The results given by ENERCALC 3D are very close to the reference and theoretical values. The model consists of multiple simply supported beams. Nodal moments of same signs are applied to nodes to achieve uniform shears in each member.

Reference

[1]. “Notes on ACI 318-02 Building Code Requirements for Structural Concrete”, 8th Edition, pp. 12-19, Portland Cement Association, 2002

8.3 G-03 (Axial-Flexural Design of Concrete Columns)

Objective

To verify the axial-flexural design of the rectangular and circular concrete columns

Problem Description

The following concrete columns [Ref 1, 2] are to be designed according to ACI 318-02 code.

Beam	Dimension (in)	f_c (ksi)	f_{ys} (ksi)	P_u (kips)	Mux (ft-kips)	Muy (ft-kips)
1 [Ref.1]	Rectangular 16 x 16	3	60	249 (compression)	55	110
2 [Ref. 2]	Circular 26	4	60	1600 (compression)	150	0
3 [Ref. 3]	Rectangular 20 x 12	4	60	255 (compression)	63.75	127

Finite Element Model

3 beam elements with appropriate material and design criteria assigned

Model type: 3D Frame

Results

The design results are compared with the [Ref 1] and [Ref 2] in the following table.

Beam	ENERCALC 3D		Reference
	Bars	Unity Check	Bars
1	12#7 (4 on each side)	0.976	12#7 (4 on each side) or 8#8 (3 on each side)
2	13#10	0.982	12#10
3	8#9 (3 on each side)	0.915	8#9 (3 on each side)

Comments

The first column is biaxially loaded and therefore a 3D frame model is used. [Ref 1] gives 12#7 (4 on each side) bars or 8#8 (3 on each side) bars based on Equivalent Eccentricity Method and Bresler Reciprocal Load Method respectively. The program gives 12#7 bars (4#7 on each side) if trial bar size starts with #7 and bar layout uses ‘equal sides’ option. If 8#8 bars (3#8 on each side) are used, the program gives a unity check value of 1.024 (and therefore the design fails). Since the program always tries to find the first section that will pass the unity check (< 1.0), we need to limit the maximum reinforcement ratio (say 3% in this case) in order to see the unity check of the 8#8 bars (3#8 on each side) section. In addition, we also need to set the start and end bar sizes to be #8 and bar layout to be ‘equal sides’ in the column design criteria for comparison.

The second column is a circular spiral column. The program gives 13#10 bars while [Ref 2] gives 12#10. If 12#10 bars are used, the program gives a unity check value of 1.008 (and therefore the design fails). Practically speaking, 12#10 should be regarded as ok.

Each column is modeled with one 3D beam element with one support flag of 111100 (fixed in Dx, Dy, Dz and Dox) and the other support flag of 011100 (fixed in Dy, Dz and Dox). Nodal moments and forces are applied in respective directions. Since no slenderness is considered, very small effective length factors are used.

Reference

[1]. James G. MacGregor & James K. Wight, “Reinforced Concrete – Mechanics and Design”, 4th Edition, pp.529-532, Pearson Prentice Hall, 2005

[2]. James G. MacGregor & James K. Wight, “Reinforced Concrete – Mechanics and Design”, 4th Edition, pp.519, Pearson Prentice Hall, 2005

[3]. Arthur H. Nilson, David Darwin, Charles W. Dolan, “Design of Concrete Structures”, 13th Edition, pp. 278, McGraw-Hill Higher Education, 2004

8.4 G-04 (Axial-Flexural Design of Concrete Slender Columns)

Objective

To verify the axial-flexural design of the rectangular concrete column (braced)

Problem Description

The following concrete braced column [Ref 1] is to be designed according to ACI 318-02 code. The clear concrete cover to stirrup is 1.5 inches. Use fc = 4 ksi, fy = 60 ksi

Size (in)	18 x 18
Total length (ft)	13
Unbraced length (ft)	13
Effective length factor	0.87
Dead Pu (kips)	230 (compression)
Dead Mu-top (ft-kips)	2
Dead Mu-bottom (ft-kips)	-2
Live Pu (kips)	173 (compression)
Live Mu-top (ft-kips)	108
Live Mu-bottom (ft-kips)	100

Finite Element Model

1 beam elements with appropriate material and design criteria assigned

Model type: 2D Frame

Results

The following table shows some intermediate and final results during the design. The program gives comparable results with the reference [Ref 1].

	ENERCALC 3D	[Ref 1]
Cm	0.960	0.96
β_{d}.	0.499	0.50
Moment magnification factor	1.145	1.15
Pu (kips)	552.8	553
Mu (ft-kips)	200.6	201
Bars	8 # 9	4 # 10 + 4 # 9

Comments

Since this is a braced column, we do not need to perform the 2nd order analysis for the design.

Reference

[1]. Arthur H. Nilson, David Darwin, Charles W. Dolan, "Design of Concrete Structures", 13th Edition, pp. 304, McGraw-Hill Higher Education, 2004

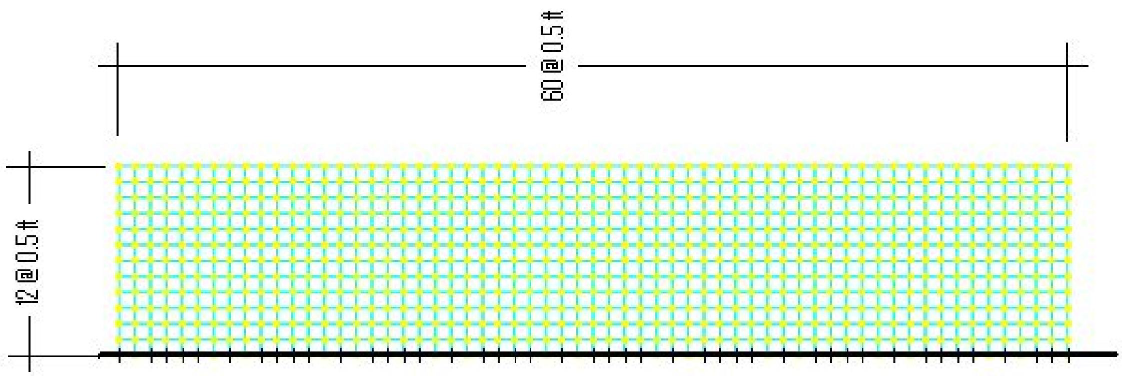
8.5 G-05 (Flexural Design of Cantilever Concrete Slab)

Objective

To verify the flexural design of the concrete slab

Problem Description

The 6 ft cantilever concrete slab shown below has a length of 30 ft and a thickness of 7.5 in. It is subjected to a uniform load of 350 lb/ft^2. Design the flexural reinforcement for the slab according to ACI 318-02 code. The concrete cover (c.c.) is 1.0 inch. Use fc = 4 ksi, fy = 60 ksi
E = 3644 ksi, v = 0.15



Finite Element Model

12 x 60 shell elements, each of which has a size of 0.5 x 0.5 ft.

Model type: 2D Plate Bending, Use Kirchhoff thin plate bending

Results

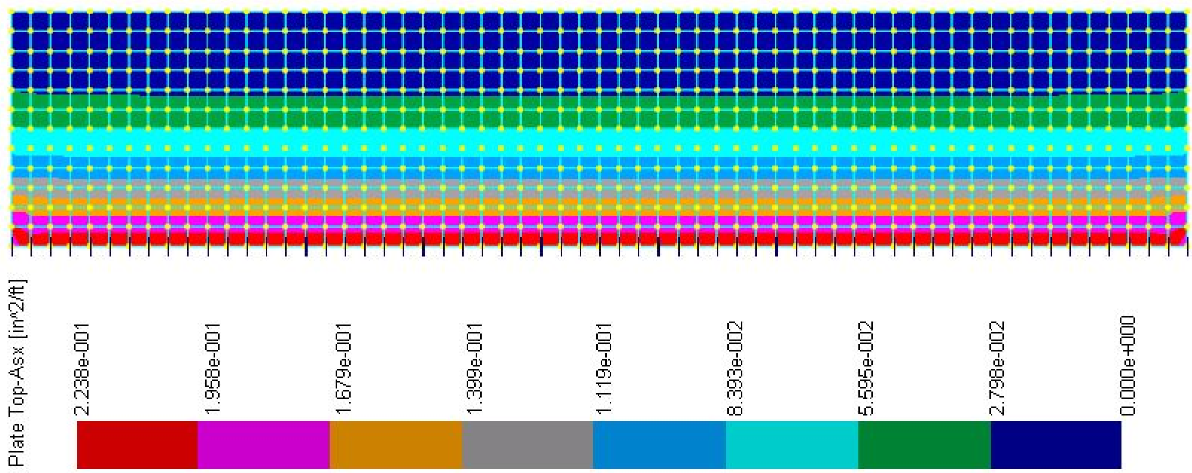
The maximum design moment (Wood-Armer moment) in top-X direction Top-Mux = -6.381 kip-ft/ft. The program gives the corresponding top-X direction steel Top-Asx = 0.2238 in^2/ft, which is consistent with the following hand calculation.

$$R_u = \frac{M_u}{\phi(bd^2)} = \frac{6.381*12*1000}{0.9*(12*6.5^2)} = 167.8\,psi$$

$$\rho = \frac{0.85 f_c'}{f_y}\left(1-\sqrt{1-\frac{2R_u}{0.85 f_c'}}\right) = \frac{0.85*4000}{60000}\left(1-\sqrt{1-\frac{2*167.8}{0.85*4000}}\right) = 0.00287$$

$$A_s = \rho(bd) = 0.00287 \; *12*6.5 = 0.22386 \text{ in}^2\text{/ft}$$

The contour (rotated) of the top steel required in X-direction is shown below.



Comments

No minimum top or bottom reinforcement is considered in this example. The Kirchhoff thin plate (instead of the MITC4 thick plate) formulation is used for analysis. This is generally recommended for models that contain only rectangular elements of thin or moderately thick plates (shells).

Reference

None

Part IX

9 Steel Design

9.1 H-01 (W Steel Beam)

Objective

To verify the steel W-shaped beam design in flexure

Problem Description

Select the lightest W section for the simply supported beam of L = 50ft, Lb = 25 ft. The superimposed load is 0.4 kip/ft dead load and 1.0 kip/ft live load.

Use A992 steel.

[Ref 1, pp 435-437].

Finite Element Model

1 beam elements with appropriate material and design criteria assigned

Model type: 2D Frame

Results

The following table shows some intermediate and final results during the design. The program gives comparable results with the reference [Ref 1].

	ENERCALC 3D	[Ref 1]
Cb	1.30073	1.30
Lp (ft)	9.36033	9.36
Lr (ft)	30.3592	30.3
Mu (ft-kips)	686.295	688
Phi-Mnx (ft-kips)	740.751	740

Reference

[1]. Charles Salmon, John Johnson and Faris Malhas, “Steel Structures” 5th Edition, Pearson Prentice Hall, 2009

9.2 H-02 (W Steel Column)

Objective

To verify the steel W-shape column design in combined axial and flexure

Problem Description [Ref .1, Example H.4]

Select an ASTM A992 W-shape with a 10-in nominal depth to carry the following load effects:

Pu = 30 kips, Mux = 90 kip-ft, Muy = 12 kip-ft.

The unbraced length is 14 ft and the ends are pinned. Cb = 1.14. The member is non-sway.

Finite Element Model

1 beam elements with appropriate material and design criteria assigned

Model type: 3D Frame

Results

The following table shows some intermediate and final results during the design. The program gives comparable results with the reference [Ref 1].

	ENERCALC 3D	[Ref 1]
Designed Section	W10x33	W10x33
B1	1.0176	1.02
Lp (ft)	6.85247	6.85
Lr (ft)	21.7757	21.8
Phi-Pn (kips)	252.522	253
Phi-Mnx (ft-kips)	136.59	137
Phi-Mny (ft-kips)	52.5	52.5
Critical Ratio	0.978576	0.979

Reference

[1]. AISC “Design Examples”, Version 14.1

9.3 H-03 (C Steel Beam)

Objective

To verify the steel channel beam capacity check in flexural and deflection

Problem Description [Ref .1, Example F.2-1A]

Check the capacity of the channel section C15x33.9 for the following beam

Simply supported L = 25 ft.

Limit the live load deflection to L/360.

The nominal loads are a uniform dead load of 0.23 kip/ft and a uniform live load of 0.69 kip/ft.

The beam is continuously braced.

Finite Element Model

1 beam elements with appropriate material and design criteria assigned

Model type: 2D Frame

Results

The following table shows some intermediate and final results during the design. The program gives comparable results with the reference [Ref 1].

	ENERCALC 3D	[Ref 1]
Mu (kips-ft)	107.813	108
Phi-Mnx (ft-kips)	137.16	137
Max live load deflection (in)	0.663206	0.664
Live load deflection limit (in)	0.833333	0.833

Reference

[1]. AISC “Design Examples”, Version 14.1

9.4 H-04 (HSS Steel Column)

Objective

To verify the steel HSS column capacity check in axial direction

Problem Description [Ref .1, Example E.10]

Check the capacity of HSS12x8x3/16 column in axial compression.

Fy = 46 ksi, L = 30 ft, Kx = Ky = 0.8, Kz = 1.0, Lu = 30 ft, Cb = 1.0.

Finite Element Model

1 beam elements with appropriate material and design criteria assigned

Model type: 3D Frame

Results

The following table shows some intermediate and final results during the design. The program gives comparable results with the reference [Ref 1].

	ENERCALC 3D	[Ref 1]
Phi-Pn (kips)	142.063	142

Reference

[1]. AISC “Design Examples”, Version 14.1

9.5 H-05 (Round HSS Steel Column)

Objective

To verify the steel round HSS column capacity check in shear

Problem Description [Ref .1, Example G.5]

Check the capacity of HSS16X0.375 column in shear.

Fy = 42 ksi, L = 32 ft

Finite Element Model

1 beam elements with appropriate material and design criteria assigned

Model type: 3D Frame

Results

The following table shows some intermediate and final results during the design. The program gives comparable results with the reference [Ref 1].

	ENERCALC 3D	[Ref 1]
Phi-Vnx (kips)	195.048	195

Reference

[1]. AISC “Design Examples”, Version 14.1

9.6 H-06 (Double Angle Steel Column)

Objective

To verify the steel double angle column axial capacity

Problem Description [Ref .1, Example E.6]

Check the capacity of 2L5x3x1/4x3/4LLBB column in axial compression.

Fy = 36 ksi, L = 8 ft, Kx = Ky = Kz = 1.0, Lux = Luy = Luz = 8 ft.

Connector distance = 32 in = 2.66667 ft.

Finite Element Model

1 beam elements with appropriate material and design criteria assigned

Model type: 3D Frame

Results

The following table shows some intermediate and final results during the design. The program gives comparable results with the reference [Ref 1].

	ENERCALC 3D	[Ref 1]
Phi-Pn (kips)	64.2975	64.3

Reference

[1]. AISC "Design Examples", Version 14.1

9.7 H-07 (WT Steel Beam)

Objective

To verify the steel WT beam flexural capacity

Problem Description [Ref .1, Example F.10]

Check the capacity of WT6x5 in flexure for the simply supported beam of L = 6 ft. The load is 0.08 kip/ft dead load and 0.24 kip/ft live load.

Use A992 steel. The beam is continuously braced.

Finite Element Model

1 beam elements with appropriate material and design criteria assigned

Model type: 2D Frame

Results

The following table shows some intermediate and final results during the design. The program gives identical results with the reference [Ref 1]. In the next few pages, we will include the step-by-step calculation procedures output by the program.

	ENERCALC 3D	[Ref 1]
Mu (kip-ft)	2.16	2.16
Phi-Mnx (kip-ft)	7.32	7.32

Reference

[1]. AISC “Design Examples”, Version 14.1

Member: 1; Critical Load Combination: Default; Critical Distance (x100 percent): 0.5000

Section Input

Section WT5X6

A = Ag = 1.77 in^2; bf = 3.96 in; tf = 0.21 in; tw = 0.19 in; d = h = 4.94 in; y = 1.36 in; Cw = 0.0255 in^6;
Zx = 2.2 in^3; Sx = 1.22 in^3; Ix = 4.35 in^4; rx = 1.57 in; Zy = 0.869 in^3; Sy = 0.551 in^3;
Iy = 1.09 in^4; ry = 0.785 in; J = 0.0272 in^4;
Using Effective Length Method; Consider Multiplier B1 for P-δ Effect
Pu = Pr = -0 kips; Mux = Mxr = 2.16 kip-ft; Muy = Myr = 0 kip-ft; Cmx = 1 ; Cmy = 1 ; Vux = 0 kips; Vuy = 0 kips;
Fy = 50 ksi; Cb = 1.13636 ; Lb = 0 ft; Kx = 1 ; Ky = 1 ; Kz = 1 ; Lx = 6 ft; Ly = 6 ft; Lz = 6 ft; Total Dy = 0 in; Live Dy = 0 in; L = 6 ft; Total Deflection Denominator = 240 ; Live Deflection Denominator = 360 ;

Axial Capacity Calculation

b = bf / 2
Unstiffened Flange b / tf = 9.42857

$$0.56\sqrt{\frac{E}{F_y}}$$

= 13.4866

$$1.03\sqrt{\frac{E}{F_y}}$$

= 24.8057

$$\frac{b}{t} \le 0.56\sqrt{\frac{E}{F_y}}$$

$$Q_s = 1.0 \qquad \text{(E7-4)}$$

= 1

Unstiffened Stem b / t = d / tw = 26

$$0.75\sqrt{\frac{E}{F_y}}$$

= 18.0624

$$1.03\sqrt{\frac{E}{F_y}}$$

= 24.8057

$$\frac{b}{t} \geq 1.03\sqrt{\frac{E}{F_y}}$$

$$Q_s = \frac{0.69E}{F_y\left(\frac{d}{t}\right)^2} \qquad \text{(E7-15)}$$

= 0.592012

Compressive strength to account for Flexural Buckling

$$\frac{K_x L_x}{r_x}$$

= 45.8599

$$\frac{K_y L_y}{r_y}$$

= 91.7197

$$\frac{KL}{r} = \max\left(\frac{K_x L_x}{r_x}, \frac{K_y L_y}{r_y}\right)$$

= 91.7197

$$F_e = \frac{\pi^2 E}{\left(\frac{KL}{r}\right)^2} \qquad \text{(E3-4)}$$

= 34.023 ksi

Q = Qs_flange * Qs_stem = 0.592012

$$4.71\sqrt{\frac{E}{QF_y}}$$

= 147.425

$$\frac{KL}{r} \leq 4.71\sqrt{\frac{E}{QF_y}}$$

$$F_{cr} = Q\left[0.658^{\frac{QF_y}{F_e}}\right]F_y \qquad \text{(E7-2)}$$

= 20.5662 ksi

$$P_n = F_{cr} A_g \qquad \text{(E3-1)}$$

= 36.4021 kips

Compressive strength to account for Flexural-Torsional Buckling
x0 = 0 in
y0 = 1.255 in

$$F_{ey} = \frac{\pi^2 E}{\left(\frac{K_y L}{r_y}\right)^2} \qquad \text{(E4-8)}$$

= 34.023 ksi

$$\bar{r}_o^{\,2} = x_o^2 + y_o^2 + \frac{I_x + I_y}{A_g} \qquad \text{(E4-11)}$$

= 4.64847 in^2

Omit term with Cw per User Note at end of AISC Specification Section E4.

$$F_{ez} = \left(\frac{\pi^2 E C_w}{(K_z L)^2} + GJ\right)\frac{1}{A_g \bar{r}_o^{\,2}} \qquad \text{(E4-9)}$$

= 37.0257 ksi

$$H = 1 - \frac{x_o^2 + y_o^2}{\bar{r}_o^{\,2}} \qquad \text{(E4-10)}$$

= 0.661174

$$F_e = \left(\frac{F_{ey} + F_{ez}}{2H}\right)\left[1 - \sqrt{1 - \frac{4 F_{ey} F_{ez} H}{(F_{ey} + F_{ez})^2}}\right] \qquad \text{(E4-5)}$$

= 22.3996 ksi

$$4.71\sqrt{\frac{E}{QF_y}}$$

= 147.425

$$\frac{KL}{r} \le 4.71\sqrt{\frac{E}{QF_y}}$$

$$\lambda_{pf} = 0.84\sqrt{\frac{E}{F_y}}$$

= 20.2299

$$\lambda_{rf} = 1.03\sqrt{\frac{E}{F_y}}$$

= 24.8057

Web is slender

Flange compactness:

$$\lambda = \frac{b_f}{2t_f}$$

= 9.42857

$$\lambda_{pf} = 0.38\sqrt{\frac{E}{F_y}}$$

= 9.15161

$$\lambda_{rf} = 1.0\sqrt{\frac{E}{F_y}}$$

= 24.0832

Flange is non-compact

Mnx to account for Yielding

$$M_y = F_y S_x$$

= 5.08333 kip-ft

Stem in tension

$$M_p = F_y Z_x \leq 1.6 M_y \qquad \text{(F9-2)}$$

= 8.13333 kip-ft

$$M_n = M_p \qquad \text{(F9-1)}$$

= 8.13333 kip-ft

Mnx to account for Flange Local Buckling (stem in tension)

$$S_{xc} = \frac{I_x}{\bar{y}}$$

= 3.19853 in^3

$M_p = F_y Z_x$

= 9.16667 kip-ft

$\lambda_{pf} < \lambda < \lambda_{rf}$

$$M_n = M_p - (M_p - 0.7F_y S_{xc})\left(\frac{\lambda - \lambda_{pf}}{\lambda_{rf} - \lambda_{pf}}\right) \leq 1.6M_y \qquad \text{(F9-6)}$$

= 8.13333 kip-ft

Mnx = 8.13333 kip-ft

$M_{cx} = \phi_b M_{nx}$

= 7.32 kip-ft

Minor Flexural Capacity Calculation

$$M_n = M_p = F_y Z_y \leq 1.6F_y S_y \qquad \text{(F6-1)}$$

= 3.62083 kip-ft

$\lambda_{pf} < \lambda < \lambda_{rf}$

$$M_n = \left[M_p - \left(M_p - 0.7F_y S_y\right)\left(\frac{\lambda - \lambda_{pf}}{\lambda_{rf} - \lambda_{pf}}\right)\right] \qquad \text{(F6-2)}$$

= 3.58348 kip-ft

Mny to account for Lateral-Torsional Buckling = 3.58348 kip-ft

$M_{cy} = \phi_b M_{ny}$

= 3.22513 kip-ft

Flexural and Axial Interaction Calculation

$$\frac{P_r}{P_c} = \frac{P_u}{\phi_c P_n}$$

= 0

$$\frac{P_r}{P_c} < 0.2$$

$$\frac{P_r}{2P_c} + \left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}}\right) \leq 1.0 \qquad \text{(H1-1b)}$$

= 0.295082

Axial-Flexural Strength: OK

Major Shear Capacity Calculation

$A_w = dt_w$
$k_v = 1.2$
h/t_w
= 26

$1.10\sqrt{k_v E / F_y}$
= 29.02

$1.37\sqrt{k_v E / F_y}$
= 36.1431

$h / t_w \leq 1.10\sqrt{k_v E / F_y}$
$C_v = 1.0$ (G2-3)
= 1

$V_n = 0.6 F_y A_w C_v$ (G2-1)
= 28.158 kips

$\phi_v = 0.90$
$\phi_v V_n$
= 25.3422 kips

$\dfrac{V_u}{\phi_v V_n}$
= 0

Shear Strength (Major Axis): OK

Minor Shear Capacity Calculation

$A_w = b_f t_f$
$k_v = 1.2$

$h/t_w = b/t_f$

= 9.42857

$1.10\sqrt{k_v E / F_y}$

= 29.02

$1.37\sqrt{k_v E / F_y}$

= 36.1431

$h / t_w \leq 1.10\sqrt{k_v E / F_y}$

$C_v = 1.0$ (G2-3)

= 1

$V_n = 0.6 F_y A_w C_v$ (G2-1)

= 24.948 kips

$\phi_v = 0.90$

$\phi_v V_n$

= 22.4532 kips

$\frac{V_u}{\phi_v V_n}$

= 0

Shear Strength (Minor Axis): OK

Total Load Deflection Check

Total Deflection Limit = L / (Total Deflection Denominator) = 0.3 in
Total Deflection Ratio = (Total Dy) / (Total Deflection Limit) = 0

Total Load Deflection: OK

Live Load Deflection Check

Live Deflection Limit = L / (Live Deflection Denominator) = 0.2 in
Live Deflection Ratio = (Live Dy) / (Live Deflection Limit) = 0

Live Load Deflection: OK

Part X

10 Step-By-Step Examples

This part of the documentation contains example problems solved by ENERCALC 3D. They are used to demonstrate the capabilities and reliabilities of the program. They may also serve as simple tutorials for the program.

Each example contains:

- A brief description of the problem,
- Suggested steps to create the model in the program,
- Comparison of program results with theoretical or published results,
- Comments.

Many of the example problems are simple and may even be verified by hand calculations. This is deliberate because simple models are easy to construct and hand calculation is the most reliable verification method. The data files for all of the example problems are provided in the “Verifications” subdirectory under the program directory. They have the file extensions of “r3a”. You may open these files, perform the analyses, and review the results. However, in order to get yourself familiar with the program, you are strongly encouraged to create these models from scratch.

Suggested modeling steps list the major steps to create each model. These steps serve only as a guide and not an exact step-by-step procedure in the creation of the model. We trust you as an engineer to be creative in using the many different model-creation methods in the program. The General Modeling Guide on the following page is a good starting point. All examples use the default settings in the program unless specified. For example, if no load case or load combination is defined, the “Default” load case or “Default” combination will be used. No stress averaging is used for finite elements unless explicitly specified.

Result checking for each problem usually starts with displacements. The reason for this is simple. The program uses the stiffness method and therefore is displacement-based. If the displacements were wrong, nothing else would be right. Other results such as forces and moments may be more relevant or important to you as an engineer. However, they are not the primary verification parameters and are provided where applicable.

Important comments are summarized at the end of each example. They explain the modeling techniques and results.

10.1 General Modeling Guide

Activity	Menus
Set up units.	**Settings and Tools** > Units & Precisions
Define materials, sections, and thicknesses	**Create** > Member Properties > Materials (or Sections), **Create** > Shell Properties > Materials (or Thicknesses)
Construct geometry. Start with generating commands whenever possible; draw individual nodes and elements whenever necessary	**Create** > Templates > 2D Truss/Frame, Rectangular Frames, etc.; **Create** > Nodes, Members, Shells
Select nodes or elements	**Create** > Window/Point Select, Line Select, Select by Properties, Invert Selection, etc.
Hide or Show	**View** > Hide Selected, Show All, etc.
Assign materials, sections, and thicknesses	**Create (or Modify)** > Member Properties > Materials (or Sections), **Create (or Modify)** > Shell Properties > Materials (or Thicknesses)
Define boundary conditions	**Create** > Boundary Conditions > Supports, Springs, etc.
Define load cases and load combinations	**Create** > Load Cases, Load Combinations
Assign loads	**Create** > Draw Loads > Nodal Loads, Point Loads, Line Loads, Surface Loads, etc.
Assign masses	**Create** > Generate Loads > Additional Masses
Modify input data	**Modify or Tables**
Define response spectra	**Tables** > Response Spectra Library
Review Input	**View** > Display Options, Load Diagram, Quick Render
Set analysis options	**Analysis** > Analysis Options
Perform analysis	**Analysis** > Static Analysis, Frequency Analysis, Response Spectrum Analysis
Review analysis results	**Analysis Results** > Shear & Moment Diagram, Contour Diagram, Deflection Diagram, Mode Shape
View or print reports	**Reporting** > Print Options, Print Text Report, Print Current View, etc.

Tips:

1. Use Modify > Undo to undo the last step.

2. Use table input when you want to combine it with graphical input, or when you are not comfortable with graphical input.

3. Try to remember some useful keyboard shortcuts
   UP or DOWN or LEFT or RIGHT for panning
   [CTRL] + UP or DOWN or LEFT or RIGHT for zooming
   [SHIFT] + UP or DOWN or LEFT or RIGHT for rotating
   F8 for quick rendering
   ESC to clear selection or get out of trouble. Press twice if you have to.

4. Views and selections may be saved and recalled.

5. Use 128-bit floating point skyline solver for numerically sensitive structures such as one with rigid diaphragms.

10.2 Example 1: A Cantilever Beam

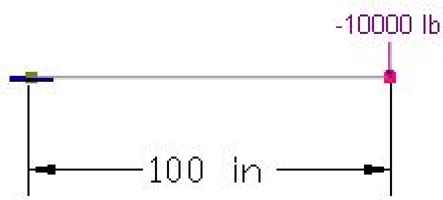
Problem Description

A 100-inch long cantilever beam is subjected to a tip load of -10,000 lbs.

Material properties: E = 2.9e7 psi, v = 0.3

Section properties: I_x = 200 in^4, A_y = 8.33333 in^2

Analyze the beam for the following two cases:
a). Model the beam with one frame element. Verify the vertical displacement and rotation at the tip of the beam, with/without the shear deformation considered.
b). Model the beam with 1,000; 10,000; 20,000; and 50,000 members. Analyze the each model with the 64-bit and 128-bit floating point solver. Compare the vertical displacements without shear deformation considered.



Suggested Modeling Steps

- Set proper units from Settings and Tools > Units & Precisions.
- Generate the beam geometry by Create > Templates > Rectangular Frames. For example, to generate 1,000 members (each with 0.1 inch in length), enter a distance list of “1000@0.1” in the X direction. Do not enter anything for the Y and Z directions.
- Select all members, define and assign the material properties by Modify > Member Properties > Materials. Make sure “Assign active material to currently selected elements” is checked in the dialog box.
- Select all members, define and assign the section properties by Modify > Member Properties > Sections. Make sure “Assign active section to currently selected members” is checked in the dialog box.
- Press ESC key to unselect all nodes and elements. Select the first node by Create > Select > Select Nodes, and assign it a fixed support by Create > Boundary Conditions > Support.
- Select the last node by Create > Select > Select Nodes, and assign it a nodal load of -10,000 lb in the global Y direction. The load is assigned to the built-in load case called “Default”. ENERCALC 3D also provides a load combination called “Default” which is 1.0 * “Default” load case by default.
- Set the analysis options by Analysis > Analysis Options. Choose the model type “2D Frame”. Check or uncheck “Consider shear deformation on members”. Select the 64-bit or 128-bit floating point skyline solver.

Results

The displacement at the tip of the beam may be calculated by hand as follows:

$$G = \frac{E}{2(1+\nu)} = 11{,}153{,}846$$ psi

$$\Delta = \frac{PL^3}{3EI} = -0.5747$$ in (shear deformation ignored)

$$\Delta = \frac{PL^3}{3EI} + \frac{PL}{A_y G} = -0.5855$$ in (shear deformation considered)

$$\theta = \frac{PL^2}{2EI} = -0.00862$$ radian

The following table shows the tip displacement and rotation of the beam modeled with one element. The comparison between the program and theoretical results is excellent.

	Without shear deformation		With shear deformation	
	ENERCALC 3D	Theoretical	ENERCALC 3D	Theoretical
Displacement	-0.5747	-0.5747	-0.5855	-0.5855
Rotation	-0.00862	-0.00862	-0.00862	-0.00862

The following table shows the tip displacements of the beam modeled with 1000; 10,000; 20,000; and 50,000 elements. Shear deformations are ignored. The four models are solved with the 64-bit (skyline) floating point and the 128-bit floating point solvers of the program.

Solver	Number of elements			
	1,000	10,000	20,000	50,000
64-bit floating point	-0.5748	-0.6522	-0.1534	No solution
128-bit floating point	-0.5747	-0.5747	-0.5747	-0.5747

Comments

This is probably the simplest structural model that can be solved by either hand or an analysis program. However, it could be turned into a very challenging numerical problem as shown in the example. The standard 64-bit floating point (double precision) solver, which is the predominant and only solver in almost all other analysis programs, tends to deteriorate in solution accuracy as the number of elements increases. In the example, the 64-bit floating point becomes unstable after 10,000 elements. For the model with 50,000 elements, some diagonal terms in the global stiffness matrix even become negative during factorization process. The solver has to terminate and the solution is not obtainable anymore. **Getting no results is better than getting incorrect results. Try this model on your familiar structural analysis software!**

ENERCALC 3D implements a unique 128-bit floating point (quad precision) solver that is extremely accurate and stable in solution. Its superiority is demonstrated in that it gives consistent and correct results up to 50,000 elements. You are encouraged to try even more elements to solve this problem. Just make sure you have enough computer memory to handle large models. If you generate a large model by splitting existing members, make sure you renumber the nodes after splitting to minimize the bandwidth in the model.

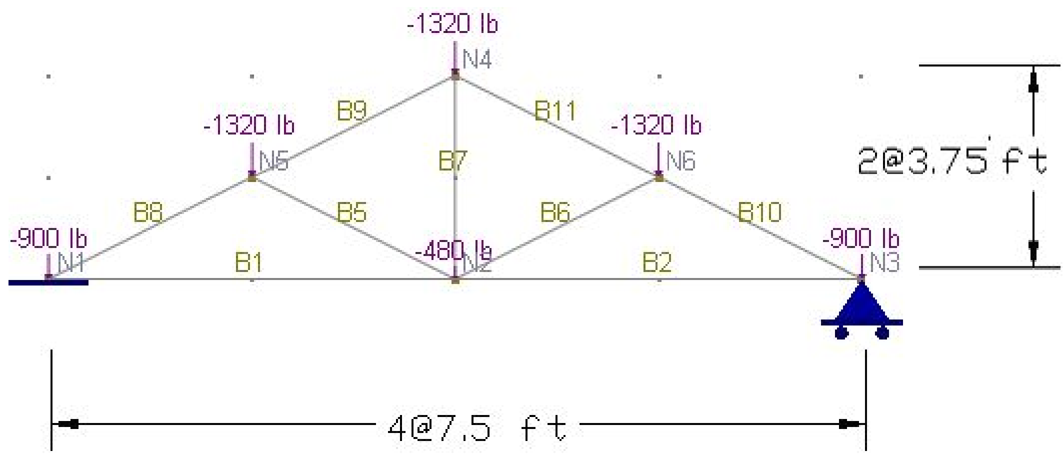
10.3 Example 2: A Truss

Problem Description

A truss with a span of 30 ft and a height of 7.5 ft is loaded with six concentrated loads at joints [Ref. 9, pp355].

Default material and section properties in the program are used.

Determine the axial forces of the truss members and the support reactions



Suggested Modeling Steps

- Set proper units from Settings and Tools > Units & Precisions.
- Generate the drawing grid by Create > Grids & Snaps > Drawing Grid Setup. Enter a distance list of “4@7.5” for the X direction and a distance list of “2@3.75” for the Y direction.
- Draw the truss members by Create > Members. Point to the intersections of the drawing grid and left-click the mouse from point to point. The drawing action is continuous. Right click the mouse to start drawing from a new location.
- Assign the nodal loads to the joints by Create > Draw Loads > Nodal Loads.
- Set the analysis options by Analysis > Analysis Options. Choose the model type “2D Truss”.

Results

The comparison between the program and the referenced results is excellent.

	ENERCALC 3D	[Ref. 9]
Chord B1 – Axial force (kips)	4.44	4.44
Chord B8 – Axial force (kips)	-4.964	-4.96
Support Reaction (kips)	3.12	3.12

Comments

No displacements are given in the reference and therefore not compared. Default material and section properties are used because the truss is determinant and the displacements are not desired.

10.4 Example 3: Linear and Non-linear Nodal Springs

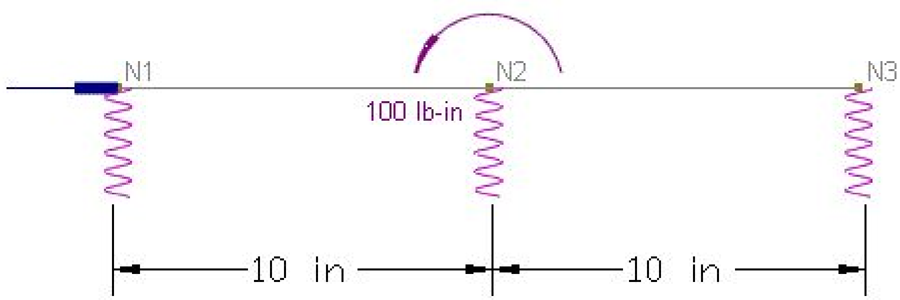
Problem Description

A 2-span continuous beam is supported by three springs. Each span is 10 inches long.

A concentrated moment M = 100 lb-in is applied at the middle spring.

Default material and section properties in the program are used.

Spring constants: K_y = 10 lb/in
The left and middle springs are linear.
Analyze the model for the following two cases.
a). The right spring is linear
b). The right spring is compression only



Suggested Modeling Steps

- Set proper units from Settings and Tools > Units & Precisions.
- Input nodal coordinates for Nodes 1, 2, 3 by Tables > Nodes
- Input the two members by Tables > Members. Use default material (=1), section (=1), and local angle (=0) for both members.
- Input the three nodal springs by Tables > Springs > Nodal Springs. Spring flags for the left and middle springs are "000000". Spring flag for the right spring is "000000" for case a) and "010000" for case b). Enter the spring constant K_y = 10 for all springs.
- Input a support at the N1 by Tables > Supports. The support has the flag of "100000" and 0s for all forced displacements.
- Input the nodal moment for N2 by Tables > Nodal Loads. Enter "5" for the load direction (OZ) and "100" for the load value.
- Set the analysis options by Analysis > Analysis Options. Choose the model type "2D Frame". Set the maximum nonlinear iterations to be "10".

Results

In case a), a force couple is developed in the left and right springs. The middle spring has a zero force. F_{couple} = M / (20 in) = 5 lb. $\Delta_{3y} = F_{couple}$ / Ky = 0.5 in.

In case b), a force couple is developed in the left and middle springs. The right spring is eliminated because it is compression-only and a positive displacement occurs at N3. F_{couple} = M / (10 in) = 10 lb. $\Delta_{2y} = F_{couple}$ / Ky = 1 in.

Displacements and spring reactions from ENERCALC 3D are shown in the following table. They are identical to the theoretical results.

	Displacements (in)			Spring reactions (lb)		
	N1	N2	N3	N1	N2	N3
Case a	-0.5	0	0.5	5	0	-5
Case b	-1	1	3	10	-10	0

Comments

The problem is linear for case a) and nonlinear for case b). The program performs 3 iterations for case b). The first iteration includes all three springs. The second iteration eliminates the compression spring. The third iteration checks for convergence.

This is a very simple problem that involves nodal springs only. More complicated problems may be solved just as easily. The program supports line and surface springs that may be applied to members and shells. Line springs may be used in modeling beams on grade and surface springs may be used in modeling mat (Winkler) foundations. Both line and surface springs may be linear or nonlinear (compression-only or tension only).

Default material and section properties are used because they do not affect the results in the example.

10.5 Example 4: A Portal Frame With P-Delta

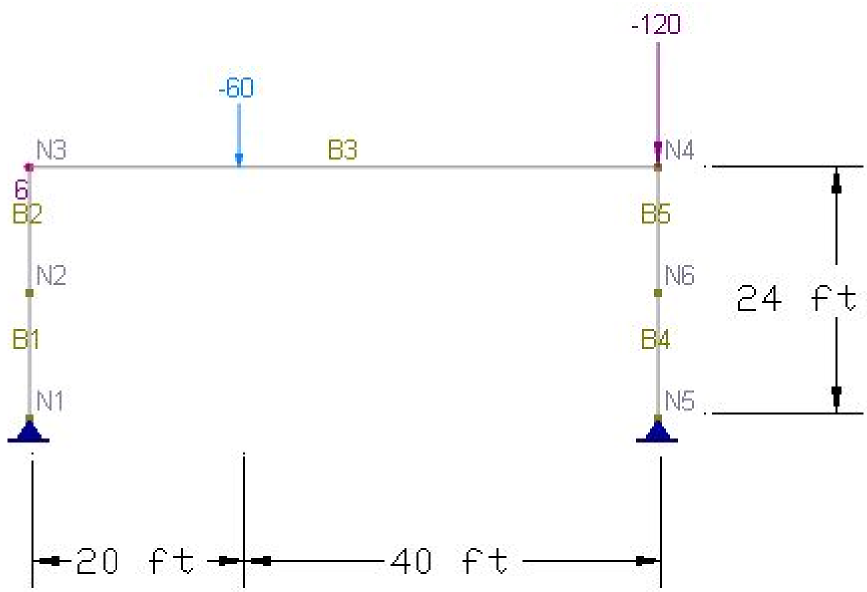
Problem Description

The following portal frame [Ref. 7, pp252] has a span of 60 ft and a column height of 24 ft. The beam is vertically loaded with 60 kips placed at 20 ft from the left end of the beam. The right column is vertically loaded with 120 kips. A horizontal load of 6 kips is applied at the joint of the beam and the left column. Each column is modeled with 2 members. The beam is modeled with a single frame element.

Columns: W10x45, A = 13.3 in^2, I_z = 248 in^4

Beam: W27x84, A = 24.8 in^2, I_z = 2850 in^4

Material: E = 2.9e7 psi, v = 0.3
Perform analysis for the following two cases:
a). First order (Linear) elastic analysis
b). Second order (P-Delta) elastic analysis



Suggested Modeling Steps

- Set proper units from Settings and Tools > Units & Precisions.
- Generate the 2D frame by Create > Templates > Rectangular Frames. Enter a distance list of “60” for the X direction and a distance list of “24” for the Y direction. Do not enter anything for the Z direction. Select “Pinned” supports at the bottom of the dialog.
- Select the lower horizontal beam generated and delete it by Modify > Delete.
- Select the two columns and split each into 2 members by Modify > Split > Split Members.
- Select all members, define and assign the material properties by Modify > Member Properties > Materials. Make sure “Assign active material to currently selected elements” is checked in the dialog box.

- Select the four columns, define and assign the column section properties by Modify > Member Properties > Sections. Make sure "Assign active section to currently selected members" is checked in the dialog box.
- Select the horizontal beam, define and assign the member section properties by Modify > Member Properties > Sections. Make sure "Assign active section to currently selected members" is checked in the dialog box.
- Assign the nodal loads and point loads of "Default" load case by Create > Draw Loads > Nodal Loads, Point Loads. Make sure you select the nodes or member beforehand.
- Create two load combinations by Create > Load Combinations. Set a load factor of 1.0 for the "Default" load case for each combination. Set the second combination to perform the P-Delta analysis.
- Set the analysis options by Analysis > Analysis Options. Choose the model type "2D Frame". Uncheck "Consider shear deformation on members".

Results

The comparison between the program and the referenced results is good.

		ENERCALC 3D	[Ref. 7]
Linear	Maximum Displacement (in)	4.387	4.4
	Max + moment in beam (in-kips)	8707.7	8708
	Max – moment in beam (in-kips)	2044.3	2044
P-Delta	Maximum Displacement (in)	8.26	8.1
	Max + moment in beam (in-kips)	9079.4	9078
	Max – moment in beam (in-kips)	2663.3	2661

Comments

The portal frame is analyzed by first order and second order elastic methods. Significant stress stiffening effect is observed. Although each physical column is modeled by 2 members, the program accounts for the P-Delta (P-Δ) effect very well even without splitting columns. However, you must split each column into more segments to account for p-delta (P-δ) effect. The same is also true when buckling analysis is desired.

The program does not perform buckling analysis directly. You may estimate the buckling load through trial-and-error with different load factors in the P-Delta load combination. The buckling load factor (λ) given by the reference [Ref. 7] is 2.2.

10.6 Example 5: Rectangular Plate

Problem Description

Two 2 x 2 inch square plates [Ref. 4, pp3-20] are clamped and simply supported along their edges respectively. Each plate is loaded with two sets of loads in two different load cases. The first set load is a point load applied at the center of the plate. The second set load is a uniform pressure applied to the entire plate. Use a 10x10 mesh.

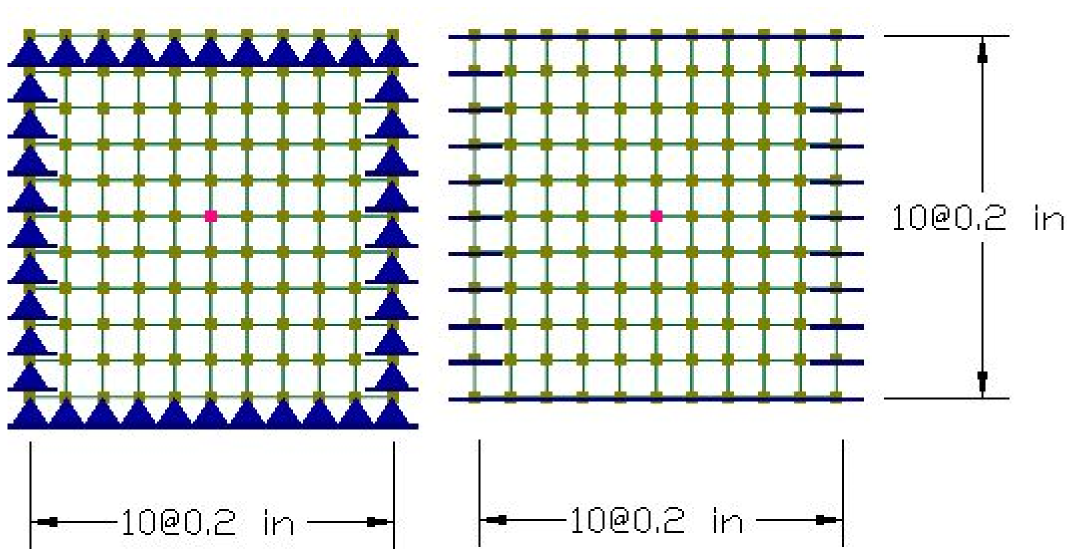
Material: E = 1.7472e7 psi; v = 0.3

Thicknesses: t = 1.0e-4 inch.

Point load P = 4e-4 lb

Uniform pressure p = 1e-4 lb/in^2

Determine the deflections at the center of plates, using both the thin Kirchhoff and the thick MITC4 plate formulations.



Suggested Modeling Steps

- Set proper units from Settings and Tools > Units & Precisions.
- Generate the first plate by Create > Templates > Rectangular Shells. Enter a distance list of “10@0.2” for the X direction and a distance list of “10@0.2” for the Y direction.
- Select all shell elements generated and copy them to a new location by Modify > Copy. Enter valid copy distances so the new plates will not overlap with the existing shells. For example, DeltaX=3, DeltaY=0, and DeltaZ = 0.
- Select all shell elements, define and assign material properties by Modify > Shell Properties > Materials. Make sure “Assign active material to currently selected elements” is checked in the dialog box.

- Select all shell elements, define and assign the shell thickness properties by Modify > Shell Properties > Thicknesses. Make sure “Assign active thickness to currently selected shells” is checked in the dialog box.
- Press ESC key to unselect all. Select the nodes along all edges of the first plate model and assign them pinned supports by Create > Boundary Conditions > Support. Select the nodes along all edges of the second plate model and assign them fixed supports by Create > Boundary Conditions > Support.
- Define two load cases named “Point” and “Uniform”.
- Define two load combinations. In the first load combination, set the load factor of 1.0 for load case “Point” and 0s for other load cases. In the second load combination, set the load factor of 1.0 for load case “Uniform” and 0s for other load cases.
- Select center nodes of the two plate models, assign them the point loads of load case “Point” by Create > Draw Loads > Nodal Loads.
- Select all shell elements, assign them the uniform loads of case “Uniform” by Create > Draw Loads > Surface Loads.
- Set the analysis options by Analysis > Analysis Options. Choose the model type “2D Plate Bending”. Check or uncheck “Use Kirchhoff thin plate bending formulation for rectangular shells”.

Results

The comparison of the deflections (inches) at the center of each plate between the program and the referenced results is excellent.

Boundary	Loading	ENERCALC 3D		[Ref. 4]
		MITC4	Kirchhoff	
Simple	Point	11.555	11.762	11.60
	Uniform	4.049	4.044	4.062
Clamped	Point	5.475	5.750	5.60
	Uniform	1.256	1.29	1.26

Comments

This is one of the standard test problems proposed to test the effectiveness of plate elements in bending [Ref. 4]. Closed form solutions exist for both plates under point and uniform loading [Ref. 5, 6]. The problem is solved using both thick (MITC4) and thin (Kirchhoff) plate bending formulations. The results from both formulations are very close and compared well with those given by the reference.

It is important to point out that the MITC4 thick plate element can be used to model both a thick plate where shear deformation may be significant and a thin plate where shear deformation is negligible. When it is used to model a very thin plate as in this example, the MITC4 produces results close to those produced by the Kirchhoff thin plate element. The MITC4 plate element is free from shear locking, and is insensitive to distortion of element geometry. It is arguably the best plate bending element currently available.

10.7 Example 6: Circular Plate On Grade

Problem Description

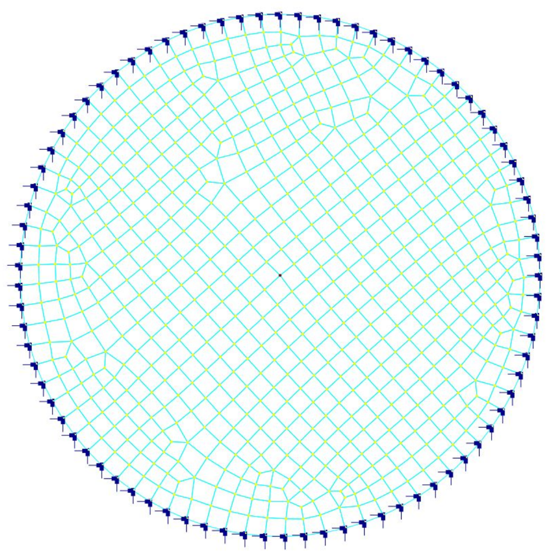
A circular steel plate with a thickness of 0.2 inch and a diameter of 20 inches is simply supported along its edge [Ref. 6 pp326-327 & pp 380-381]. The plate is loaded with a uniform load of 3 lb/in^2.

Material: E = 3e7 psi; v = 0.285

Thicknesses: t = 0.2 inch.

Determine the deflection and moment at the center for the following two cases:
a). No elastic foundation.

b). An elastic foundation with a modulus of 20 lb/in^3.



Suggested Modeling Steps

- Set proper units from Settings and Tools > Units & Precisions.
- Generate the circular plate by Create > Templates > Circular Shells. Enter a radius of 10 and segments of 80. Select “Pinned” supports along the edge.
- Select all shell elements, define and assign material properties by Modify > Shell Properties > Materials. Make sure “Assign active material to currently selected elements ” is checked in the dialog box.
- Select all plate elements, define and assign the shell thickness properties by Modify > Shell Properties > Thicknesses. Make sure “Assign active thickness to currently selected shells” is checked in the dialog box.
- Select all shell elements, assign them the surface load by Create > Draw Loads > Surface Loads.
- *For case b) only*, Select all shell elements, assign them surface springs by Create > Boundary Conditions > Springs.

- Set the analysis options by Analysis > Analysis Options. Choose the model type "2D Plate Bending". Uncheck "Use Kirchhoff thin plate bending formulation for rectangular shells".

Results

The comparison of deflections and moments (absolute values) at the center of each plate between the program and the referenced results is excellent. Moments are the same in all directions at the center.

	@ center	ENERCALC 3D	[Ref. 6]
Case a without elastic foundation	Deflection (in)	0.089	0.0883
	Moment (in-lb/in)	61.54	61.5
Case b with elastic foundation	Deflection (in)	0.064	0.0637
	Moment (in-lb/in)	43.21	43.3

Comments

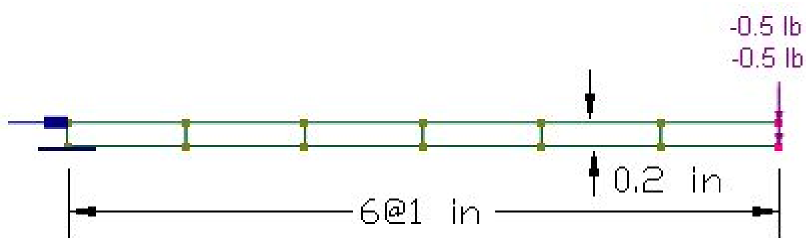
This example problem tests the reliability of the MITC4 plate bending element. It also shows how surface springs may be used to model an elastic (Winkler) foundation. Two separate models are used for case a) and case b). The generated shell elements are mostly rectangular. Some non-rectangular shell elements exist along the edge.

A relatively fine mesh is employed in order to minimize the discretization error along the edge. The default MITC4 thick plate element is used. *It is important to point out that Kirchhoff thin plate elements should not be used here due to the existence of non-rectangular elements.*

10.8 Example 7: A Cantilever Plate (In-Plane)

Problem Description

A 6 x 0.2 inch cantilever plate is loaded with two separate sets of loads [Ref. 4, pp3-20].
a). An in-plane shear of 1 lb at the tip.
b). An axial load of 1 lb at the tip.
Material: E = 1.0e7 psi; v = 0.3
Thicknesses: t = 0.1 inch.
Determine the tip displacements in the directions of applied loads, using a 6 x 1 mesh as suggested by the reference.



Suggested Modeling Steps

- Set proper units from Settings and Tools > Units & Precisions.
- Generate the first plate by Create > Templates > Rectangular Shells. Enter a distance list of “6@1” for the X direction and a distance list of “0.2” for the Y direction.
- Select all shell elements, define and assign material properties by Modify > Shell Properties > Materials. Make sure “Assign active material to currently selected elements” is checked in the dialog box.
- Select all shell elements, define and assign the shell thickness properties by Modify > Shell Properties > Thicknesses. Make sure “Assign active thickness to currently selected shells” is checked in the dialog box.
- Press ESC key to unselect all. Select the bottom-left node and assign it a fixed support. Select the top-left node and assign it a support restrained in Dx.
- Define two load cases named “InPlaneShear” and “Axial”.
- Define two load combinations. In the first load combination, set the load factor of 1.0 for load case “InPlaneShear” and 0s for other load cases. In the second load combination, set the load factor of 1.0 for load case “Axial” and 0s for other load cases.
- Select two nodes at the tip, assign each node a 0.5 lb, Y-direction nodal loads of load case “InPlaneShear” by Create > Draw Loads > Nodal Loads. Select two nodes at the tip, assign each node a 0.5 lb, X-direction nodal loads of load case “Axial” by Create > Draw Loads > Nodal Loads.
- Set the analysis options by Analysis > Analysis Options. Choose the model type “2D Plane Stress”. Check or uncheck “Use incompatible formulation for shell membrane actions or bricks”.

Results

The comparison of the displacements (inches) in the directions of loads between the program and the referenced results is mixed.

	Membrane formulation	ENERCALC 3D	[Ref. 4]
Case a)	Compatible	-0.0101	0.1081
	Incompatible	-0.1073	0.1081
Case b)	Compatible or Incompatible	3.0e-5	3.0e-5

Comments

The example problem tests the in-plane (membrane) component of the shell element. Two separate analyses are performed for case a) and case b). The incompatible membrane formulation models in-plane bending very well. The compatible membrane formulation is too stiff to model in-plane bending when a coarse mesh is used. However, both formulations work well when fine element meshes are used.

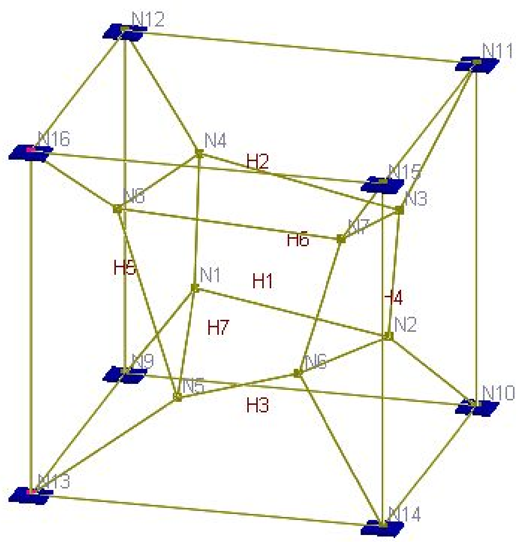
10.9 Example 8: Brick Patch Test

Problem Description

This is a patch test for a unit cube [Ref. 4 pp3-20]. The cube is modeled with 7 eight-node brick elements. Nodal coordinates, element connectivity and boundary conditions are given in the following tables. Boundary conditions are given as forced displacements. No additional loads are prescribed.

Material: E = 1.e6 psi; v = 0.25

Find stresses for each element.



Nodal coordinates (inch)

Node	X	Y	Z
1	0.249	0.342	0.192
2	0.826	0.288	0.288
3	0.85	0.649	0.263
4	0.273	0.75	0.23
5	0.32	0.186	0.643
6	0.677	0.305	0.683
7	0.788	0.693	0.644
8	0.165	0.745	0.702
9	0	0	0
10	1	0	0
11	1	1	0
12	0	1	0
13	0	0	1
14	1	0	1
15	1	1	1
16	0	1	1

Displacement field

u = 0.001 * (2x + y + z) / 2

v = 0.001 * (x + 2y + z) / 2

w = 0.001 * (x + y + 2z) / 2

Forced displacements (inch) on boundary

NODE	Dx	Dy	Dz
9	0	0	0
10	0.001	0.0005	0.0005
11	0.0015	0.0015	0.001
12	0.0005	0.001	0.0005
13	0.0005	0.0005	0.001
14	0.0015	0.001	0.0015
15	0.002	0.002	0.002
16	0.001	0.0015	0.0015

All strains are constant. For example

$$\varepsilon_x = \frac{\partial u}{\partial x} = 0.001$$

$$\varepsilon_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = 0.001$$

Element Connectivity

Element	Node1	Node2	Node3	Node4	Node5	Node6	Node7	Node8
1	1	2	3	4	5	6	7	8
2	4	3	11	12	8	7	15	16
3	9	10	2	1	13	14	6	5
4	2	10	11	3	6	14	15	7
5	9	1	4	12	13	5	8	16
6	9	10	11	12	1	2	3	4
7	5	6	7	8	13	14	15	16

Suggested Modeling Steps

- Set proper units from Settings and Tools > Units & Precisions.
- Input the nodal coordinates by Tables > Nodes.
- Modify the default material by Tables > Materials.
- Input the bricks by Tables > Bricks. Use the default material (=1).
- Input the boundary conditions by Tables > Supports. Enter the support flag “111000” for each support. Enter the forced displacements according to the table above.
- Set the analysis options by Analysis > Analysis Options. Choose the model type “3D Brick”.

Results

The comparison of stresses (psi) between the program and the referenced results is excellent. Each stress component is uniform in all seven elements.

	Sxx	Syy	Szz	Sxy	Syz	Sxz
ENERCALC 3D	1999.982	1999.982	1999.982	399.999	399.999	399.999
[Ref. 4]	2000	2000	2000	400	400	400

Comments

The brick element passes the patch test. Therefore, “the results for any problem solved with the element will converge toward the correct solution as the elements are subdivided.” [Ref. 4] The tiny differences in stresses are due to the penalty approach employed in support enforcement.

10.10 Example 9: Scodelis-Lo Roof

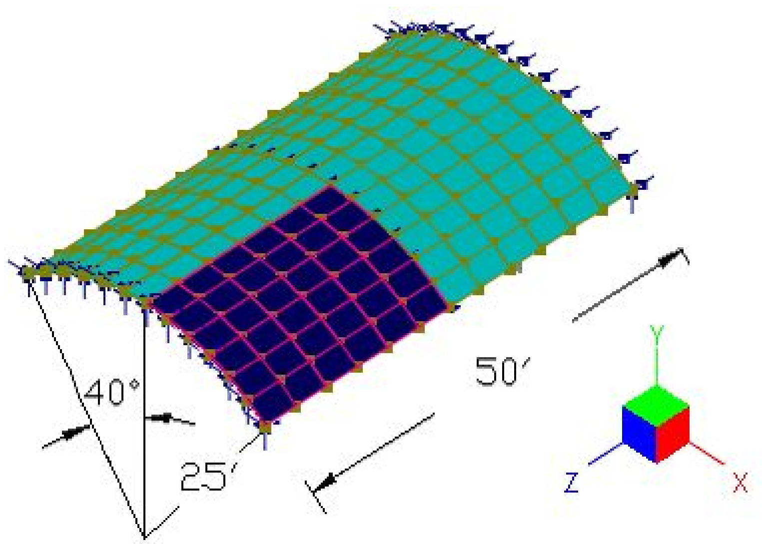
Problem Description

The Scodelis-Lo barrel roof [Ref. 4 pp3-20, Ref. 2] has a length of 50 ft, a radius of 25 ft, and a sweeping angle of 80 degrees. The roof is supported on rigid diaphragms along its two curved edges (D_x and D_y fixed, but not D_z). The two straight edges are free. A surface load of -90 lb/ft^2 in the global Y direction (self weight) is applied to the entire roof.

Material: E = 4.32e8 lb/ft^2 (3e6 psi); v = 0.0;

Thickness: t = 0.25 ft.

Find the maximum deflection and moments.



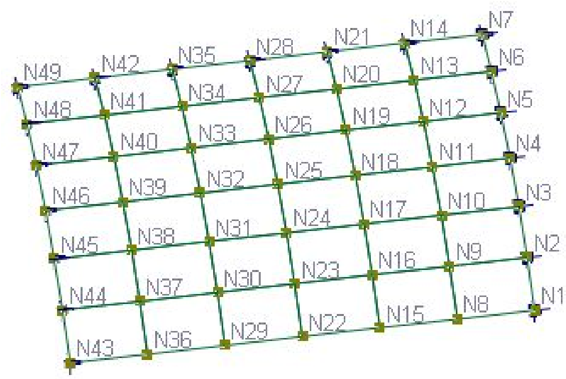
Suggested Modeling Steps

Due to the symmetry, only a quarter of the roof is modeled. A 6 x 6 mesh is used. The boundary conditions are specified in the following table.

Nodes	Fixed DOFs
N1 to N6	Z, OX, OY
N7	X, Z, OX, OY, OZ
N14, N21, N26, N35, N42	X, OY, OZ
N43 to N48	X, Y, OZ
N49	X, Y, OY, OZ

- Set proper units from Settings and Tools > Units & Precisions.

- Generate members along an arc by Create > Templates > Arc Members. Enter a radius of 25, segments of 6, start angle 50, end angle 90.
- Select all nodes and members, extrude members to shells by Modify > Extrude > Extrude Members to Shells. Enter a distance list of “6@4.1666” and direction of the global Z. Check both “Merge nodes and elements” and “Delete selected members after extrusion”.

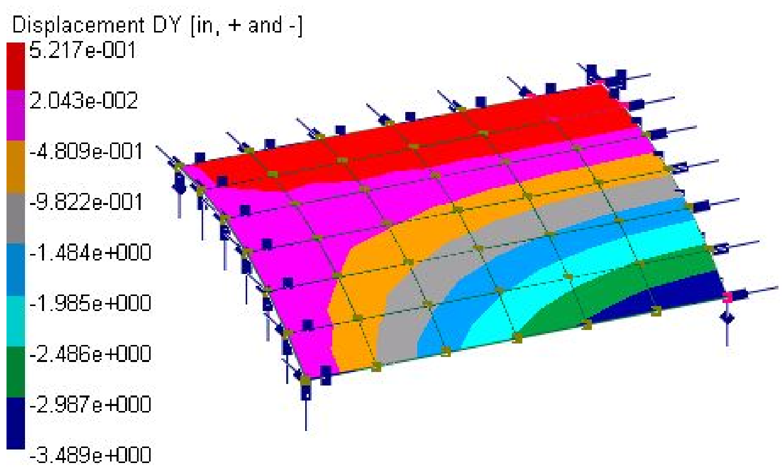


- Select all shell elements, define and assign the shell thickness properties by Modify > Shell Properties > Thicknesses. Make sure “Assign active thickness to currently selected shells” is checked in the dialog box.
- Select the boundary nodes and apply proper supports as specified above by Create > Boundary Conditions > Support. You need to select and apply multiple times.
- Select all shell elements, assign surface load by Create > Draw Loads > Surface Loads
- Set the analysis options by Analysis > Analysis Options. Choose the model type “3D Frame & Shell”. Check or uncheck “Use Kirchhoff thin plate bending formulation for rectangular shells”. Check or uncheck “Use incompatible formulation for shell membrane actions or bricks”.

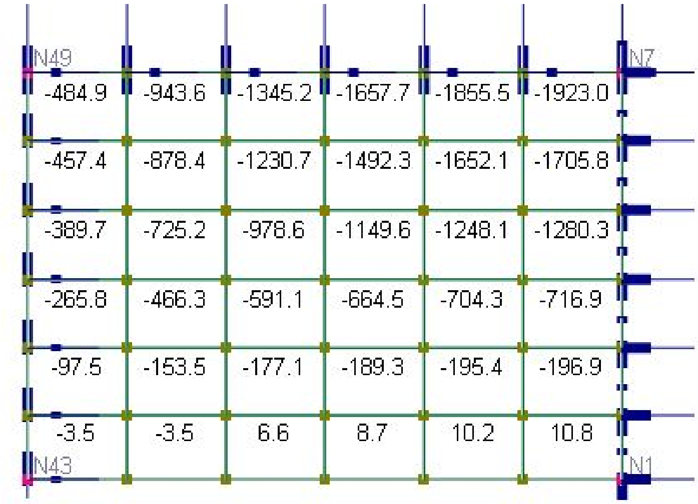
Results

The comparison of displacements and moments between the program and the referenced results is excellent. Theoretical maximum vertical displacement is given by MacNeal & Harder [Ref. 4, pp3-20]. Other theoretical values are approximate readings (with different sign convention for moments) from graphs given by Zienkiewicz [Ref. 2 pp350-351]. The maximum D_y and M_{yy} occur at the mid-point along the free edges. The maximum M_{xx} occurs at the center of the longitudinal middle section. The maximum D_z and M_{xy} occur at the corner points at supports.

Membrane	Compatible		Incompatible		References
Bending	Kirchhoff	MITC4	Kirchhoff	MITC4	
Displacement Vertical (in)	-3.475	-3.489	-3.672	-3.687	-3.629 [Ref. 4]
Displacement Longitudinal (in)	0.1317	0.1317	0.1414	0.1414	app. 0.144 [Ref. 2]
M_{xx} (ft-lb/ft)	-1954	-1923	-2093	-2056	app. 2100 [Ref. 2]
M_{yy} (ft-lb/ft)	636.0	633.9	667.9	666	app.-650 [Ref. 2]
M_{xy} (ft-lb/ft)	-1204	-1199	-1264	-1260	app. 1300 [Ref. 2]



Displacement contour (MITC4-bending, compatible formulation)



M_{xx} contour (MITC4-bending, compatible formulation)

Comments

The example is the de-facto standard test problem for shells due to the strong coupling of the bending and membrane actions. The problem is solved using the shell element with different membrane and bending formulations from which excellent results are obtained. The incompatible membrane formulation yields results closer to the referenced values.

The use of symmetry saves computing time and memory, but requires careful thinking with regard to the boundary conditions. You may model the entire roof by simply fixing D_x, D_y along the curved edges and D_z at the longitudinal central section.

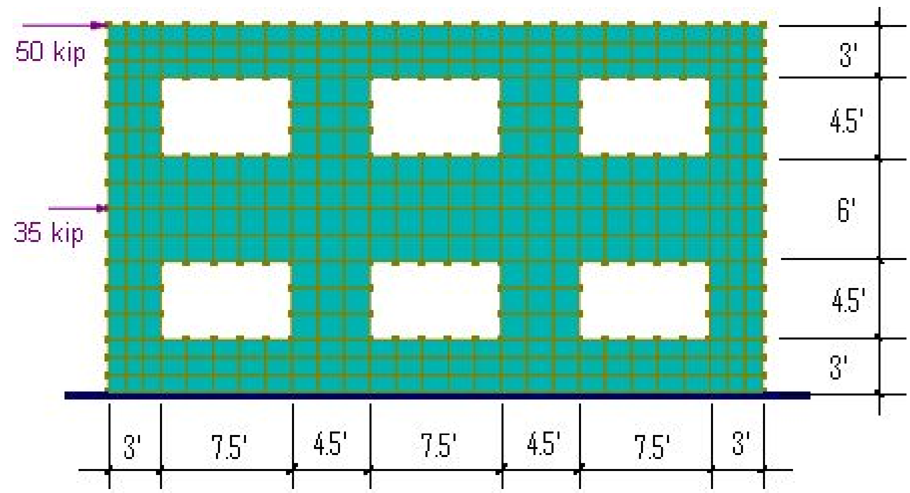
10.11 Example 10: A Shear Wall

Problem Description

A two-story concrete shear wall is subjected to two horizontal point forces at the floor levels. The wall is 37.5 ft long and 21 ft high, with six openings of 7.5 x 4.5 ft.

Material: E = 4e6 psi; v = 0.15

Thicknesses: t = 12 inch.

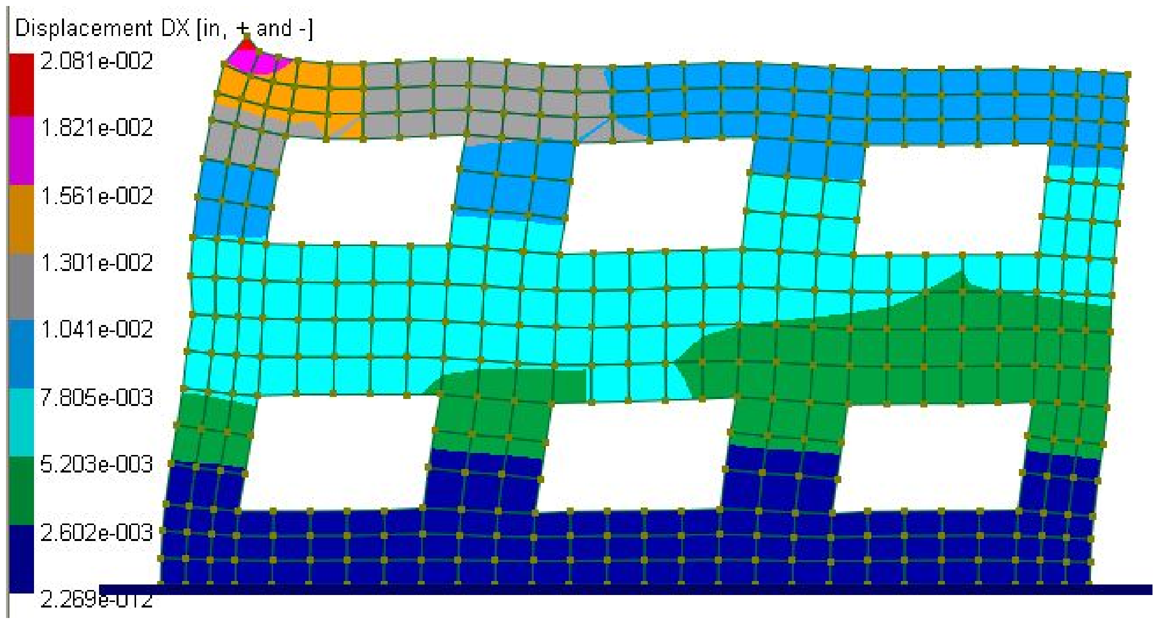


Suggested Modeling Steps

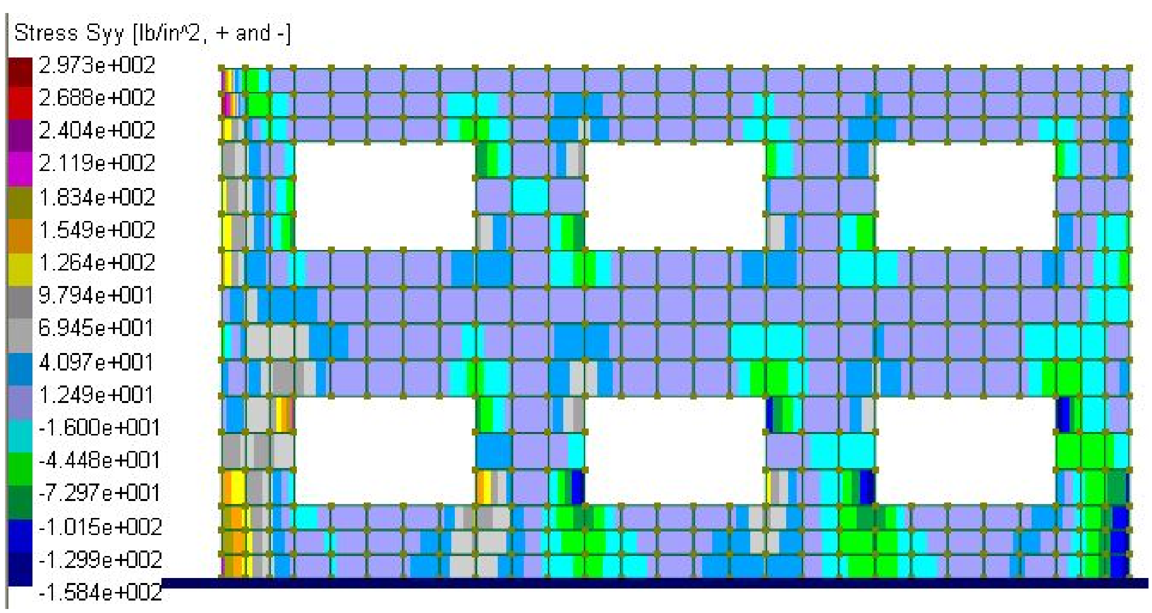
- Set proper units from Settings and Tools > Units & Precisions.
- Generate the plate by Create > Templates > Rectangular Shells. Enter a distance list of “3@1, 21@1.5, 3@1” for the X direction and a distance list of “3@1, 10@1.5, 3@1” for the Y direction.
- Select the middle eight nodes at each opening and delete them. The shells that are connected to these nodes are deleted automatically.
- Select all shell elements, define and assign material properties by Modify > Shell Properties > Materials. Make sure “Assign active material to currently selected elements” is checked in the dialog box.
- Select all shell elements, define and assign the shell thickness properties by Modify > Shell Properties > Thicknesses. Make sure “Assign active thickness to currently selected shells” is checked in the dialog box.
- Press ESC key to unselect all. Select the nodes at the bottom and assign them fixed supports.
- Select the two nodes at each story level and assign them nodal loads by Create > Draw Loads > Nodal Loads.
- Set the analysis options by Analysis > Analysis Options. Choose the model type “2D Plane Stress”. Check “Use incompatible formulation for shell membrane actions or bricks”.

Results

No comparison of results is available. Displacement D_x contour and Stress S_{yy} contour is provided in the following.



Displacement D_x contour on deflected shape

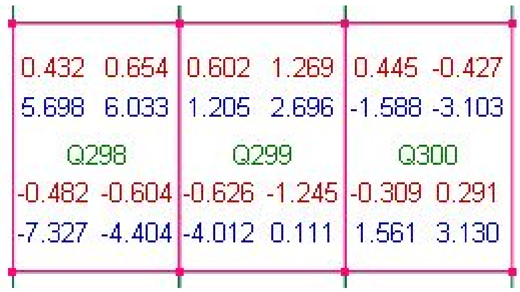


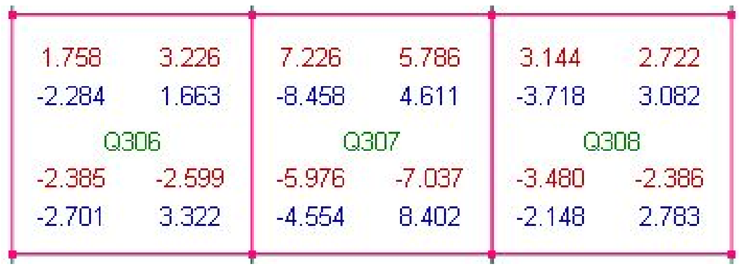
Stress S_{yy} contour

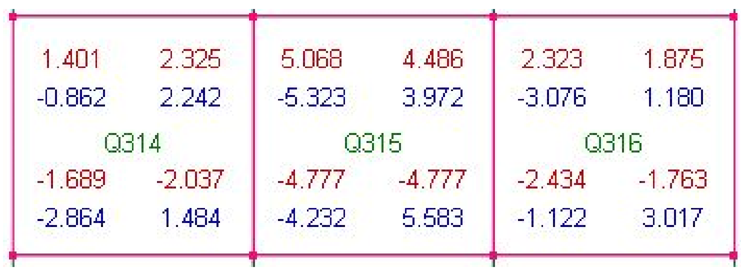
To verify the results, the horizontal shear is checked at the middle elevation of the second story openings. The following table shows the “Membrane nodal resultants” of four piers by View > Display Options (annotation mode = “Annotate selected entities” to avoid congestion of texts). You may also view the same nodal resultants in a spreadsheet by Analysis Results

> Shell Forces, Moments & Stresses > Shell Nodal Resultants. You can then copy and paste selected data to your preferred spreadsheet program to perform summation or other computations. *It is important to point out that nodal resultants are expressed in the element local coordinate systems.*

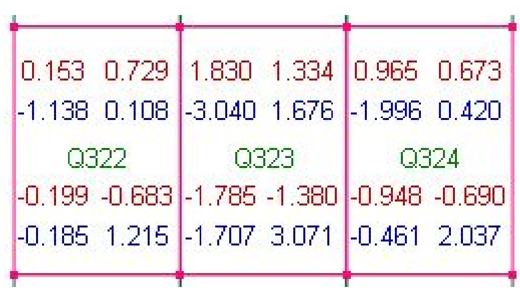
<table>
<tr><td>Pier 1

ΣF_x =
0.432 + 0.654
+ 0.602 + 1.269
+ 0.445 - 0.427
= 2.975 kips</td><td></td></tr>
<tr><td>Pier 2

ΣF_x =
1.758 + 3.226
+ 7.226 + 5.786
+ 3.144 + 2.722
= 23.862 kips</td><td></td></tr>
<tr><td>Pier 3

ΣF_x =
1.401 + 2.325
+ 5.068 + 4.486
+ 2.323 + 1.875
= 17.478 kips</td><td></td></tr>
<tr><td>Pier 4

ΣF_x =

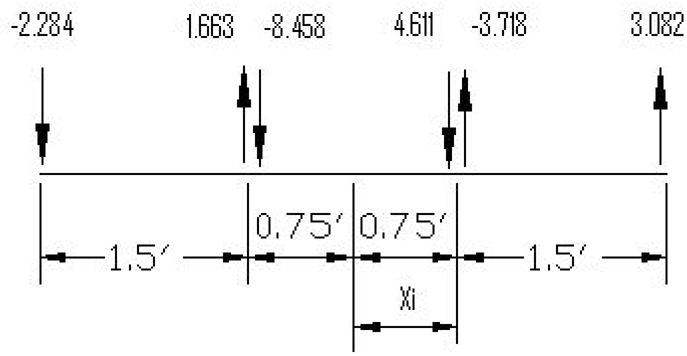
0.153 + 0.729
+ 1.83 + 1.334
+ 0.965 + 0.673
= 5.684 kips</td><td></td></tr>
<tr><td colspan="2">All Piers ΣF_x = 2.975 + 23.862 + 17.478 + 5.684 = 49.999 (app.= 50 kips)</td></tr>
</table>

Membrane nodal resultants of four piers at the middle elevation of the second story

Comments

The example problem shows how to perform structural analysis on a shear wall. Although no comparison of results is available, we demonstrate the reliability of the program by checking the horizontal shear.

In designing concrete sections, we generally need forces and moments instead of stresses. We may acquire axial forces and moments in the same manner as in shears. For example, to determine the moment at the second pier above, we may sum the moments by nodal resultants F_y about the center of the pier.



F_{yi} (kips)	X_i (ft)	$F_{yi} * X_i$ (ft-kips)
-2.284	-2.25	5.139
1.663	-0.75	-1.24725
-8.458	-0.75	6.3435
4.611	0.75	3.45825
-3.718	0.75	-2.7885
3.082	2.25	6.9345
ΣF_y = -5.104		sum = 17.8395

Internal Forces and Moment at middle of the second pier
Axial Force = -5.104 kips
Shear Force = 23.862 kips
Moment = 17.8395 ft-kips

You are encouraged to model this wall with members and compare the results with those in this example. Care should be exercised in segmenting the members and assigning them appropriate section properties. Since the sections of the members are relatively deep, shear deformations must be considered.

10.12 Example 11: Frequencies of Cantilever Beam

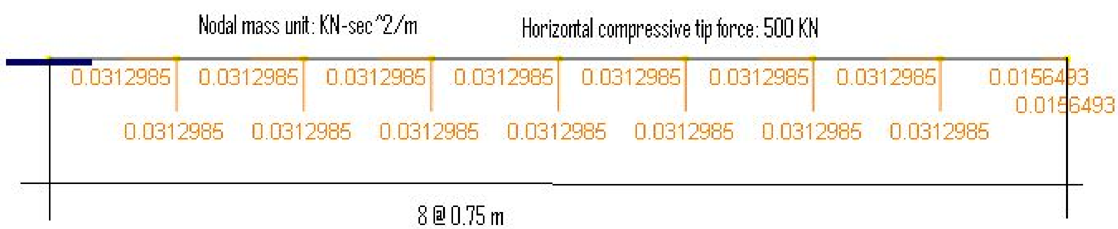
Problem Description

Analyze the vibration frequencies for the following cantilever beam (L = 6m) under its own weight.

Material properties: E = 20600 KN/cm^2, v = 0.3, weight density = 7850 Kgf/m^3

Section properties: I_x = 4079.07 cm^4, A_x = 53.1612 cm^2, A_y = A_z = 0

The beam is optionally subjected to a compressive horizontal tip load of P = 500 KN
Analyze the beam for the following two cases:
a). Find the lowest 3 frequencies without the effect of axial load
b). Find the lowest 3 frequencies with the effect of axial load



Suggested Modeling Steps

- Set proper units from Settings and Tools > Units & Precisions.
- Generate the beam geometry by Create > Templates > Rectangular Frames. For example, to generate 8 members (each with 0.75 m), enter a distance list of “8@0.75” in the X direction. Do not enter anything for the Y and Z directions.
- Select all members, define and assign the material properties by Modify > Member Properties > Materials. Make sure “Assign active material to currently selected elements” is checked in the dialog box.
- Select all members, define and assign the section properties by Modify > Member Properties > Sections. Make sure “Assign active section to currently selected members” is checked in the dialog box.
- Press ESC key to unselect all nodes and elements. Select the first node by Create > Select > Select Nodes, and assign it a fixed support by Create > Boundary Conditions > Support.
- Apply self weight by running Create > Draw Loads > Self Weights. Set self weight direction to be global Y and self weight multiplier -1.
- Set the analysis options by Analysis > Analysis Options. Choose the model type “2D Frame”. Uncheck “Consider shear deformation on members”.
- From Analysis > Frequency Analysis, check “Convert loads to masses”, set number of modes 3, number of iteration vectors 8, tolerance of eigenvalue 1e-6 and maximum number of subspace iterations 18.

For Case a), do the following steps

- Run Frequency Analysis from Analysis > Frequency Analysis

For Case b) do the following steps

- From Tables > Masses > Calculated Masses, click on “Convert to Additional Masses”. This is to avoid converting the external load to mass (although it is not necessary in this case because the load is not in the gravity direction).
- Select the last node by Create > Select > Nodes, and assign it a nodal load of -500 KN in the global X direction.
- From Create > Load Combinations, set the default load combination to “Perform P-Delta Analysis on this load combination”.
- From Analysis > Frequency Analysis, make sure “Convert loads to masses” is unchecked. Then click on Run Frequency Analysis.

Results

The frequencies without considering axial load can be calculated based with the following formulae [Ref. 14]:

$\varpi_n = \alpha_n \sqrt{\frac{EI}{mL^4}}$ and $f_n = 2\pi\varpi_n$

where m is the linear mass density

m = 7850 * 53.1612 = 41.731542 kg/m

I = 4.07907 10^{-5} m^4

L = 6 m

E = 2.06 10^{-11} N/m^2

$\alpha_1 = 3.51602$; $\alpha_2 = 22.0345$; $\alpha_3 = 61.6972$

$$\varpi_n = \alpha_n \sqrt{\frac{2.06 * 10^{11} * 4.07807 * 10^{-5}}{41.731542 * 6^4}} = 12.4646\alpha_n$$

There are no closed form formulae for calculating frequencies when axial load influence is considered. The results are therefore compared with another finite element program, AxisVM 6.0

The following table shows the first three frequencies modeled with 8 elements. The comparison between the program and theoretical results is excellent. The comparison between the program and AxisVM 6.0 is identical.

Frequency	**Without axial load considered**		**With axial load considered**	
	ENERCALC 3D	Theoretical (exact)	ENERCALC 3D	AxisVM 6.0
f_1 (Hz)	6.9255	6.98	2.6005	2.60
f_2 (Hz)	42.6551	43.71	39.4754	39.48
f_3 (Hz)	117.5983	122.39	115.0347	115.03

Comments

The comparison between the program and theoretical results is deemed excellent because we used only 8 elements for the discretization. The frequencies given by the program are lower than the exact ones. Notice the mass allocated to the support is lost in the computation. If we employed more elements, the finite element frequencies would definitely be closer to the exact continuous ones.

When axial load is considered, as in Case b, the stress-stiffness concept used by ENERCALC 3D to determine P-Delta effects is applied. In this approach, compressive axial load effectively reduces the flexural stiffness of a member (axial tension increases the flexural stiffness). With a lower stiffness, and equal mass, the frequencies are reduced.

10.13 Example 12: Frequencies of Rectangular Plate

Problem Description

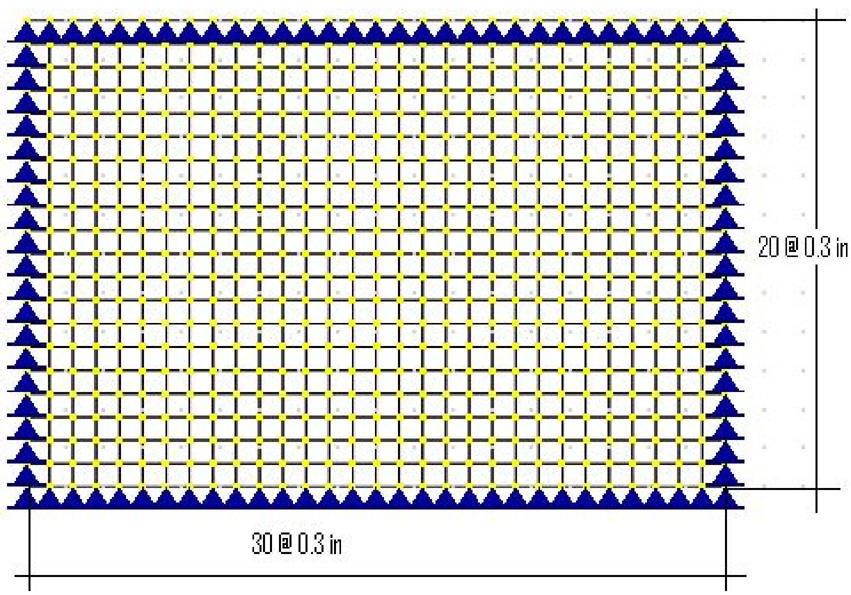
A 9 x 6 inch plate is simply supported along its edges.

Material: E = 3e7 psi; v = 0.3, weight density = 0.282938 lb/in^3

Thicknesses: t = 0.15 inch.

Use a 30x20 mesh.

Determine the first three circular frequencies of the plate, using both the thin Kirchhoff and the thick MITC4 plate formulations.



Suggested Modeling Steps

- Set proper units from Settings and Tools > Units & Precisions.
- Generate the plate by Create > Templates > Rectangular Shells. Enter a distance list of “30@0.3” for the X direction and a distance list of “20@0.3” for the Y direction.
- Select all shell elements, define and assign material properties by Modify > Shell Properties > Materials. Make sure “Assign active material to currently selected elements” is checked in the dialog box.
- Select all shell elements, define and assign the plate thickness properties by Modify > Shell Properties > Thicknesses. Make sure “Assign active thickness to currently selected shells” is checked in the dialog box.
- Press ESC key to unselect all. Select the nodes along all edges of the model and assign them pinned supports by Create > Boundary Conditions > Support.
- Apply self weight by running Create > Draw Loads > Self Weights. Set self weight direction to be global Z and self weight multiplier 1.
- Set the analysis options by Analysis > Analysis Options. Choose the model type “2D Plate Bending”. Check or uncheck “Use Kirchhoff thin plate bending formulation for rectangular shells”.

- From Analysis > Frequency Analysis, check “Convert loads to masses”, set number of modes 3, number of iteration vectors 8, tolerance of eigenvalue 1e-6, and maximum number of subspace iterations 18. Click on Run Frequency Analysis.

Results

The circular frequencies of a simply supported rectangular plate are calculated according to the following [Ref. 6]:

$$\varpi_n = \pi^2\left(\frac{m^2}{a^2} + \frac{n^2}{b^2}\right)\sqrt{\frac{Et^3}{12(1-\nu^2)\rho}}$$

where E = 3e7 psi; t = 0.15 in; v = 0.3; a = 9 in; b = 6 in; ρ = 0.282938 / 386 * 0.15 = 1.0995e-4 lb-sec^2/in^3

For m = 1, n = 1: $\varpi_1 = \pi^2\left(\frac{1^2}{9^2} + \frac{1^2}{6^2}\right)\sqrt{\frac{3e7 * 0.15^3}{12(1-0.3^2)1.0995e-4}} = 3636$ rad/sec

For m = 2, n = 1: $\varpi_2 = \pi^2\left(\frac{2^2}{9^2} + \frac{1^2}{6^2}\right)\sqrt{\frac{3e7 * 0.15^3}{12(1-0.3^2)1.0995e-4}} = 6993$ rad/sec

For m = 1, n = 2: $\varpi_3 = \pi^2\left(\frac{1^2}{9^2} + \frac{2^2}{6^2}\right)\sqrt{\frac{3e7 * 0.15^3}{12(1-0.3^2)1.0995e-4}} = 11189$ rad/sec

The comparison of the circular frequencies between the program and the theoretical results is excellent.

Circular frequencies	Thin Plate Formulation	Thick Plate Formulation	Theoretical
ϖ_1 (rad/sec)	3633	3616	3636
ϖ_2 (rad/sec)	6982	6938	6993
ϖ_3 (rad/sec)	11179	11150	11189

Comments

A relatively fine mesh is employed in this example. The thin plate finite element frequencies are closer to the theoretical results based on classical thin plate theory. The frequencies given by thick plate formulation are a little smaller than those given by thin plate formulation. This is expected because thick plate formulation accounts for shear deformation and the plate is therefore modeled with less stiffness.

10.14 Example 13: Design of Two Braced Concrete Columns

Problem Description

Two concrete columns A and B are part of a braced frame [Ref 16, pp568]. The frame is analyzed and the results of the two columns are listed below.

	Column A	Column B
Size (in)	14 x14	14 x14
Total length (ft)	20	24
Unbraced length (ft)	18	22
Effective length factor	0.77	0.86
Dead Pu (kips)	80	50
Dead Mu-top (ft-kips)	-60	42.4
Dead Mu-bottom (ft-kips)	-21	-32
Live Pu (kips)	24	14
Live Mu-top (ft-kips)	-14	11
Live Mu-bottom (ft-kips)	-8	-8

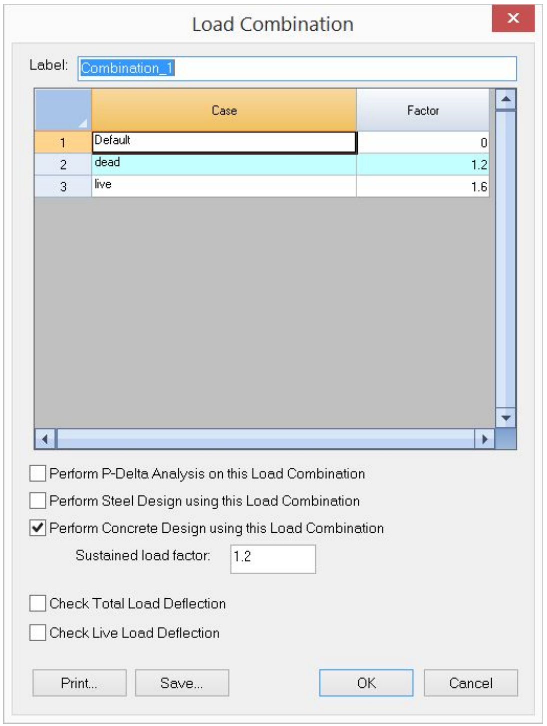
Design the columns according to ACI 318-02/05 . Use fc = 3 ksi, fy = 60 ksi

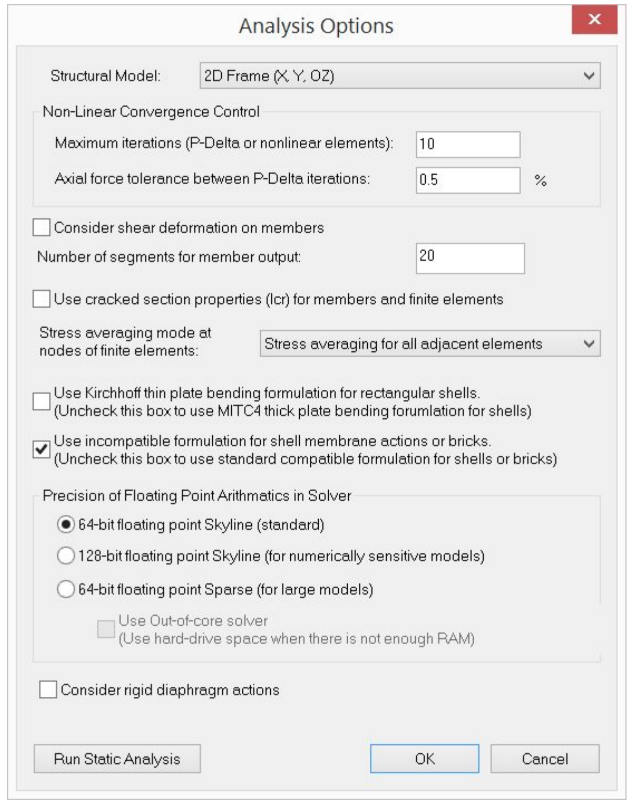
Suggested Modeling Steps

- Set proper units from Settings and Tools > Units & Precisions.
- Create two beam elements: element 1 – 20 ft, element 2– 24 ft.
- Select element 1 and 2, define and assign the standard material (Concrete fc = 3.0 ksi) by Modify > Member Properties > Materials. Make sure “Assign active material to currently selected elements” is checked in the dialog.
- Select element 1 and 2, define and assign the standard section (Rectangle 14 x 14 inch) by Modify > Member Properties > Sections. Make sure “Assign active section to currently selected members” is checked in the dialog.
- Select and assign pinned support to the start node of each member by Create > Boundary Conditions > Support. Select and assign roller support to the end node of each member by Create > Boundary Conditions > Support.
- Define Dead and Live load cases by Create > Load Cases.
- Define two load combinations: one with 1.0Dead and the other with 1.2Dead + 1.6Live. The former combination contains only the sustained load cases and will be used to calculate β_d. Combination two will be used to perform the actual design.

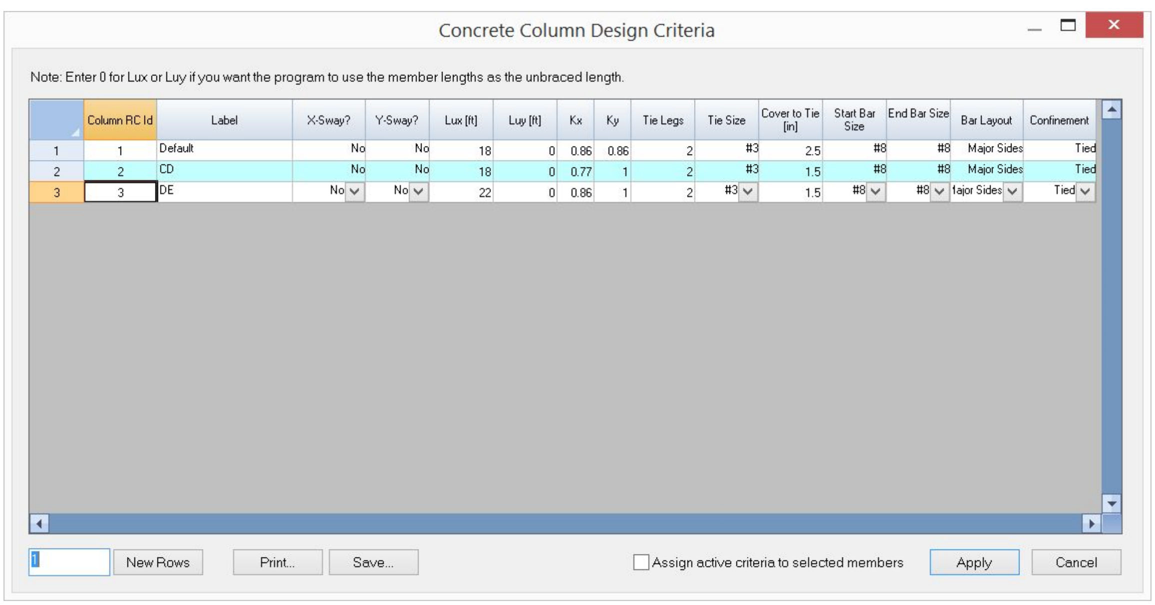
  Make sure “Perform Concrete Design using this Load Combination” is checked. Also enter sustained load factor (1.2 in this case).
- Define and apply nodal loads and moments for Dead and Live cases by Create > Draw Loads > Nodal Loads.
- Set the analysis options by Analysis > Analysis Options. Choose the model type “2D Frame”. Uncheck “Consider shear deformation on members”.

- Select ASTM_615 (English) rebar database by Concrete Design > Concrete Design Tools > Rebar Database.

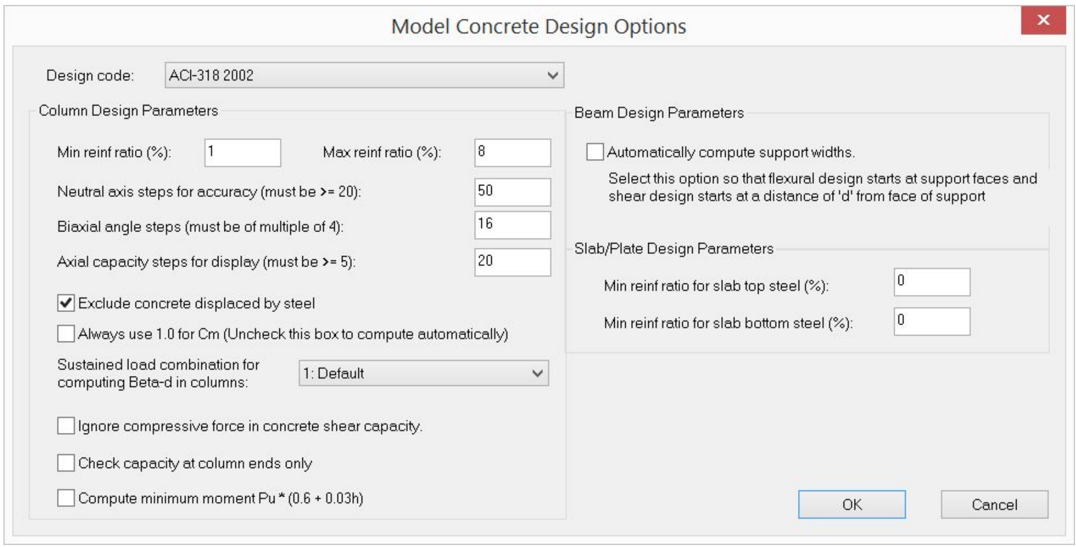




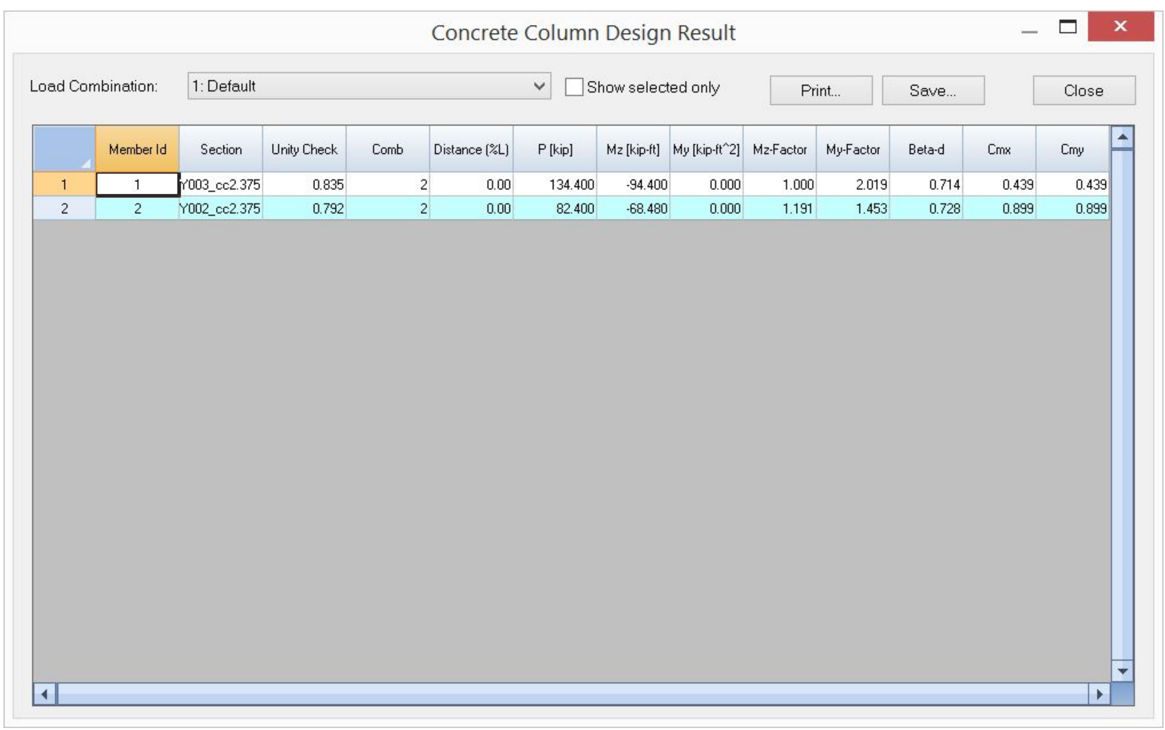
- Define and assign two column design criteria by Concrete Design > RC Design Criteria > RC Column Design Criteria. Make sure "Assign active criteria to selected members" is checked in the dialog box.



- Set model concrete design criteria by Concrete Design > RC Model Design Criteria. Make sure the sustained load combination is selected for computing β_d.
- Perform the static analysis by Analysis > Static Analysis.
- Perform concrete design by Concrete Design > Perform Concrete Design. Concrete sections will be generated automatically based on column design criteria. Exact 3D P-Mx-My capacity surfaces will be generated and are used to check against the column internal forces and moments.



- View column design results by Concrete Design > Concrete Design Output > RC Column Results. Detailed column section results such as interaction diagrams may be viewed or printed by Concrete Design > Flexural/Axial Interaction.



	Member Id	Section	Unity Check	Comb	Distance (%L)	P [kip]	Mz [kip-ft]	My [kip-ft^2]	Mz-Factor	My-Factor	Beta-d	Cmx	Cmy
1	1	Y003_cc2.375	0.835	2	0.00	134.400	-94.400	0.000	1.000	2.019	0.714	0.439	0.439
2	2	Y002_cc2.375	0.792	2	0.00	82.400	-68.480	0.000	1.191	1.453	0.728	0.899	0.899

Results

The following table shows some intermediate results during the design. The program gives comparable results with the reference [Ref 16].

	Column A		Column B	
	ENERCALC 3D	[Ref 16]	ENERCALC 3D	[Ref 16]
Cm	0.439	0.438	0.899	0.900
β_d	0.714	0.714	0.728	0.728
Moment magnification factor	1.000	1.000	1.191	1.200

The program chooses 6#8 bars for column A and 4#8 bars for column B. The reference gives 4#8 bars for both column A and column B. The program gives the unit check of 1.038 for column A if 4#8 bars were used. For practical applications, a unit check of slightly over 1.0 is probably acceptable.

Comments

This example shows the program can be used to design multiple concrete columns in a fast fashion. The loads are applied as nodal forces and moments. These loads are usually obtained from analysis results. For columns that are part of an unbraced frame, second-order analysis must be used, with consideration to stiffness adjustment according to ACI 318-02/05.

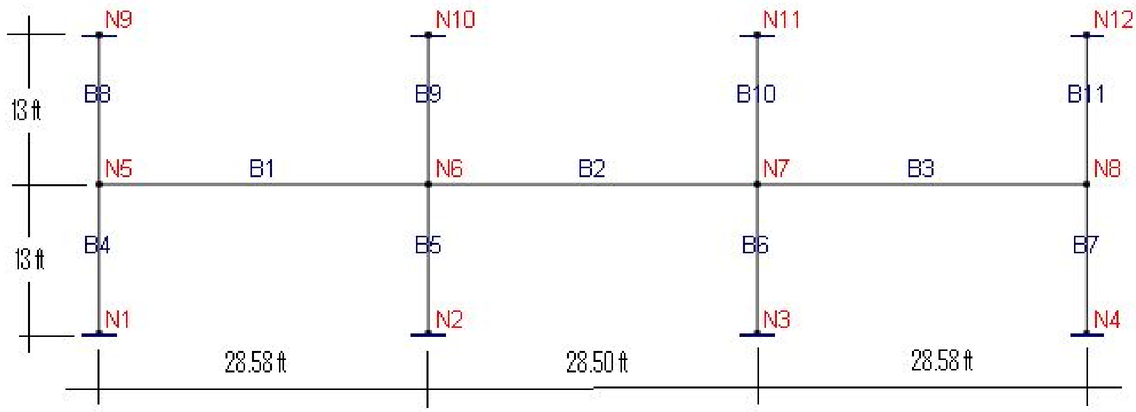
10.15 Example 14: Design of a Continuous Concrete Beam

Problem Description

The following sub-frame [Ref 17, pp 7-43], which consists of one continuous beam plus top and bottom columns framing into the beam, is used to perform flexural and shear design of the continuous beam under vertical loads.

Member sizes: beam = 36 x 19.5 in; exterior columns =16 x 16 in; interior columns = 18 x 18 in. Story height = 13 ft.

Service Loads: Dead = 3.9 kips/ft (including self-weight); Live = 1.8 kips/ft

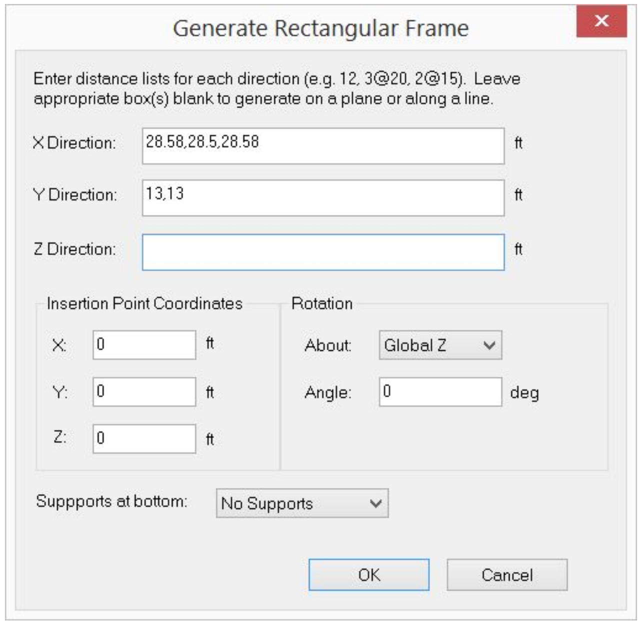


Design the continuous beam according to ACI 318-02/05. Use fc = 4 ksi, fy = 60 ksi

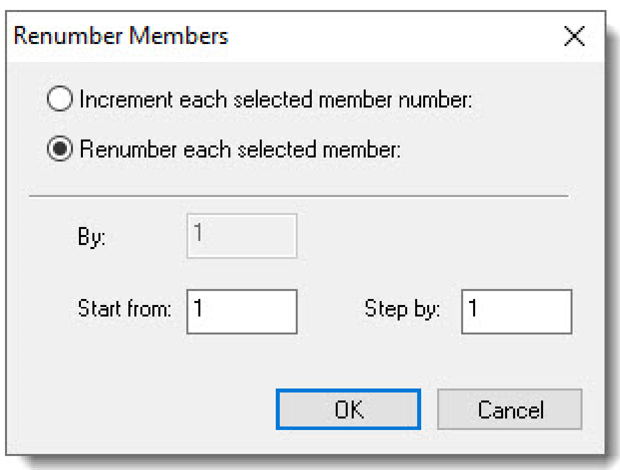
Suggested Modeling Steps

- Set proper units from Settings and Tools > Units & Precisions.

- Generate rectangular frame by Create > Templates > Rectangular Frames as follows:



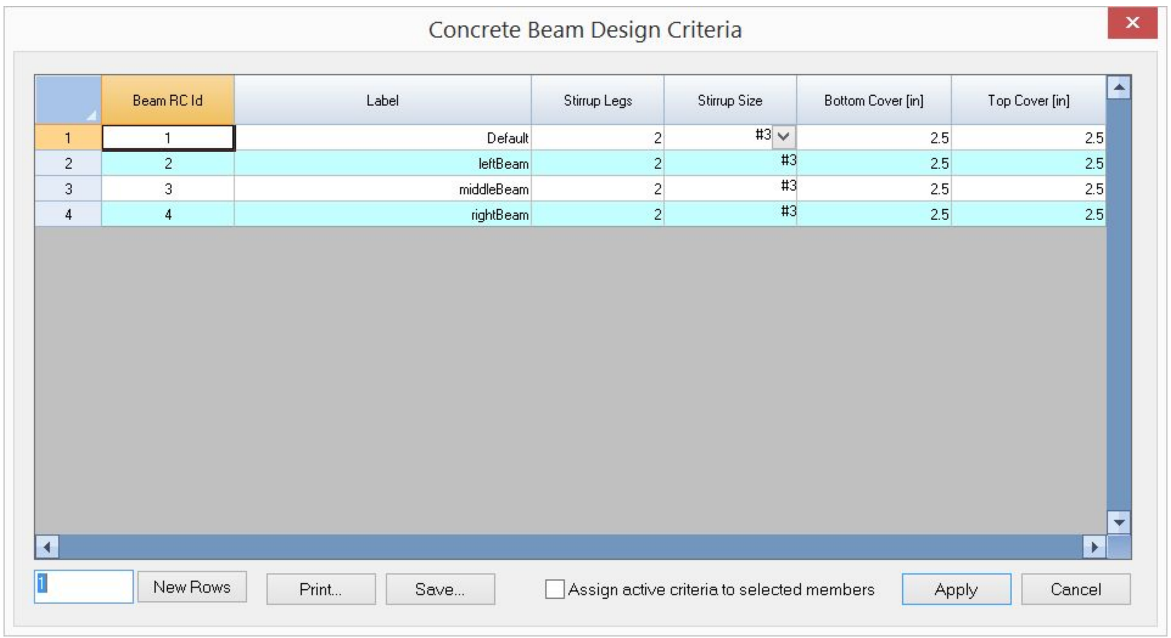
- Select and delete top and bottom beam elements (element 1, 2, 3, 7, 8 and 9 that were generated).
- Select far end nodes of columns and assign fixed supports to them.
- Select all members and renumber each selected member by running Modify > Renumber > Renumber Selected Members, as shown below



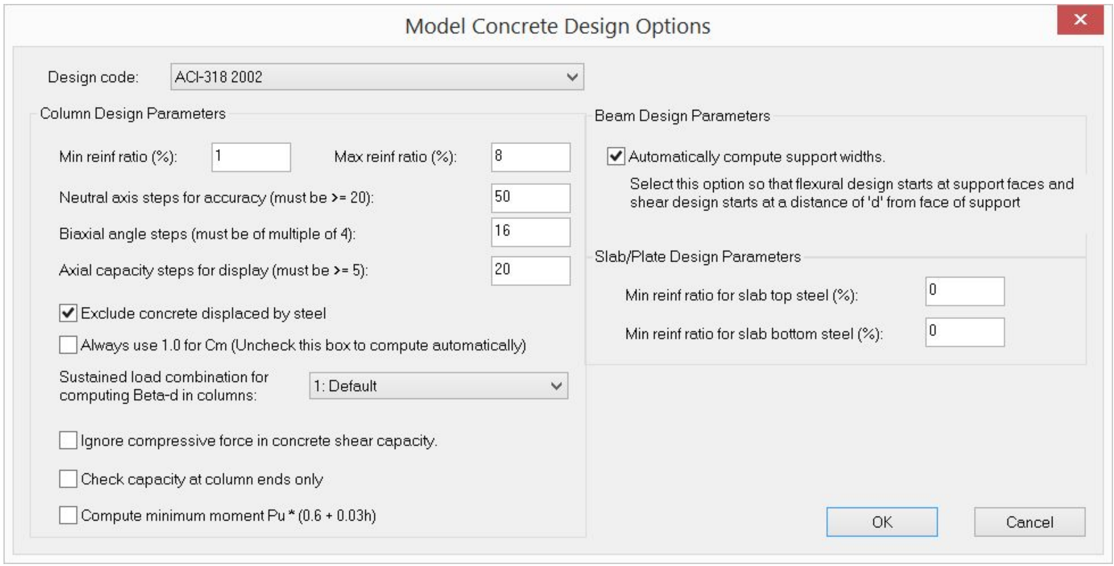
- Define three rectangular sections 36 x 19.5 in, 18 x 18 in and 16 x 16 in using Regular Section in Modify > Member Properties > Sections. Assign each of these sections to appropriate elements
- Define 4.0 ksi material using Std Material in Modify > Member Properties > Materials. Assign this material to all.
- Define five load cases: Dead, Live1, Live2, Live3 and Live4 by Create > Load Cases. Note Live1, 2, 3 and 4 cases are used for live load patterning. Live1 loading is applied

to element 1 and 2. Live2 loading is applied to elements 1 and 3. Live3 is applied to element 2 only. Live4 is applied to elements 2 and 3.

- Define four new load combinations: a). 1.2Dead + 1.6Live1, b). 1.2Dead + 1.6Live2 and c). 1.2Dead + 1.6Live3. d). 1.2Dead + 1.6Live4. Make sure “Perform Concrete Design using this Load Combination” is checked. Also enter sustained load factor (1.2 in this case).
- Define and apply line loads for Dead, Live1, 2, 3 and 4 cases Create > Draw Loads > Line Loads. Use View > Load Diagram to check that the loads are applied correctly.
- Set the analysis options by Analysis > Analysis Options. Choose the model type “2D Frame”. Uncheck “Consider shear deformation on members”. Run Static Analysis to make sure the model is correct before we proceed to the concrete design.
- Select ASTM_615 (English) rebar database by Concrete Design > Concrete Design Tools > Rebar Database.
- Define and assign beam design criteria by Concrete Design > RC Design Criteria > RC Beam Design Criteria.



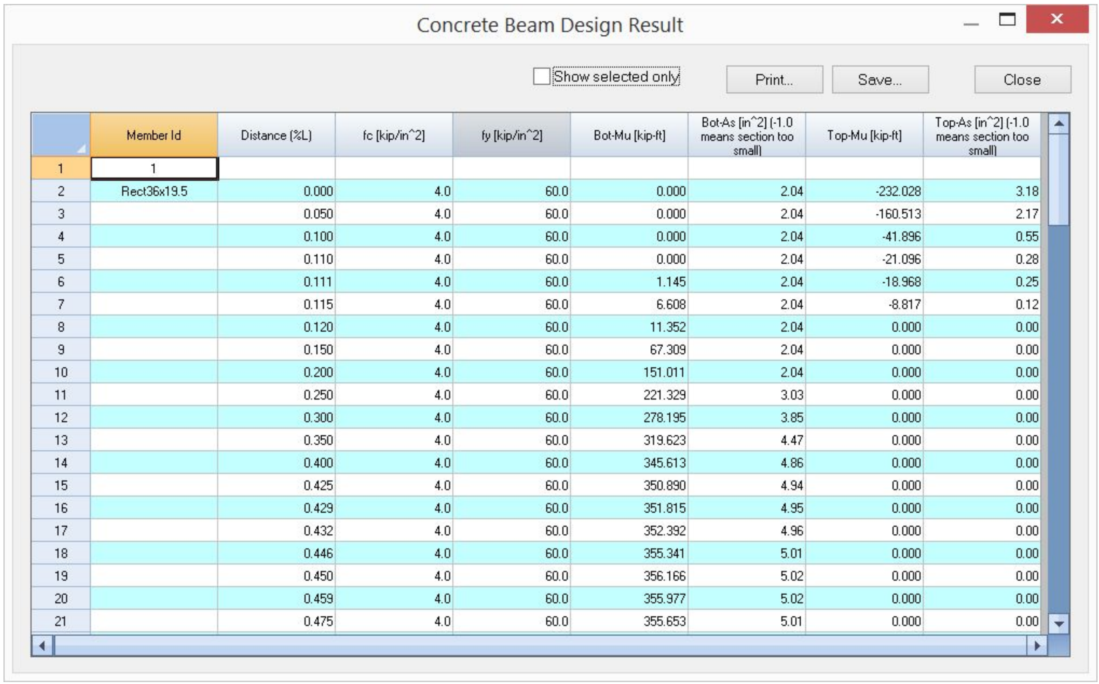
- Set model concrete design criteria by Concrete Design > RC Model Design Criteria. Make sure to select the checkbox “Automatically compute support widths”.



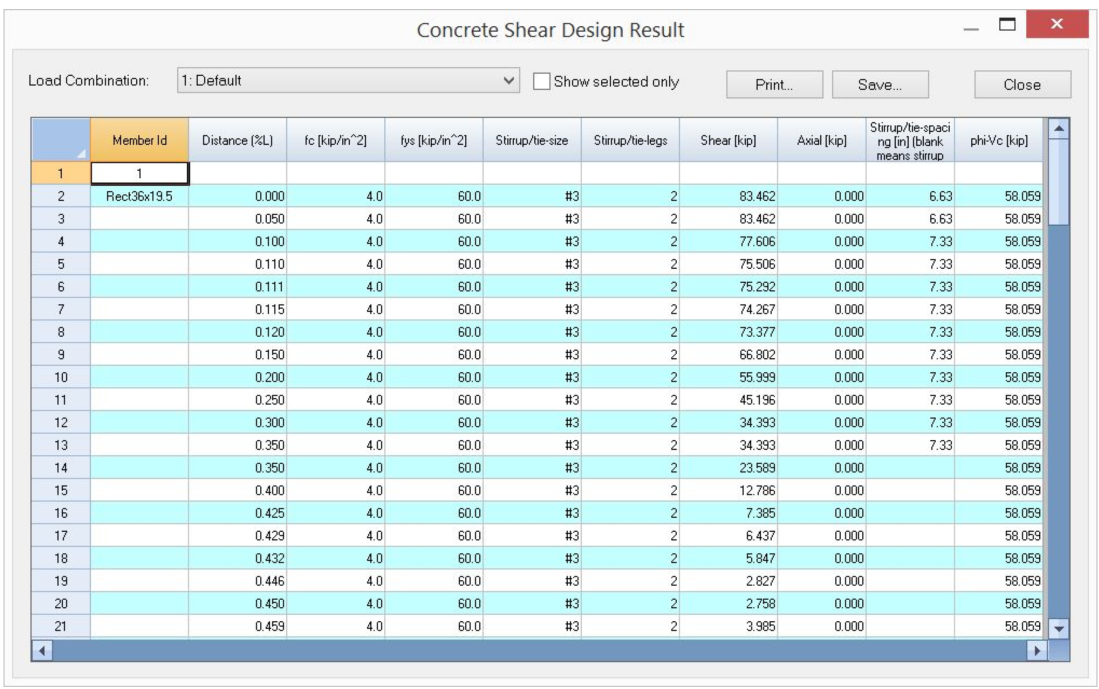
- Select all columns and exclude them from concrete design by Concrete Design > Exclude Elements.

- Perform concrete design by Concrete Design > Perform Concrete Design.

- To view the beam design results in tabulated form, run Concrete Design > Concrete Design Output > RC Beam Results for flexural design and Concrete Design > Concrete Design Output > Member Shear Results for shear design.

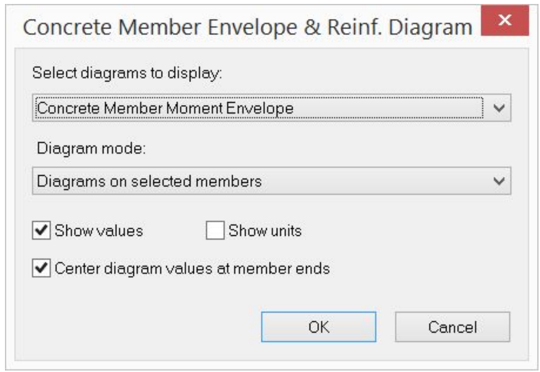


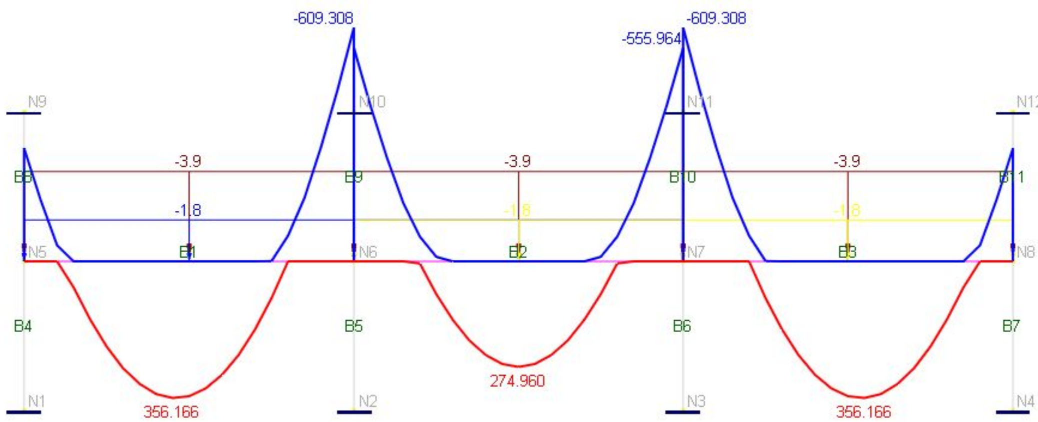
	Member Id	Distance (%L)	fc [kip/in^2]	fy [kip/in^2]	Bot-Mu [kip-ft]	Bot-As [in^2] (-1.0 means section too small)	Top-Mu [kip-ft]	Top-As [in^2] (-1.0 means section too small)
1	1							
2	Rect36x19.5	0.000	4.0	60.0	0.000	2.04	-232.028	3.18
3		0.050	4.0	60.0	0.000	2.04	-160.513	2.17
4		0.100	4.0	60.0	0.000	2.04	-41.896	0.55
5		0.110	4.0	60.0	0.000	2.04	-21.096	0.28
6		0.111	4.0	60.0	1.145	2.04	-18.968	0.25
7		0.115	4.0	60.0	6.608	2.04	-8.817	0.12
8		0.120	4.0	60.0	11.352	2.04	0.000	0.00
9		0.150	4.0	60.0	67.309	2.04	0.000	0.00
10		0.200	4.0	60.0	151.011	2.04	0.000	0.00
11		0.250	4.0	60.0	221.329	3.03	0.000	0.00
12		0.300	4.0	60.0	278.195	3.85	0.000	0.00
13		0.350	4.0	60.0	319.623	4.47	0.000	0.00
14		0.400	4.0	60.0	345.613	4.86	0.000	0.00
15		0.425	4.0	60.0	350.890	4.94	0.000	0.00
16		0.429	4.0	60.0	351.815	4.95	0.000	0.00
17		0.432	4.0	60.0	352.392	4.96	0.000	0.00
18		0.446	4.0	60.0	355.341	5.01	0.000	0.00
19		0.450	4.0	60.0	356.166	5.02	0.000	0.00
20		0.459	4.0	60.0	355.977	5.02	0.000	0.00
21		0.475	4.0	60.0	355.653	5.01	0.000	0.00



	Member Id	Distance (%L)	fc [kip/in^2]	fys [kip/in^2]	Stirrup/tie-size	Stirrup/tie-legs	Shear [kip]	Axial [kip]	Stirrup/tie-spacing [in] (blank means stirrup	phi-Vc [kip]
1	1									
2	Rect36x19.5	0.000	4.0	60.0	#3	2	83.462	0.000	6.63	58.059
3		0.050	4.0	60.0	#3	2	83.462	0.000	6.63	58.059
4		0.100	4.0	60.0	#3	2	77.606	0.000	7.33	58.059
5		0.110	4.0	60.0	#3	2	75.506	0.000	7.33	58.059
6		0.111	4.0	60.0	#3	2	75.292	0.000	7.33	58.059
7		0.115	4.0	60.0	#3	2	74.267	0.000	7.33	58.059
8		0.120	4.0	60.0	#3	2	73.377	0.000	7.33	58.059
9		0.150	4.0	60.0	#3	2	66.802	0.000	7.33	58.059
10		0.200	4.0	60.0	#3	2	55.999	0.000	7.33	58.059
11		0.250	4.0	60.0	#3	2	45.196	0.000	7.33	58.059
12		0.300	4.0	60.0	#3	2	34.393	0.000	7.33	58.059
13		0.350	4.0	60.0	#3	2	34.393	0.000	7.33	58.059
14		0.350	4.0	60.0	#3	2	23.589	0.000		58.059
15		0.400	4.0	60.0	#3	2	12.786	0.000		58.059
16		0.425	4.0	60.0	#3	2	7.385	0.000		58.059
17		0.429	4.0	60.0	#3	2	6.437	0.000		58.059
18		0.432	4.0	60.0	#3	2	5.847	0.000		58.059
19		0.446	4.0	60.0	#3	2	2.827	0.000		58.059
20		0.450	4.0	60.0	#3	2	2.758	0.000		58.059
21		0.459	4.0	60.0	#3	2	3.985	0.000		58.059

- To view the beam design result in graphics, run Concrete Design > Concrete Design Diagrams > RC Member Envelope Diagram. The following shows the member moment envelope diagram.





Results

The following table compares the design moments between the program and the reference [Ref 17, pp 7-43]:

	Moment (ft-kips)	ENERCALC 3D	[Ref 17, pp 7-43]
End Span	Ext (-) moment	-232.0	-385.9
	(+) moment	356.1	441.1
	Int (-) moment	-523.6	-615.8
Interior Span	(+) moment	274.9	383.8

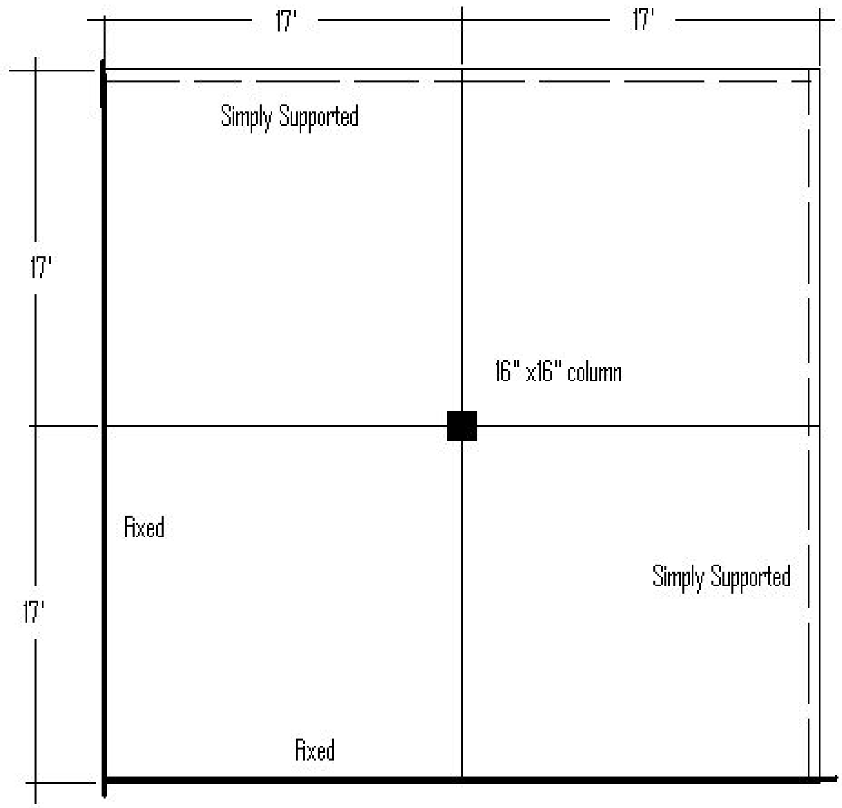
Comments

The reference [Ref 17, pp 7-43] uses the approximate coefficients method while the program uses the exact stiffness method. It is apparent the former method is quite conservative.

10.16 Example 15: Design of Concrete Slab

Problem Description

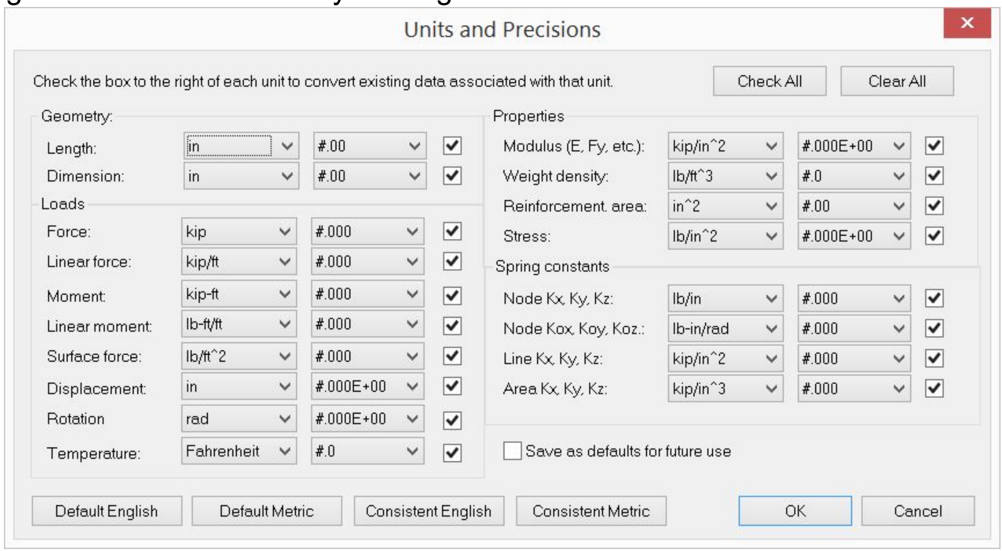
The following 34 x 34 ft flat plate is supported by two fixed edges and two simply supported edges as well as a 16 x 16 in column in the middle. [Ref 20, pp 536-540].



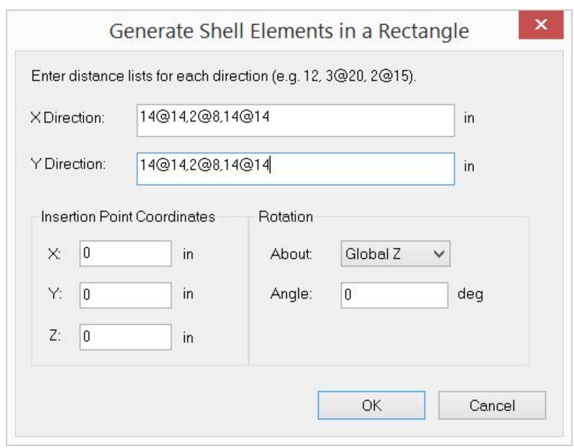
Factored load = 170 psf (including self-weight)
fc = 4 ksi, fy = 60 ksi
Slab thickness h = 6.5 in
Concrete cover: d = 1.25 in over the central column and near the intersection of the two fixed edges, d = 1.0 in for the rest of the area.

Suggested Modeling Steps

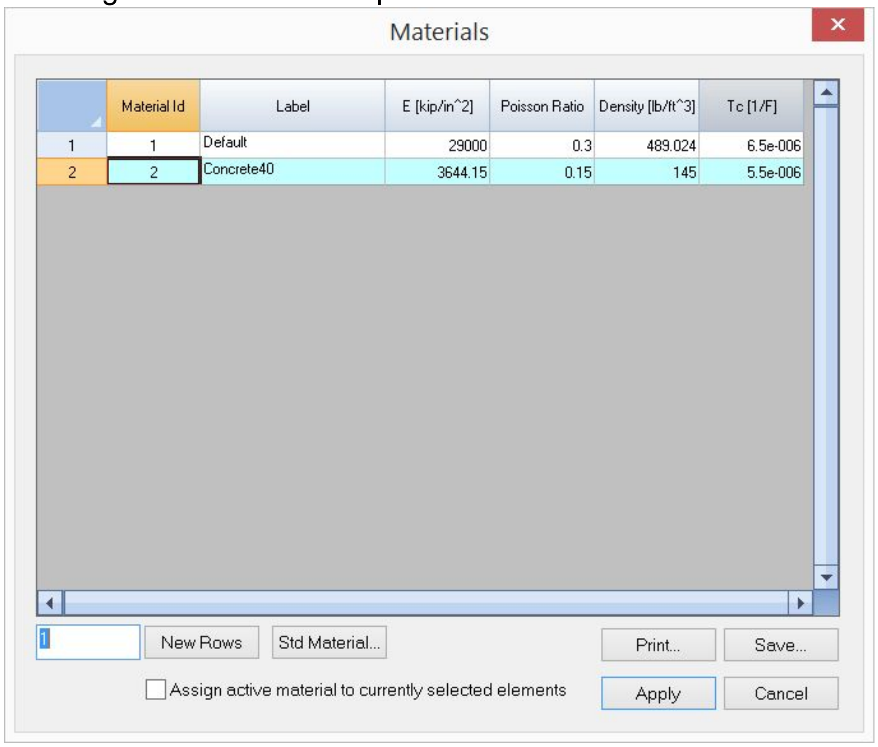
- Set proper units from Settings and Tools > Units & Precisions. In particular, set the length unit to be inch for easy mesh generation.



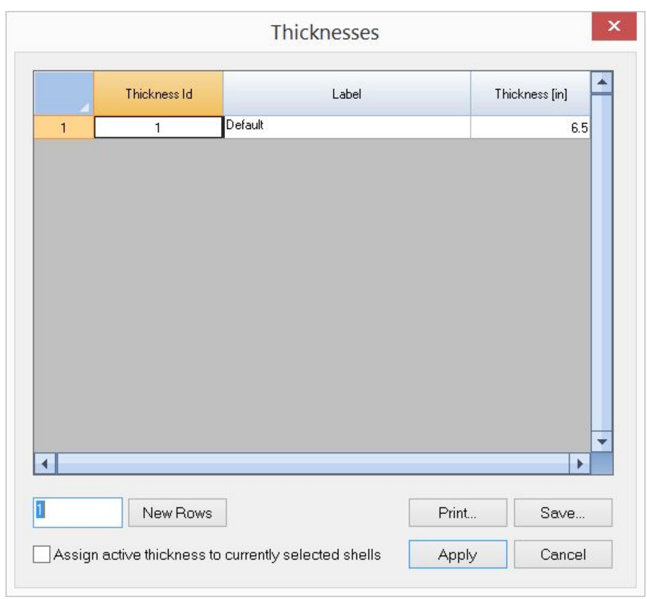
- Generate rectangular shells by Create > Templates > Rectangular Shells as follows:



- Define 4.0 ksi concrete material using Std Material in Modify > Shell Properties > Materials. Assign this material to all plates.

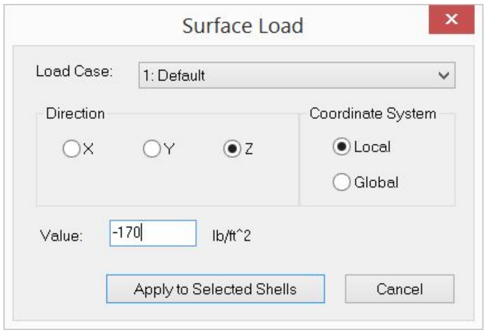


- Define a thicknesses of 6 inches using Modify > Shell Properties > Thicknesses. Assign this thickness to all plates.

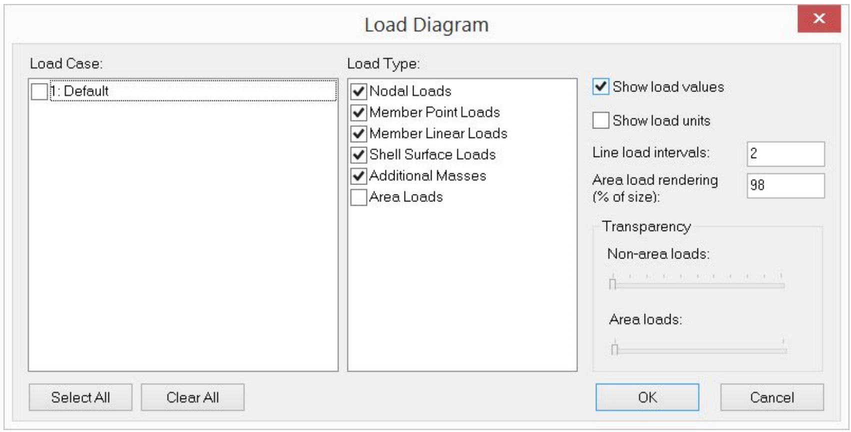


- Using Create > Boundary Conditions > Support, assign fixed supports to nodes along the left and bottom edges. Assign pinned supports to nodes along the right and top edges as well as to the column node.

- Assign normal surface load of 170 lb/ft^2 to all plates by Create > Draw Loads > Surface Loads.

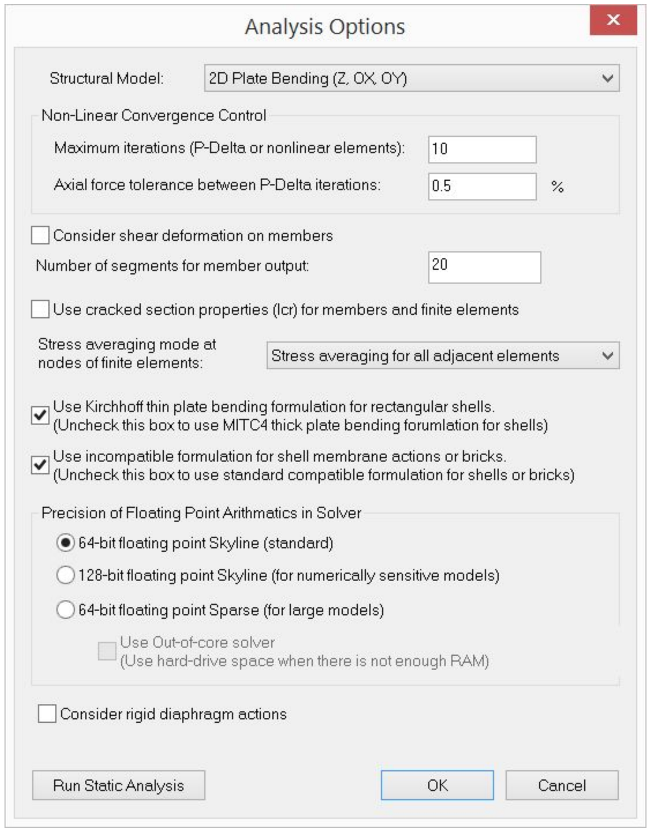


- You may turn off the display of surface loads by View > Load Diagram.

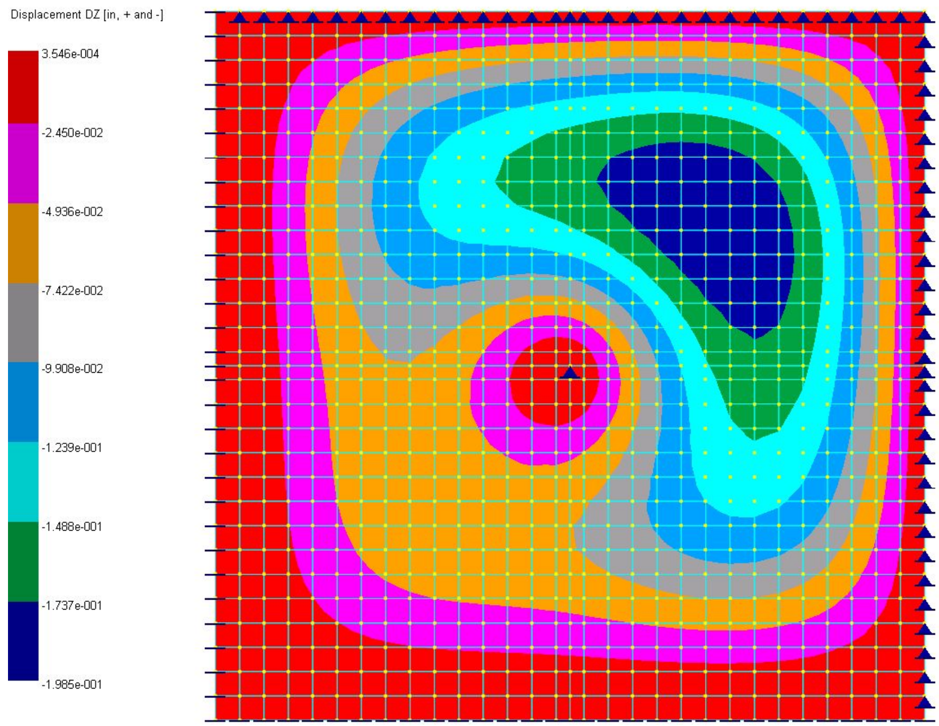


- Use the default load combination for concrete design from Create > Load Combinations.

- Set the analysis options by Analysis > Analysis Options. Choose the model type “2D Plate Bending”. Uncheck “Consider shear deformation on members”. Check “Use Kirchhoff thin plate bending formulation for rectangular shells”. *The Kirchhoff element formulation is recommended over the MITC4 bending formulation for thin plate models that contain only rectangular elements.* Run Static Analysis to make sure the model is correct before we proceed to the concrete design.



- Various analysis results may be viewed by Analysis Results > Contour Diagram. The following are Dz displacement, plate Mxx and Mxy contours.



Dz Displacement Contour

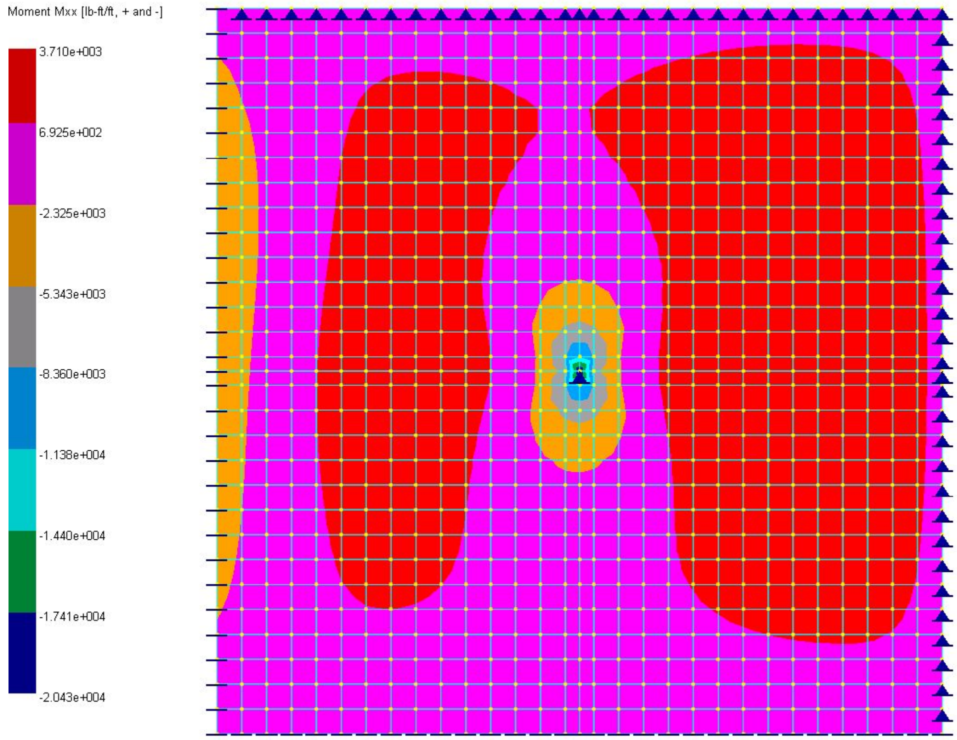


Plate Mxx Contour

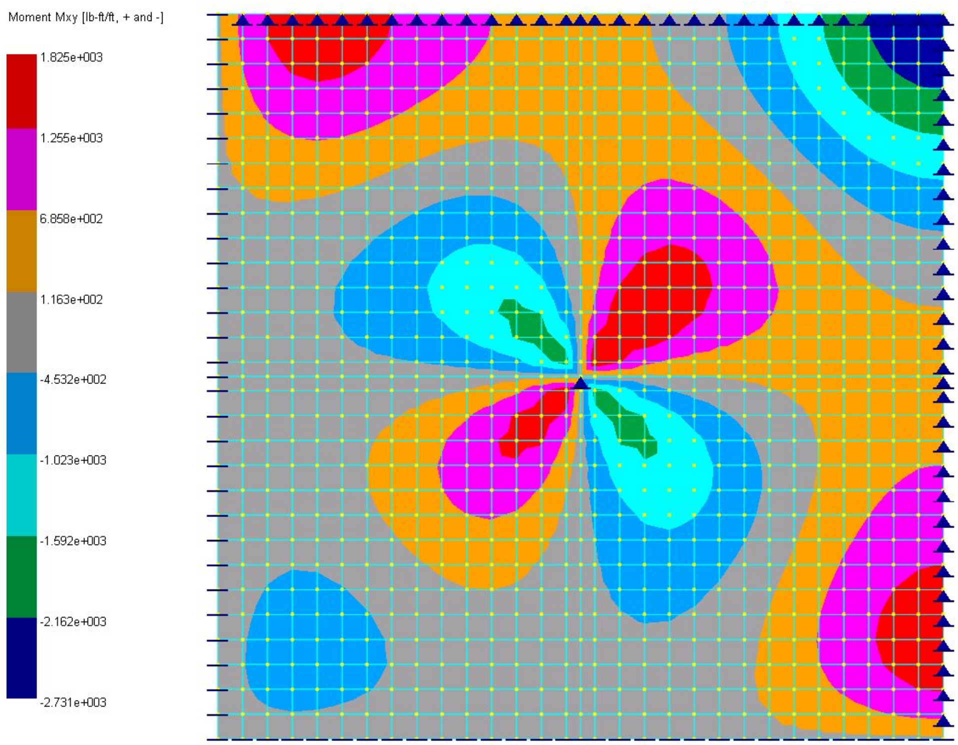
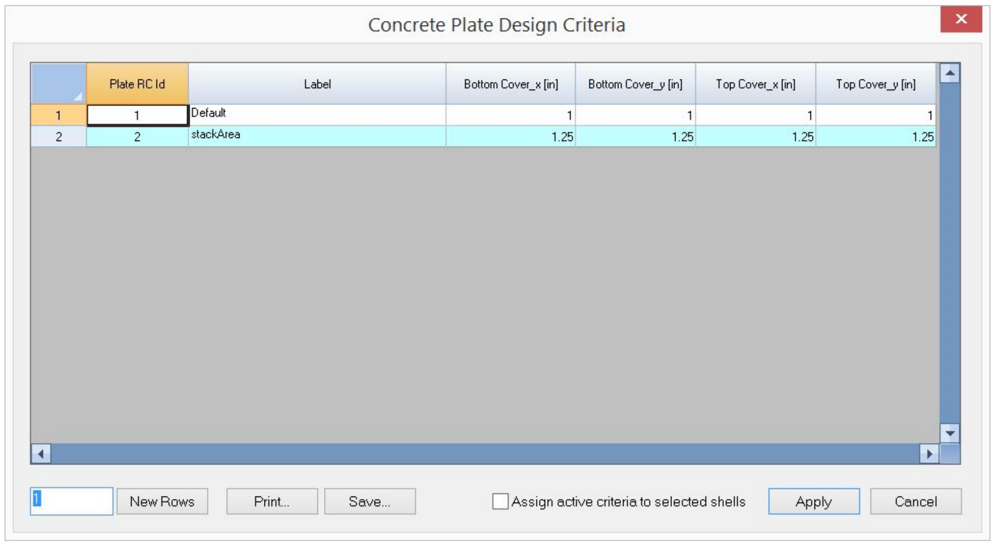


Plate Mxy Contour

- Select ASTM_615 (English) rebar database by Concrete Design > Concrete Design Tools > Rebar Database.

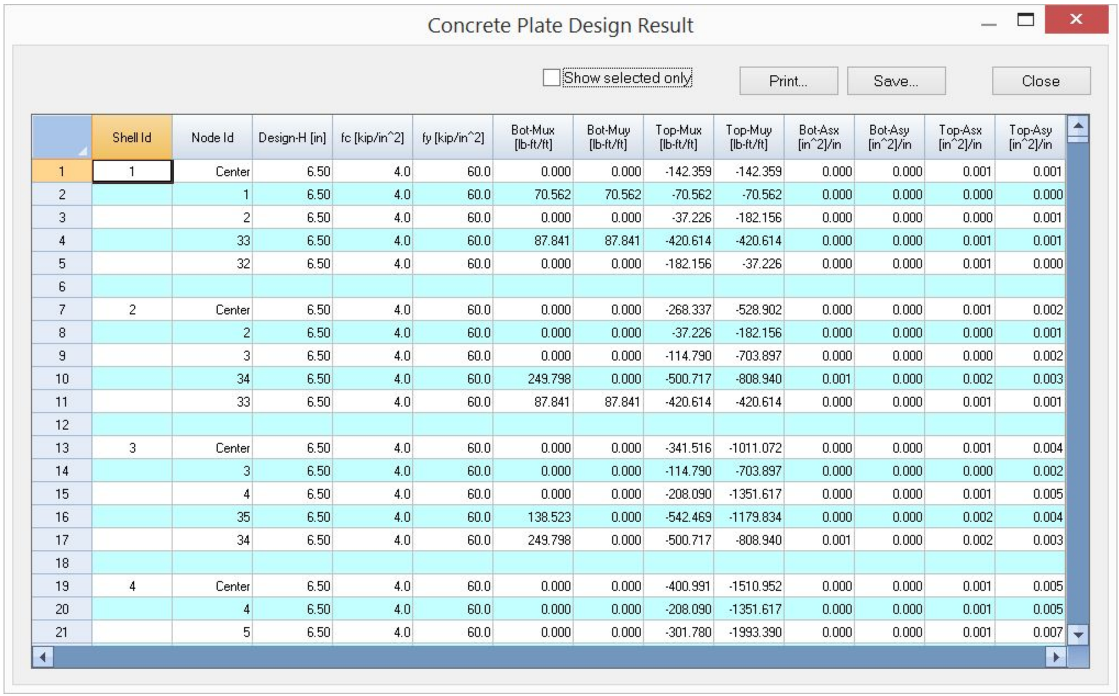
- Define two plate design criteria by Concrete Design > RC Design Criteria > RC Plate Design Criteria as follows. Assign the stackArea criteria to area where bar stacking occurs – that is, over the central column and near the intersection of the two fixed edges.



- Select the four plates over the column node and exclude these plates from concrete design by Concrete Design > Exclude Elements.

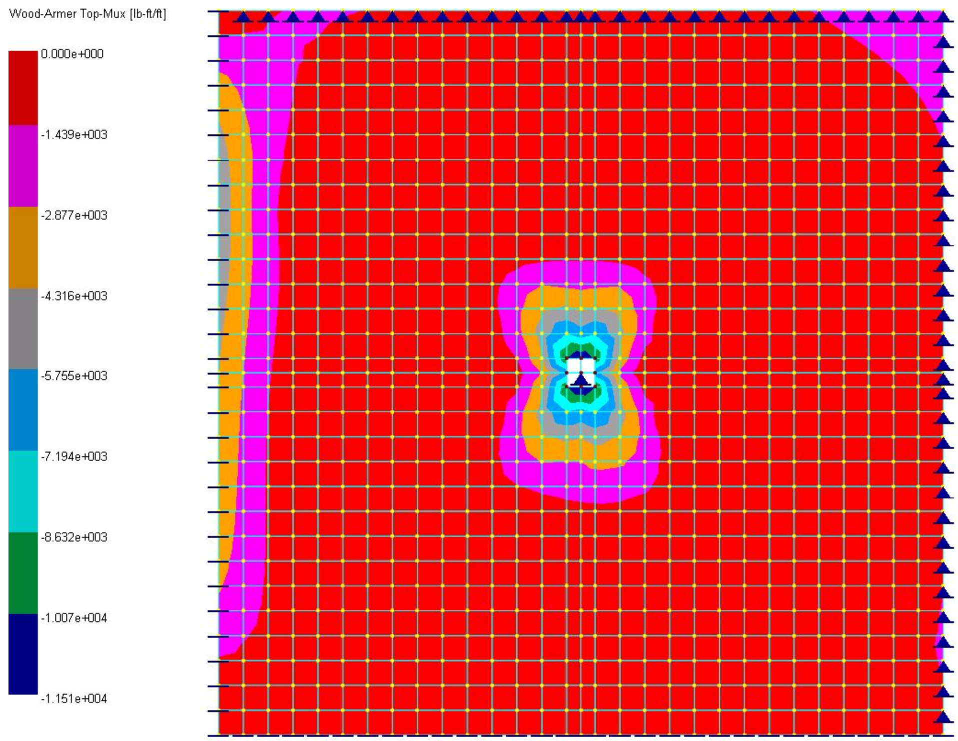
- Perform concrete design by Concrete Design > Perform Concrete Design.

- To view the plate flexural design results in tabulated form, run Concrete Design > Concrete Design Output > RC Plate Results.

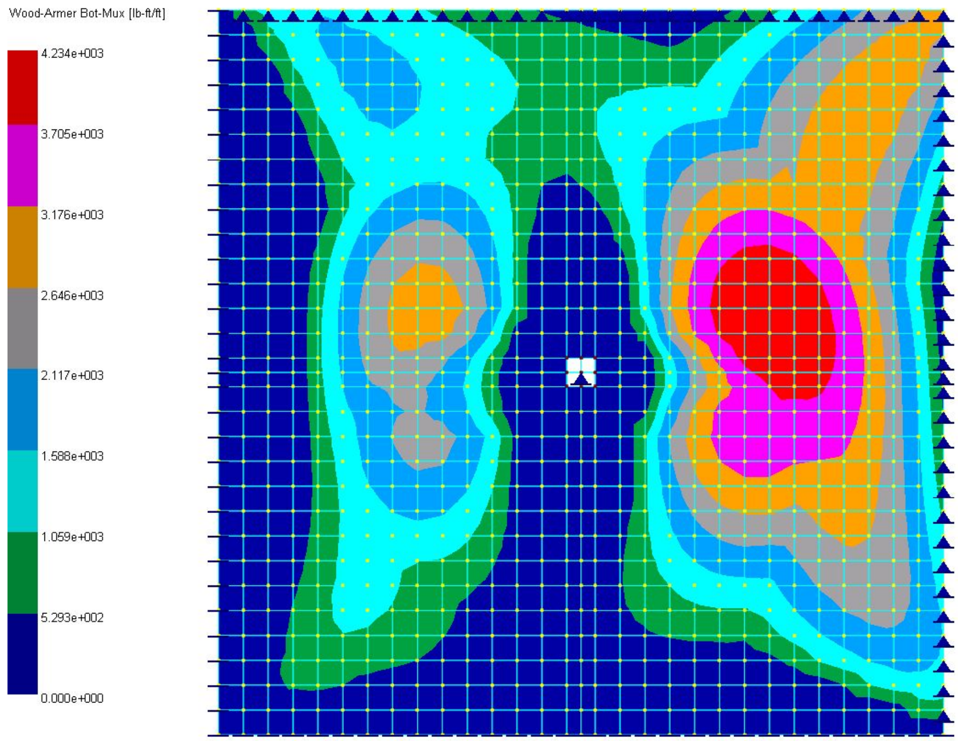


	Shell Id	Node Id	Design-H [in]	fc [kip/in^2]	fy [kip/in^2]	Bot-Mux [lb-ft/ft]	Bot-Muy [lb-ft/ft]	Top-Mux [lb-ft/ft]	Top-Muy [lb-ft/ft]	Bot-Asx [in^2]/in	Bot-Asy [in^2]/in	Top-Asx [in^2]/in	Top-Asy [in^2]/in
1	1	Center	6.50	4.0	60.0	0.000	0.000	-142.359	-142.359	0.000	0.000	0.001	0.001
2		1	6.50	4.0	60.0	70.562	70.562	-70.562	-70.562	0.000	0.000	0.000	0.000
3		2	6.50	4.0	60.0	0.000	0.000	-37.226	-182.156	0.000	0.000	0.000	0.001
4		33	6.50	4.0	60.0	87.841	87.841	-420.614	-420.614	0.000	0.000	0.001	0.001
5		32	6.50	4.0	60.0	0.000	0.000	-182.156	-37.226	0.000	0.000	0.001	0.000
6													
7	2	Center	6.50	4.0	60.0	0.000	0.000	-268.337	-528.902	0.000	0.000	0.001	0.002
8		2	6.50	4.0	60.0	0.000	0.000	-37.226	-182.156	0.000	0.000	0.000	0.001
9		3	6.50	4.0	60.0	0.000	0.000	-114.790	-703.897	0.000	0.000	0.000	0.002
10		34	6.50	4.0	60.0	249.798	0.000	-500.717	-808.940	0.001	0.000	0.002	0.003
11		33	6.50	4.0	60.0	87.841	87.841	-420.614	-420.614	0.000	0.000	0.001	0.001
12													
13	3	Center	6.50	4.0	60.0	0.000	0.000	-341.516	-1011.072	0.000	0.000	0.001	0.004
14		3	6.50	4.0	60.0	0.000	0.000	-114.790	-703.897	0.000	0.000	0.000	0.002
15		4	6.50	4.0	60.0	0.000	0.000	-208.090	-1351.617	0.000	0.000	0.001	0.005
16		35	6.50	4.0	60.0	138.523	0.000	-542.469	-1179.834	0.000	0.000	0.002	0.004
17		34	6.50	4.0	60.0	249.798	0.000	-500.717	-808.940	0.001	0.000	0.002	0.003
18													
19	4	Center	6.50	4.0	60.0	0.000	0.000	-400.991	-1510.952	0.000	0.000	0.001	0.005
20		4	6.50	4.0	60.0	0.000	0.000	-208.090	-1351.617	0.000	0.000	0.001	0.005
21		5	6.50	4.0	60.0	0.000	0.000	-301.780	-1993.390	0.000	0.000	0.001	0.007

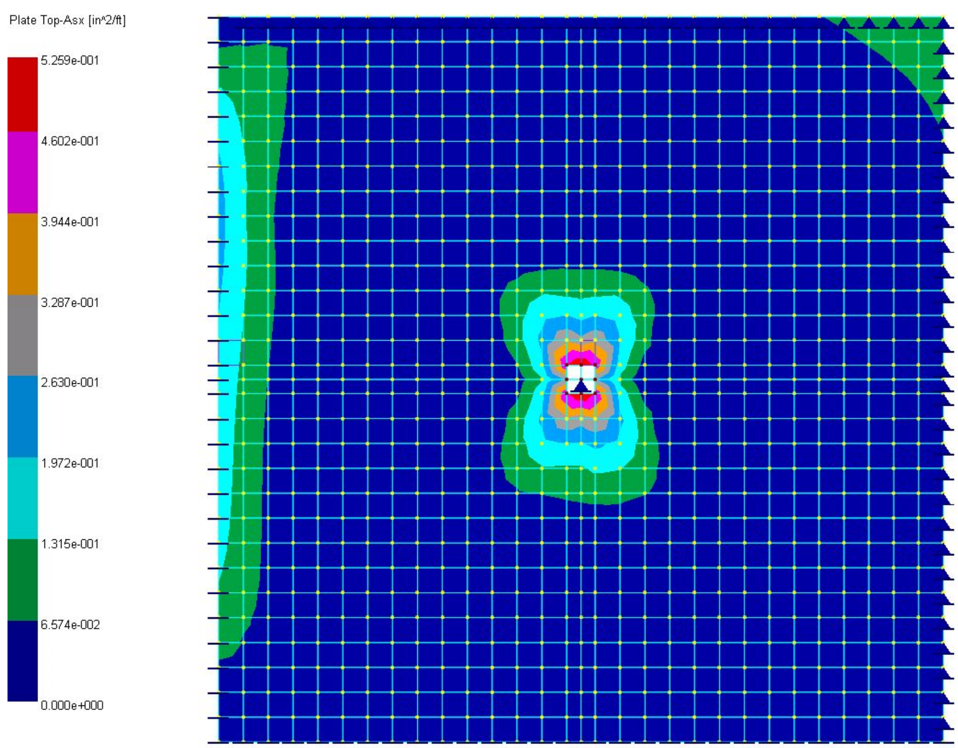
- To view the plate design result in graphics, run Concrete Design > Concrete Design Diagrams > RC Plate Envelope Contour. For illustration purposes, the X-top and X-bottom design (Wood-Armer) moment and the corresponding required steel contours are shown below. Based on reinforcement contours and some common sense, the actual reinforcement can be provided for final design.



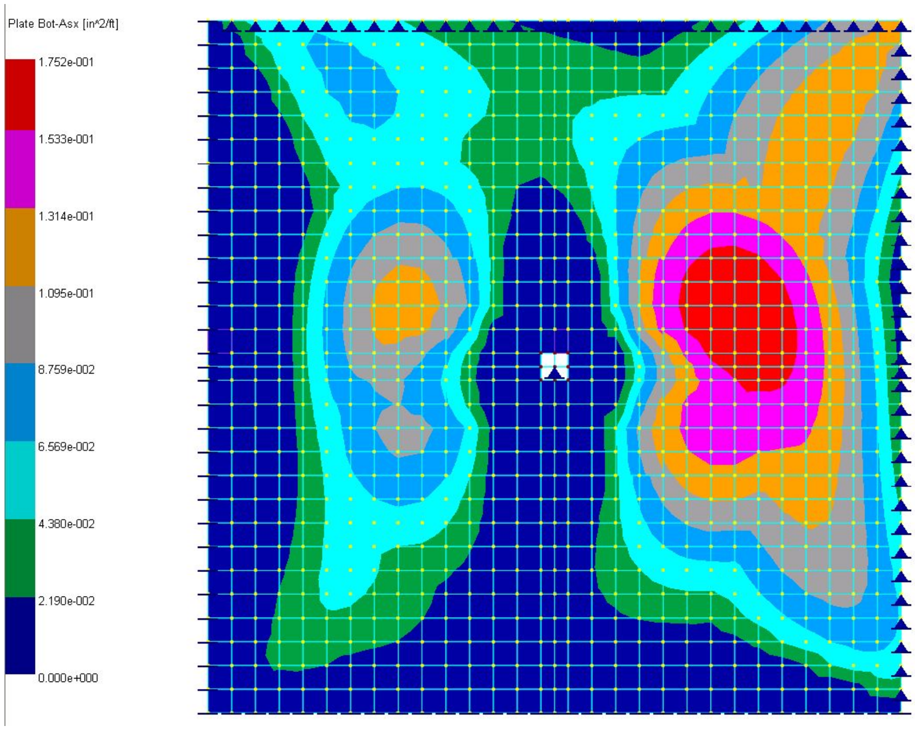
Wood-Armer Top-Mux



Wood-Armer Bottom-Mux



Required Top-Asx



Required Bottom-Asx

Results

	ENERCALC 3D	Ref 20
Negative moment over column (lb-ft/ft)	-11,510	-10,528
Negative steel over column (in^2/ft)	0.5259	0.48
Negative moment along fixed edges (lb-ft/ft)	-4,412	-3,509
Negative steel along the fixed edges (in^2/ft)	0.183	0.15
Positive moment in outer spans (lb-ft/ft)	4,234	3,789
Positive steel in outer spans (in^2/ft)	0.1752	0.16

Comments

The reference used Advanced Strip Method to compute the design moments and therefore is approximate in nature. The program computes the design (Wood-Armer) moments based on the plate element Mxx, Myy and Mxy. Although the two methods are fundamentally different, comparable results are obtained.

One of the difficulties in using finite element results to perform concrete plate (or slab) design is stress singularity. In this example, the slab stress around the column is theoretically infinite. This is reflected in stress and reinforcement spikes at the slab/column interface area.

Finer finite element mesh will generally exacerbate the problem. We alleviated the problem by excluding the four finite elements over the column from design. Appropriate averaging or redistribution of reinforcement should also be applied before the actual reinforcement is provided.

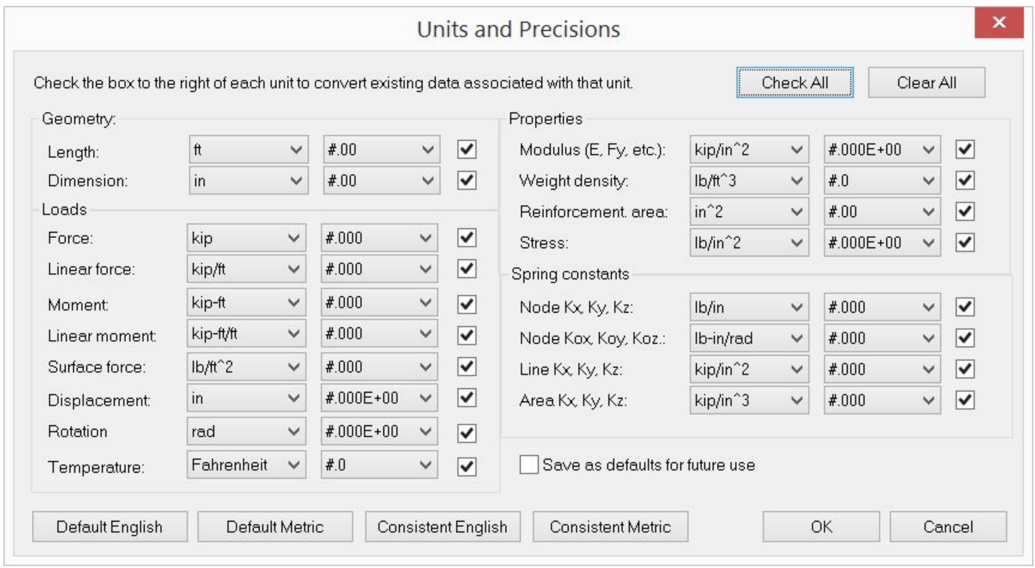
10.17 Example 16: Design of Steel Beam

Problem Description

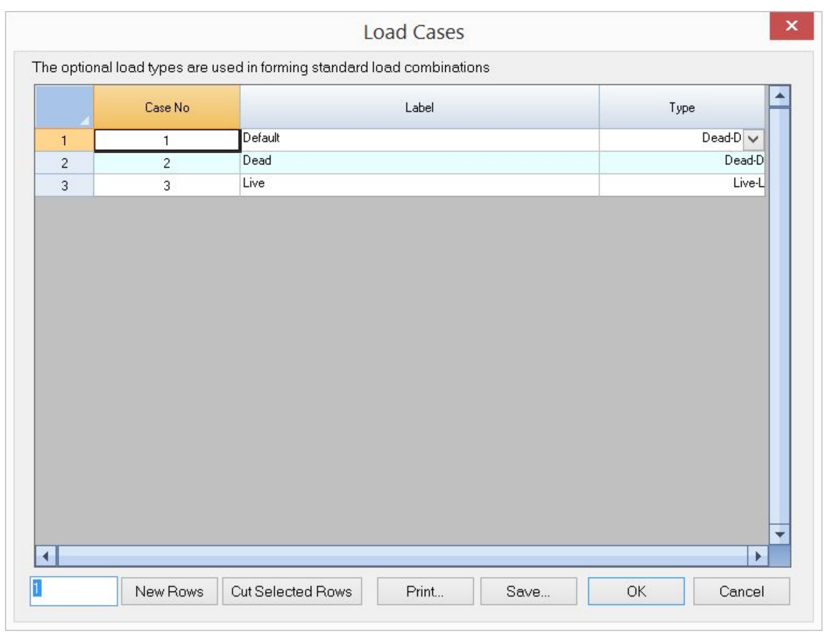
Select the lightest W section for the simply supported beam of L = 50ft, Lb = 25 ft. The superimposed load is 0.4 kip/ft dead load and 1.0 kip/ft live load. Use A992 steel. [Ref 22, pp 435-437].

Suggested Modeling Steps

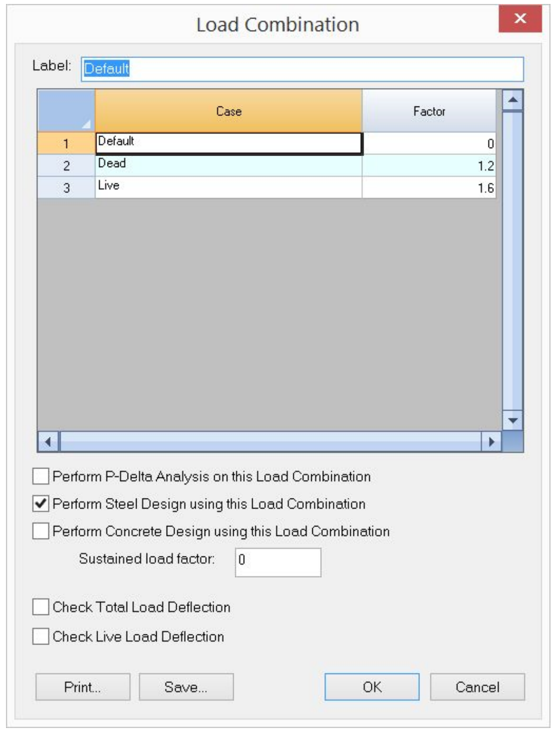
- Set proper units from Settings and Tools > Units & Precisions.



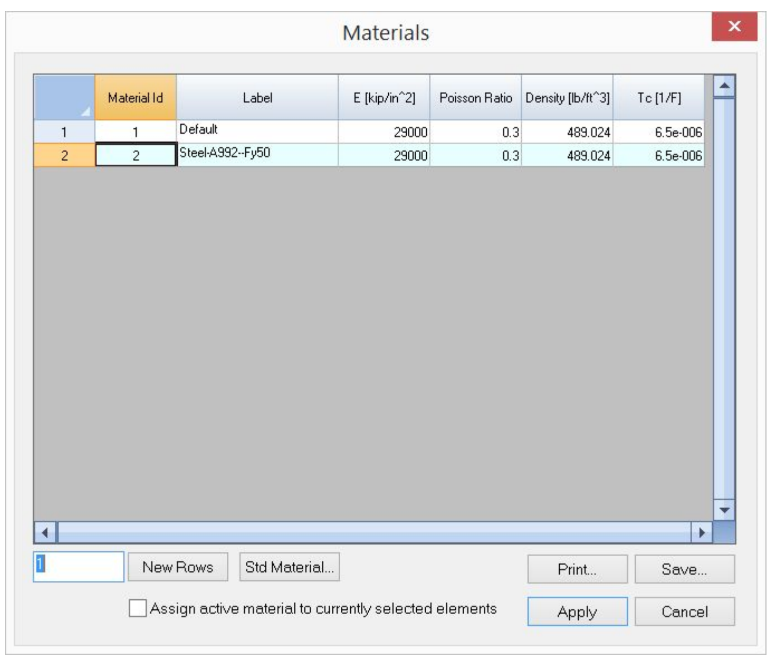
- Define load cases from Create > Load Cases



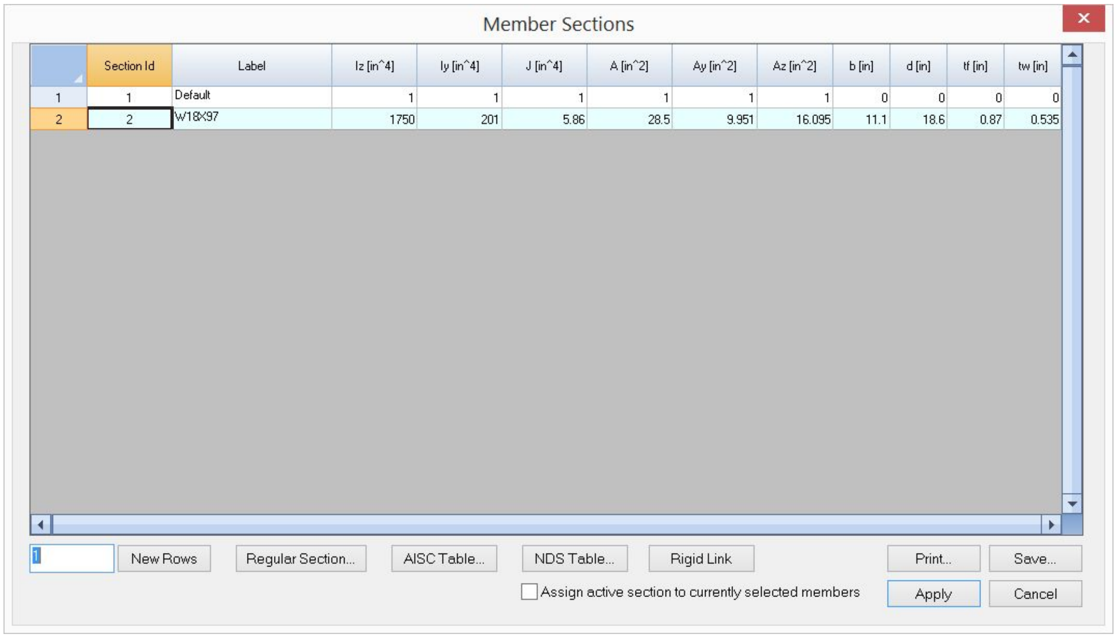
- Define the load combination from Create > Load Combinations: make sure “Perform Steel Design using this Load Combination” is checked.



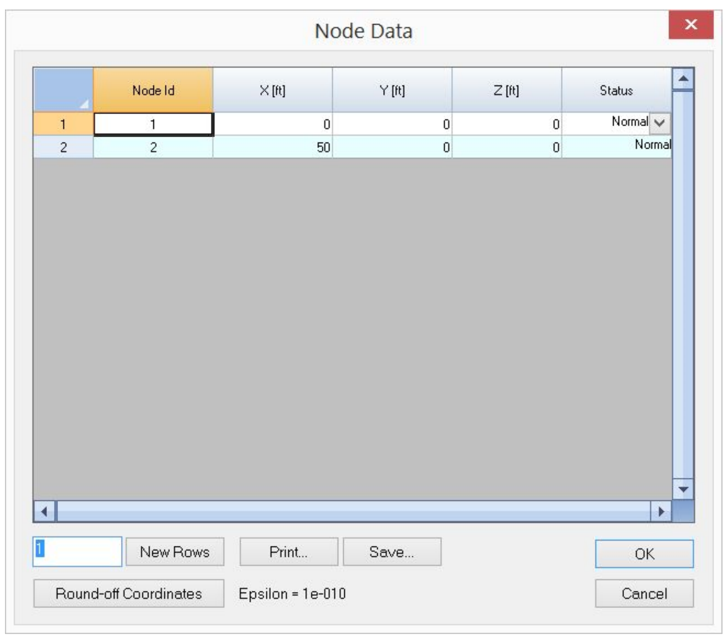
- Define the material from Modify > Member Properties > Materials using the standard steel Steel-A992--Fy50. Steel properties such as Fy and Fu are set automatically.



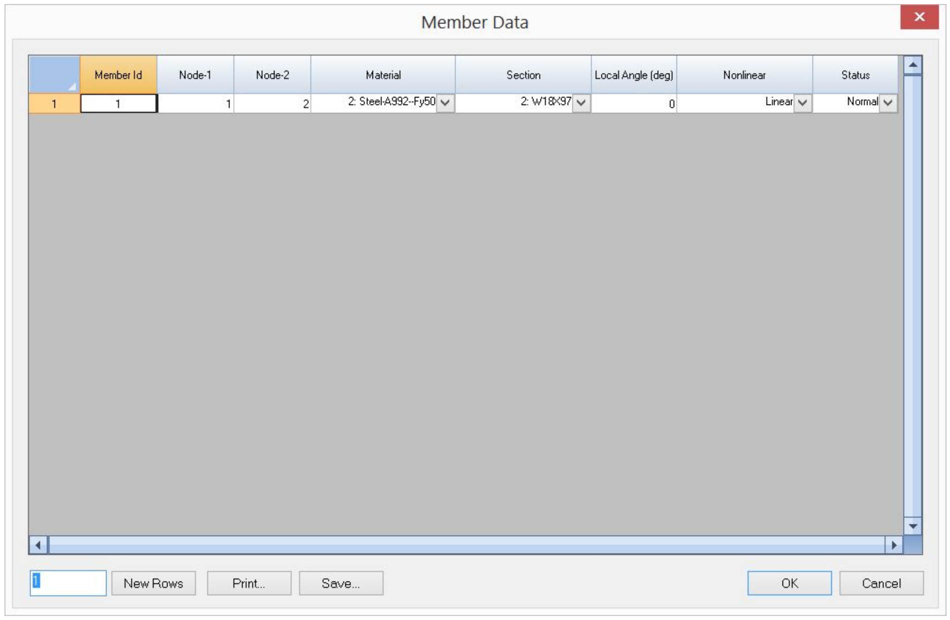
- Define the section W18x97 from Modify > Member Properties > Sections using the AISC table.



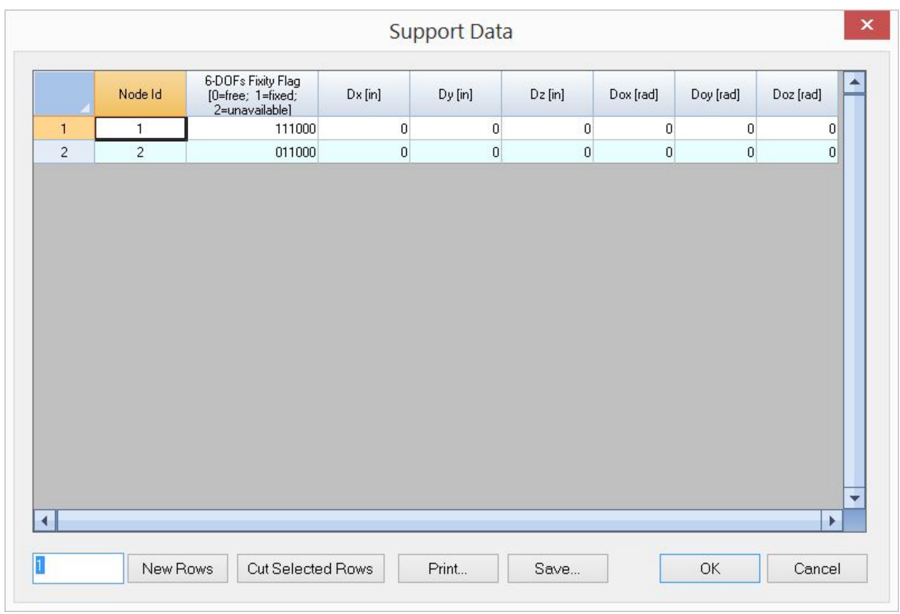
- Define the two nodes from Tables > Nodes.



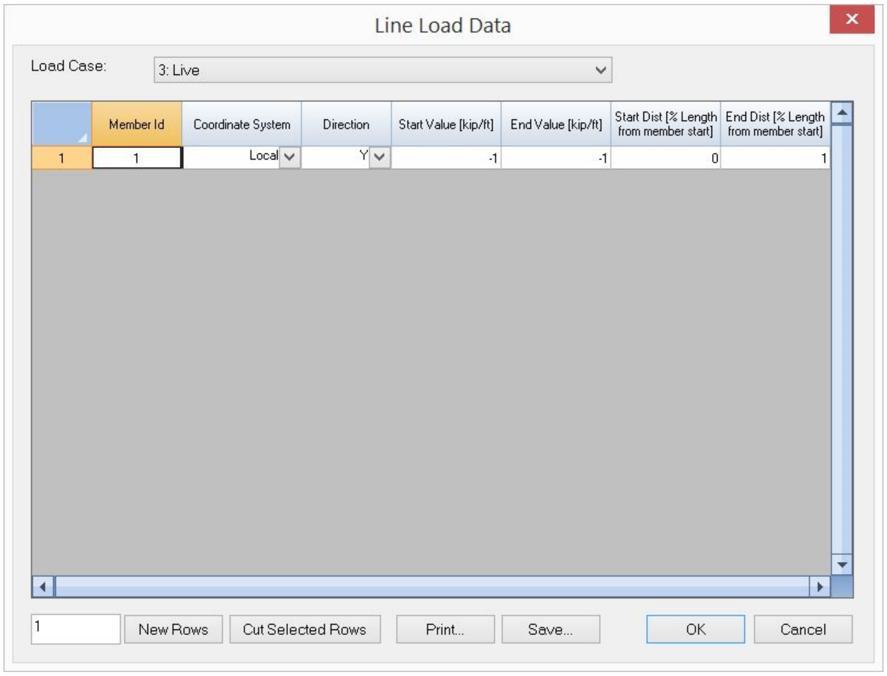
- Define the one beam from Tables > Members



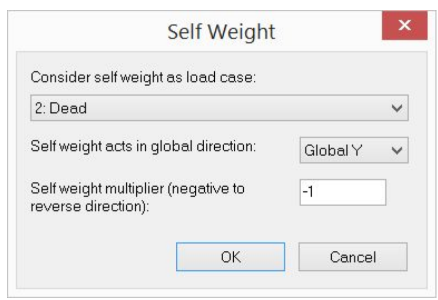
- Define the two supports from Tables > Supports



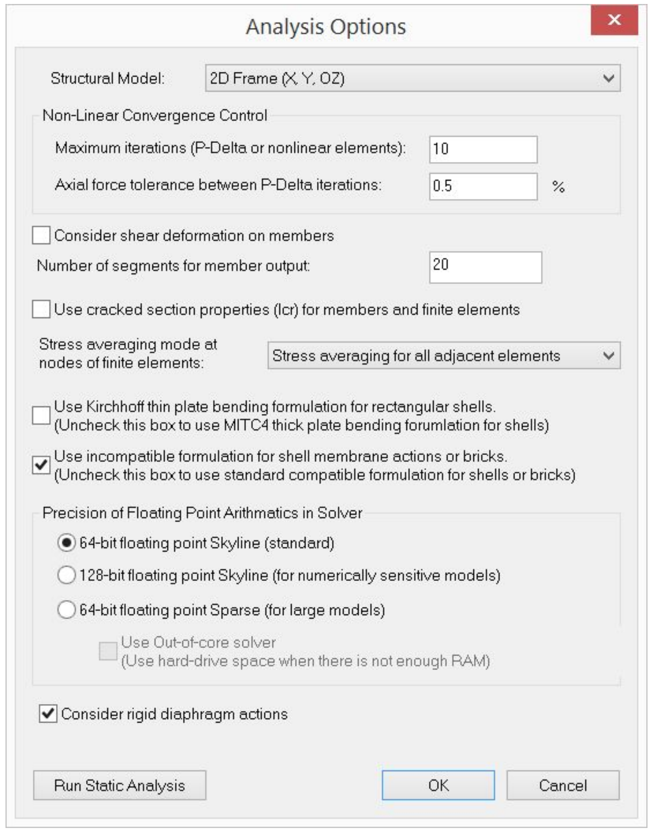
- Define both the dead and live line loads from Tables > Line Loads



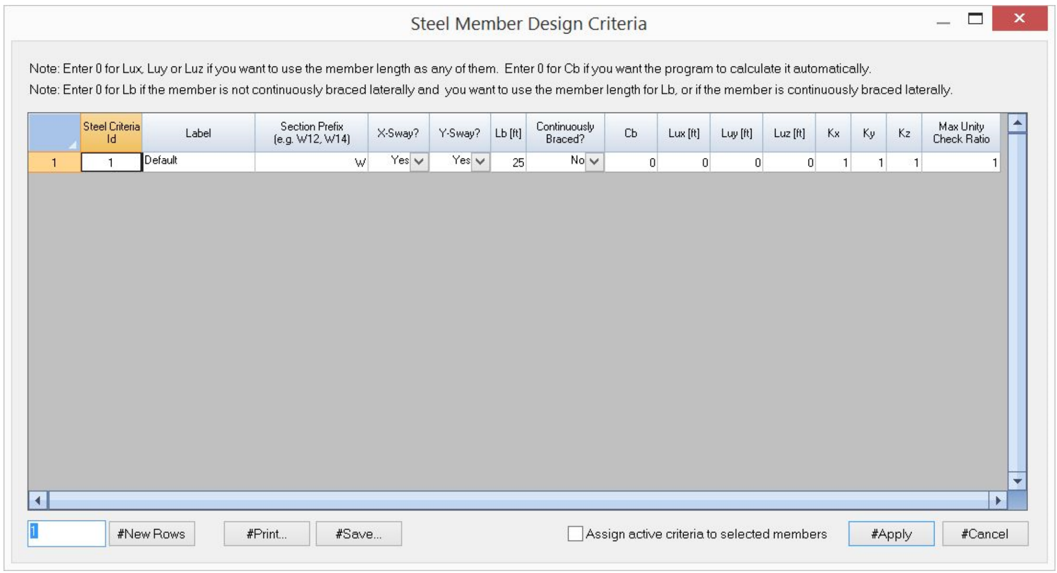
- Define the self weight from Tables > Self Weights. Make sure the self weight acts in the negative global Y direction.



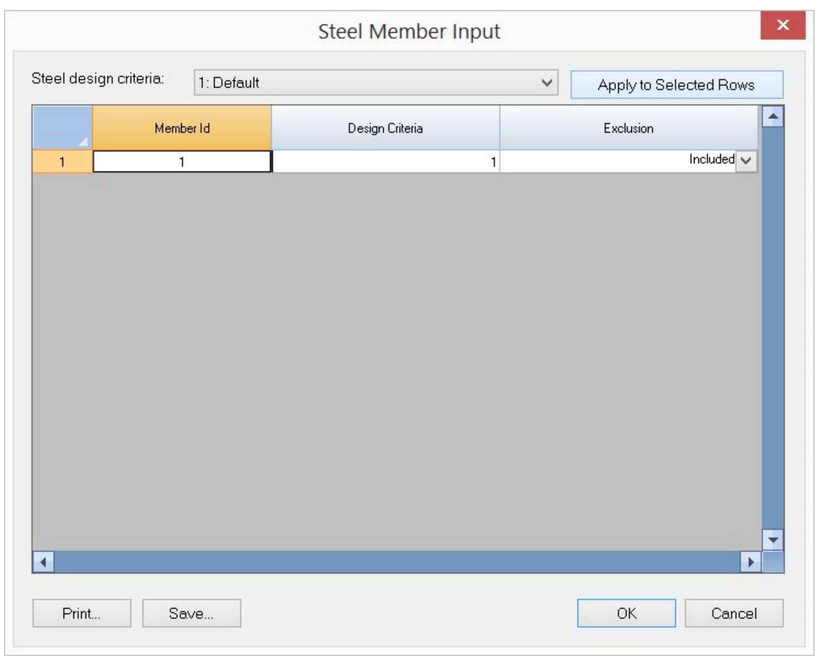
- Set structural model as 2D Frame from Analysis > Analysis Options. Run Static Analysis to make static analysis results available for steel design.



- Define the steel member design criteria from Steel Design > Steel Design Criteria > Steel Member Design Criteria. Use “W” as the section prefix as we want to find the light W section. We could also use “W12, W18” for the section prefix if we would want to use either W12 or W18x sections. Make sure Cb = 0 so we will have the program calculate it automatically. **Important: If 0 is entered for Lb for non-continuously braced, then Lb is taken as the member length. If the member is fully braced laterally, you must enter 0 for Lb.**



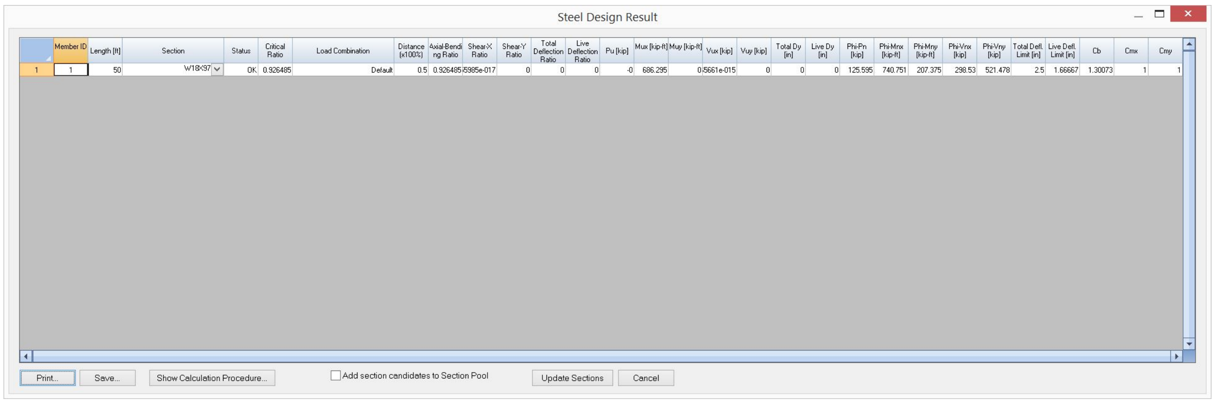
- Define the steel member input from Steel Design > Steel Member Input.



- Perform the steel design from Steel Design > Perform Steel Design.

- View the steel design results from Steel Design > Steel Design Results. By default, up to 10 candidate sections are available. The W18x97 happens to be the lightest section. Also notice that Cb is calculated automatically (Cb = 1.3). If desired, we could now update the member with a different section candidate, reanalyze the model and perform the steel design again.

  You can also view the detailed step-by-step calculation procedures for the most critical load condition on each member.



	Member ID	Length [ft]	Section	Status	Critical Ratio	Load Combination	Distance (x100%)	Axial-Bending Ratio	Shear-X Ratio	Shear-Y Ratio	Total Deflection Ratio	Live Deflection Ratio	Pu [kip]	Mux [kip-ft]	Muy [kip-ft]	Vux [kip]	Vuy [kip]	Total Dy [in]	Live Dy [in]	Phi-Pn [kip]	Phi-Mnx [kip-ft]	Phi-Mny [kip-ft]	Phi-Vnx [kip]	Phi-Vny [kip]	Total Defl. Limit [in]	Live Defl. Limit [in]	Cb	Cmx	Cmy
1	1	50	W18X97	OK	0.926485	Default	0.5	0.926485	5985e-017	0	0	0	-0	686.295	0	5661e-015	0	0	0	125.595	740.751	207.375	298.53	521.478	2.5	1.66667	1.30073	1	1

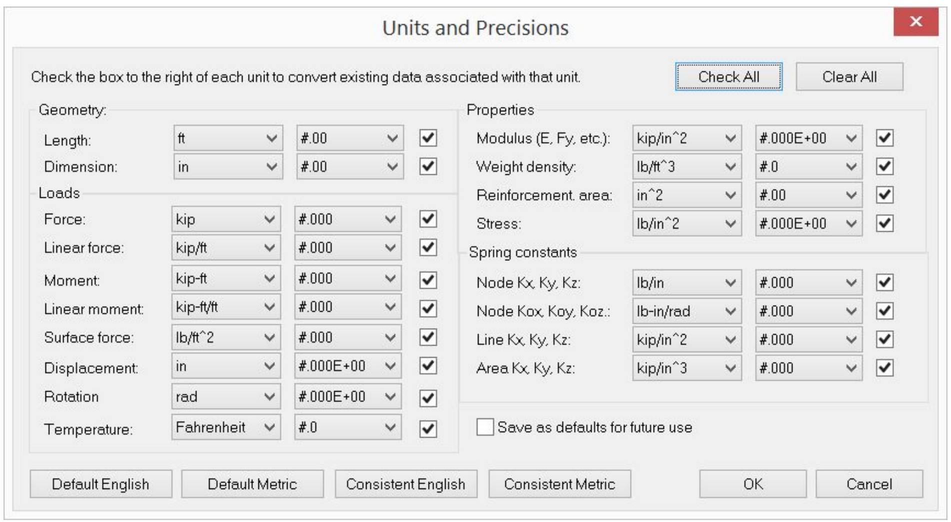
10.18 Example 17: Design of Steel Column

Problem Description

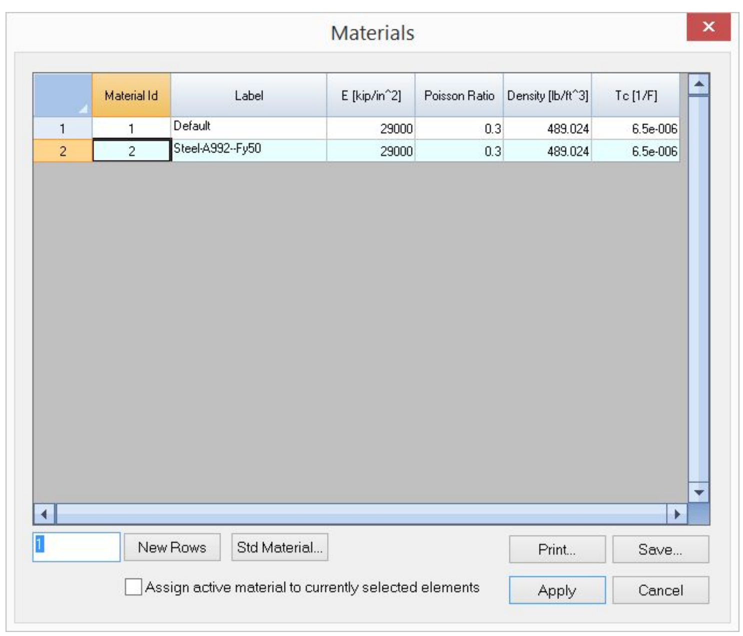
Select an ASTM A992 W-shape with a 10-in nominal depth to carry the following load effects: Pu = 30 kips, Mux = 90 kip-ft, Muy = 12 kip-ft.
The unbraced length is 14 ft and the ends are pinned. Cb = 1.14. The member is non-sway. [Ref 23, Example H.4].

Suggested Modeling Steps

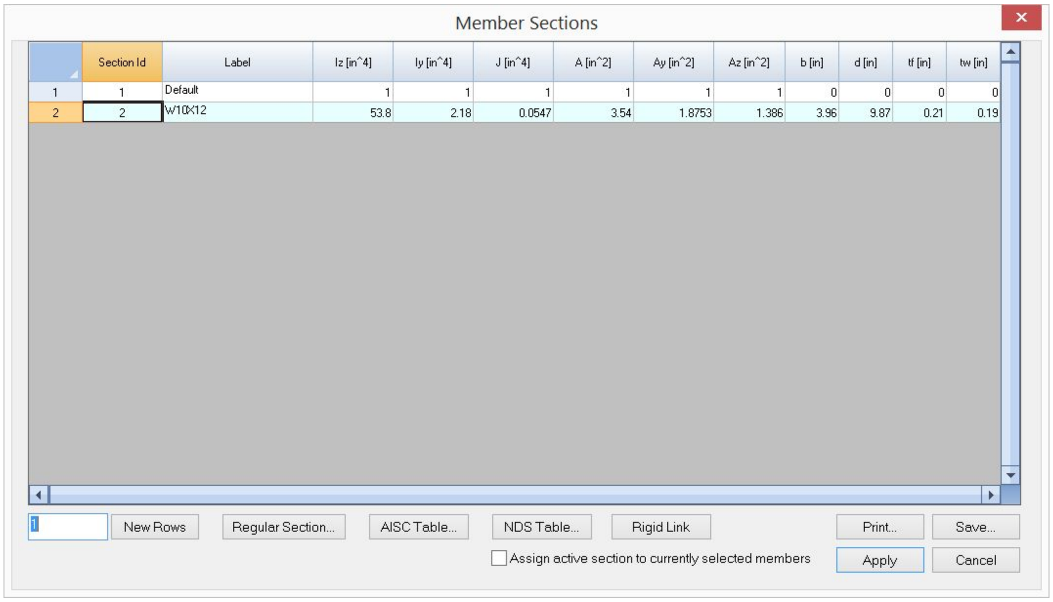
- Set proper units from Settings and Tools > Units & Precisions.



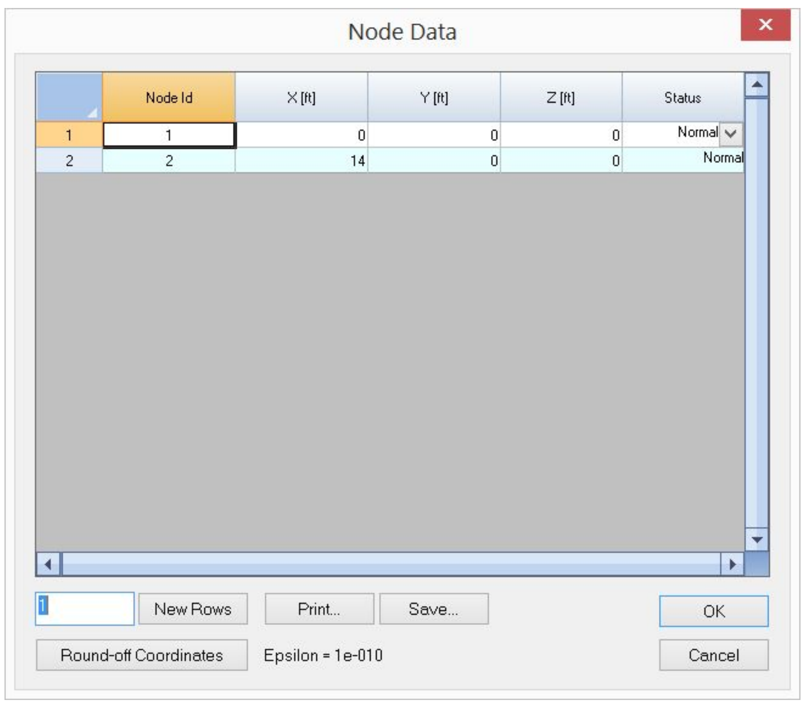
- Define the material from Modify > Member Properties > Materials using the standard steel Steel-A992--Fy50. Steel properties such as Fy and Fu are set automatically.



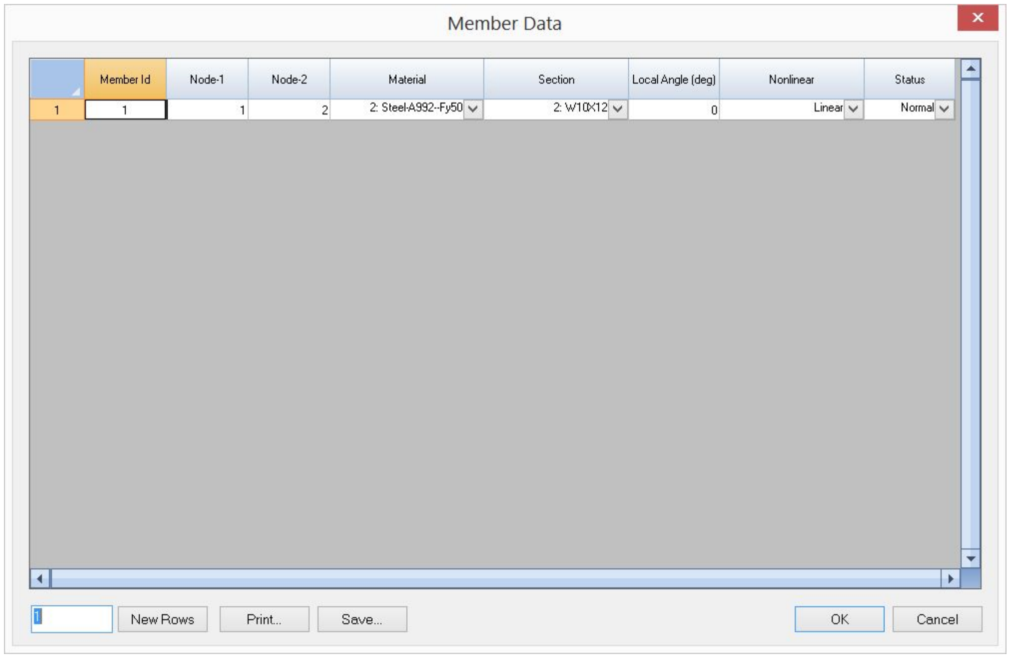
- Define the section W10x12 (or any W-shape) from Modify > Member Properties > Sections using the AISC table.



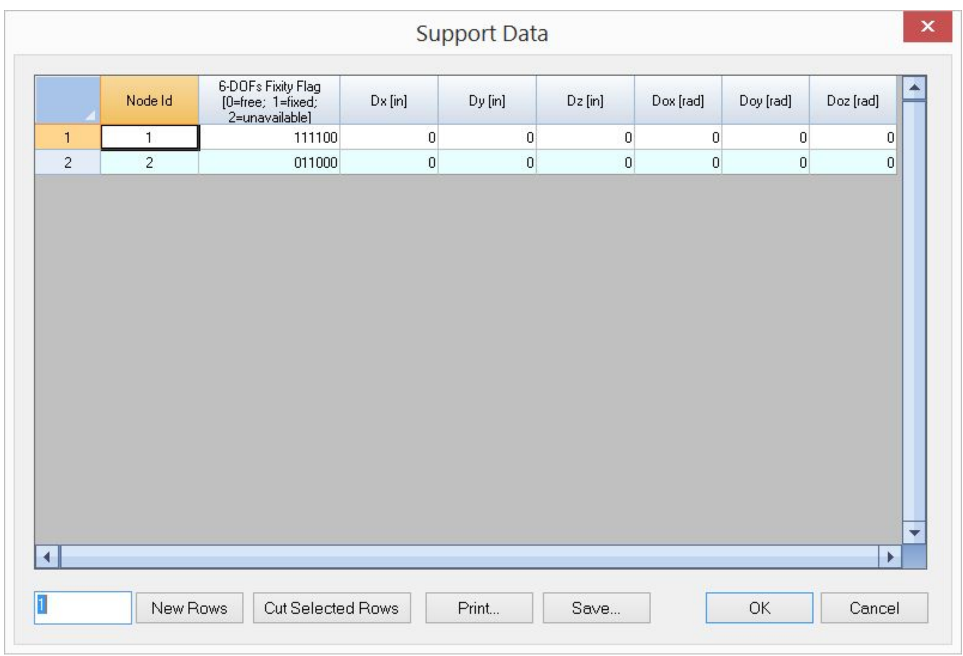
- Define the two nodes from Tables > Nodes.



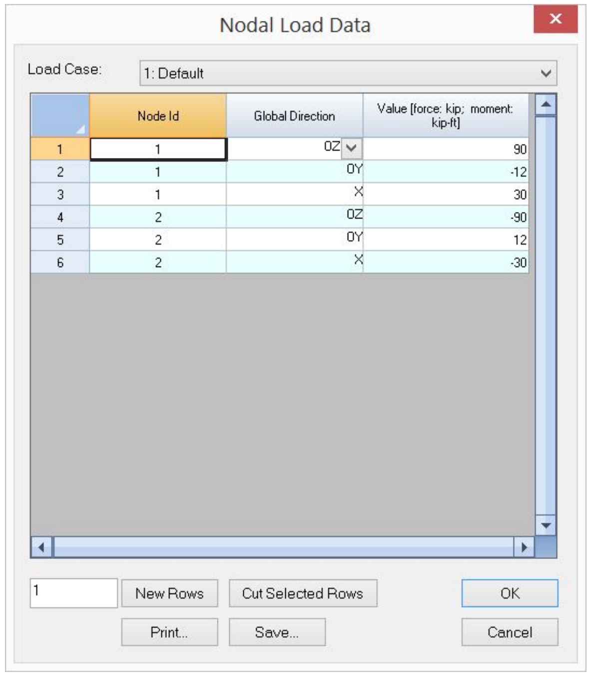
- Define the one beam from Tables > Members



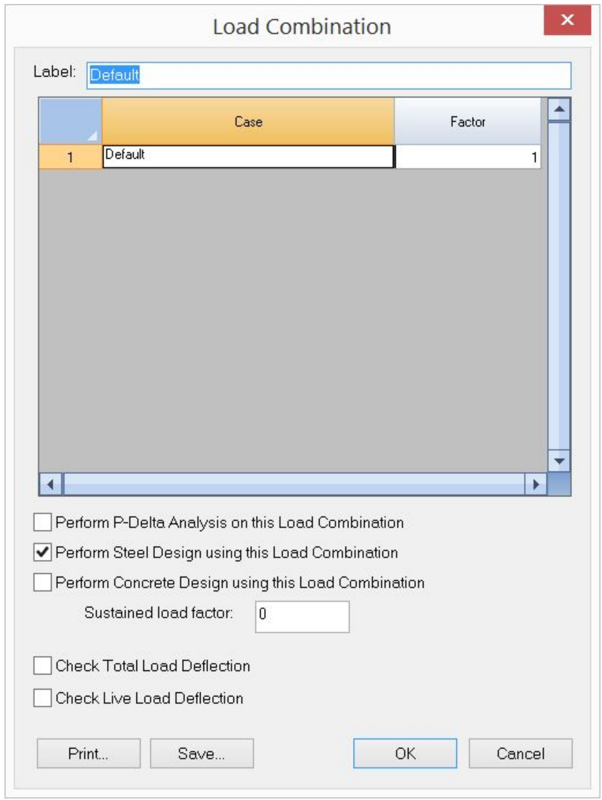
- Define the one supports from Tables > Supports. Please note the first node has X, Y Z, and OX DOFs fixed. The second node has Y and Z DOFs fixed. The fixity in OX direction at the first node is needed to ensure the stability of the 3D Frame.



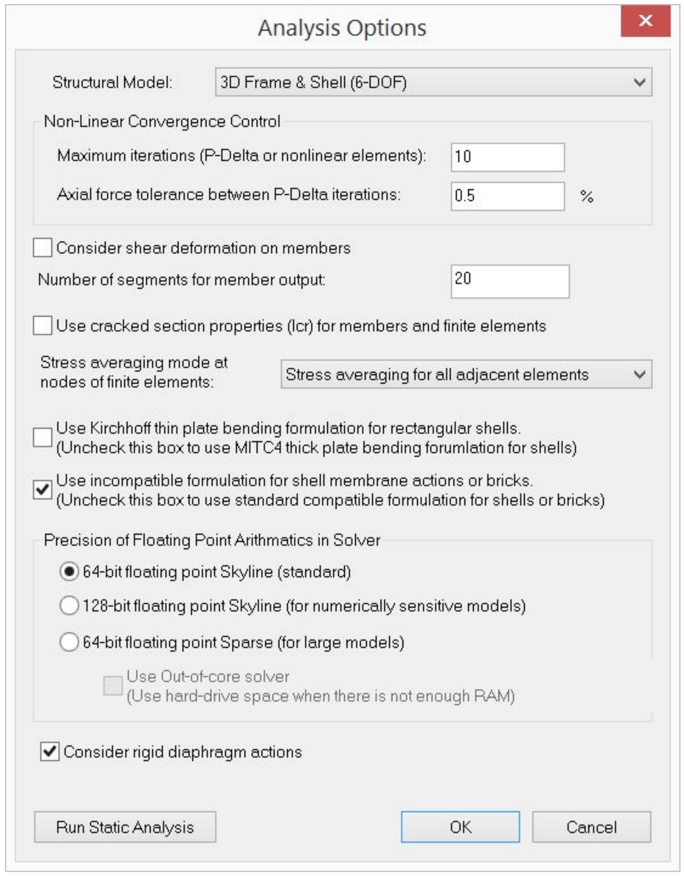
- Define the nodal loads from Tables > Nodal Loads. Please note we enter the load effects as nodal loads as we do not have the exact load condition in the original example. We need to enter Cb manually later instead of letting the program to calculate it for us automatically.



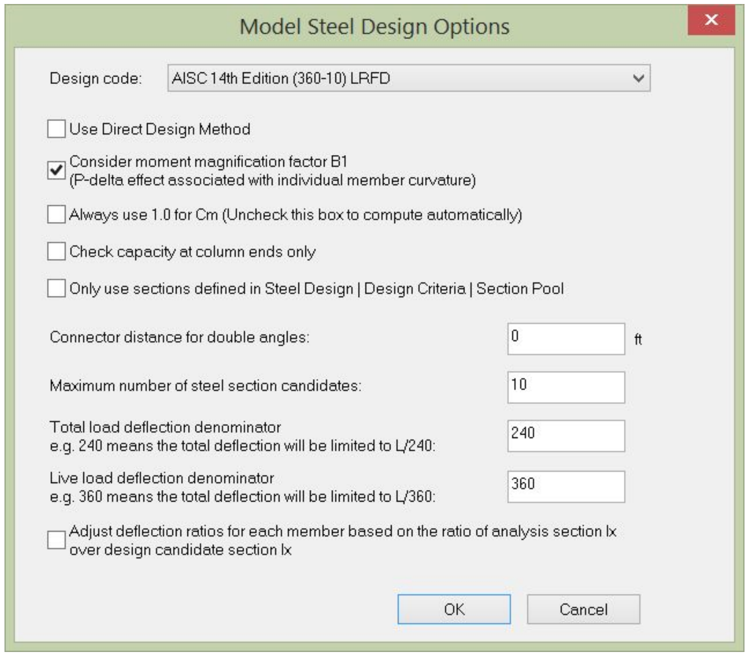
- Define the load combination from Create > Load Combinations. Make sure the "Perform Steel Design using this Load Combination" is checked.



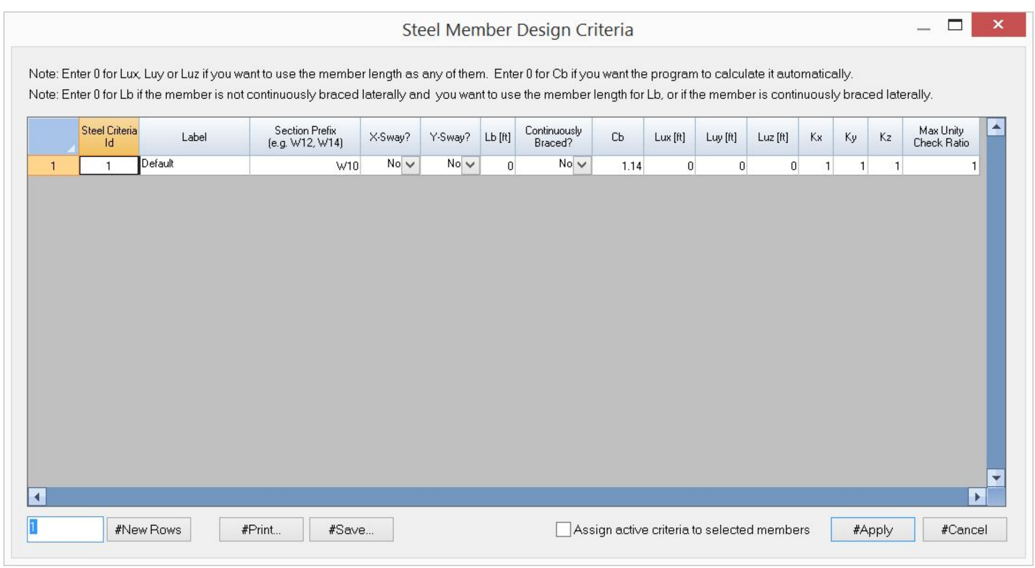
- Set structural model as 3D Frame from Analysis > Analysis Options. Run Static Analysis to make static analysis results available for steel design.



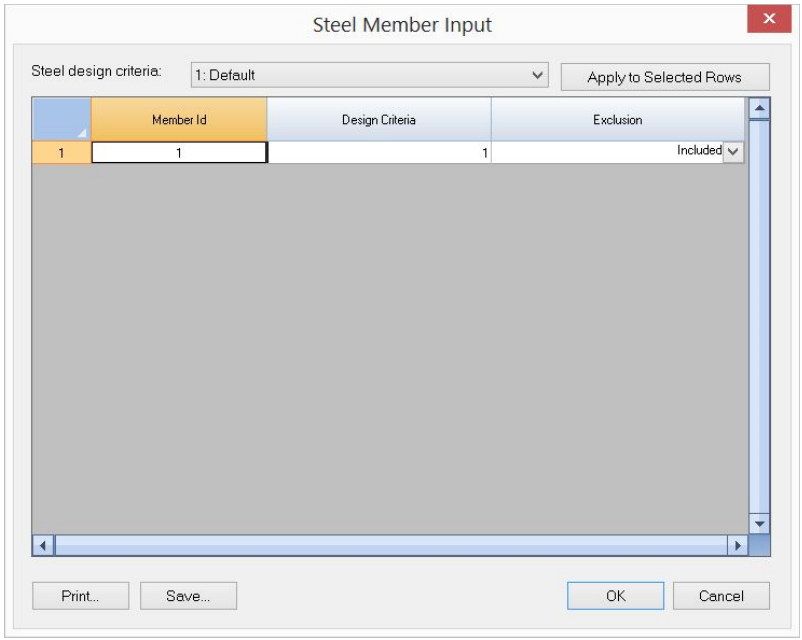
- Define the model design option from Steel Design > Steel Design Criteria > Steel Design Criteria. Make sure the “Consider moment magnification factor B1” is checked.



- Define the steel member design criteria from Steel Design > Steel Design Criteria > Steel Member Design Criteria. Use “W10” as the section prefix as we want to find the lightest W10 section. For this example, we manually enter Cb = 1.14 (The program would calculate Cb automatically if Cb is entered 0.0). Also, since we set Lb, Lux, Luy and Luz to be zero, the program will use the member actual length for each of them.



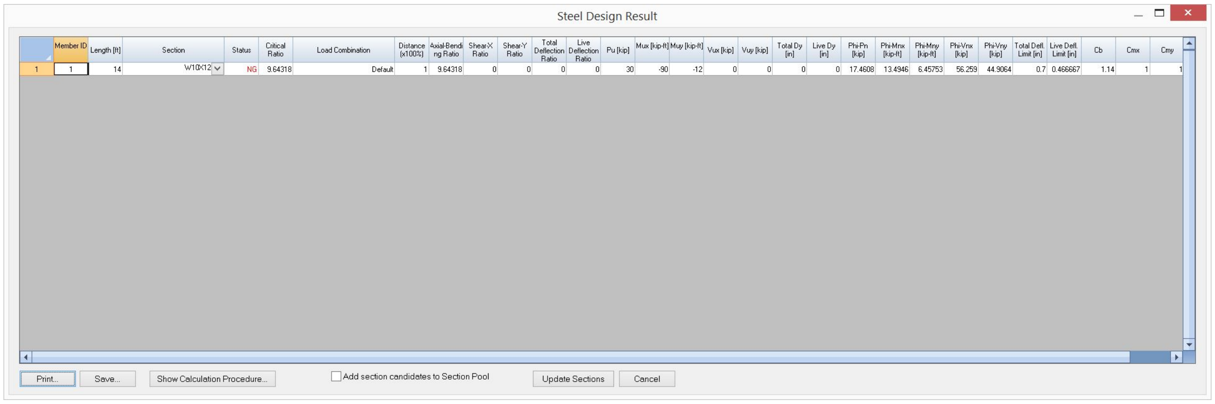
- Define the steel member input from Steel Design > Steel Member Input.

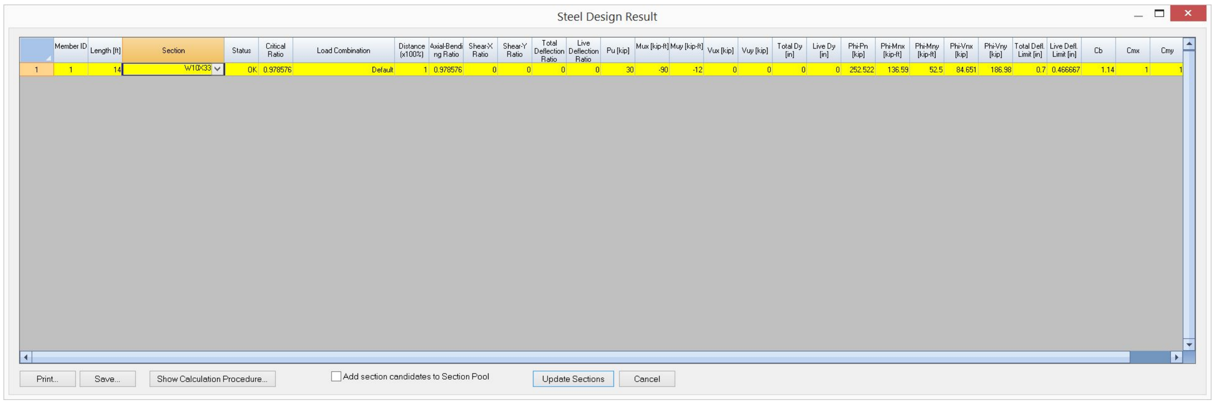


- Perform the steel design from Steel Design > Perform Steel Design.

- View the steel design results from Steel Design > Steel Design Results. By default, up to 10 candidate sections are available. The original section W10x12 is not adequate with critical ratio = 9.64318 (> 1.0). The first section that is adequate is W10x33 with critical ratio = 0.978576 (< 1.0). At this point, you can update the member section to be W10x33, reanalyze the model and perform steel design again.

You can also view the detailed step-by-step calculation procedures for the most critical load condition on each member.





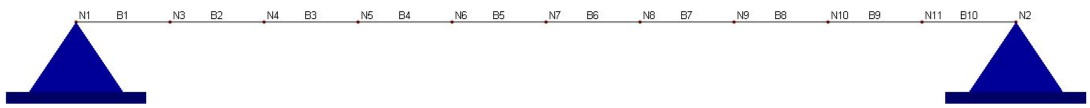
10.19 Example 18: Response Spectrum Analysis of a Beam

Problem Description

A simply supported beam (L = 20 ft) [Ref 24, Problem.4.8] is subjected to a response spectrum in the vertical direction at both supports. The beam section is of size 1.458 in x 14 in.

Material: E = 30 e6 psi, density = 6538.08 lb/ft^3

Damping: 0.0

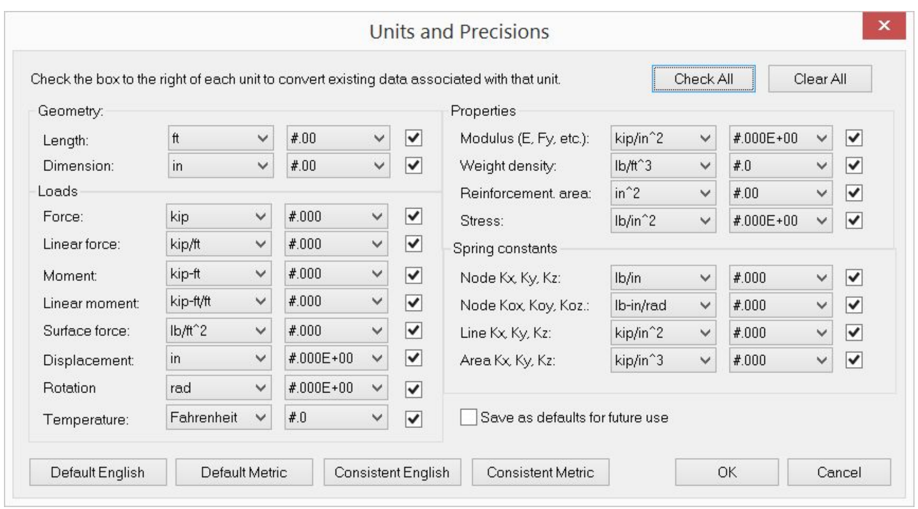


Spectrum Definition:

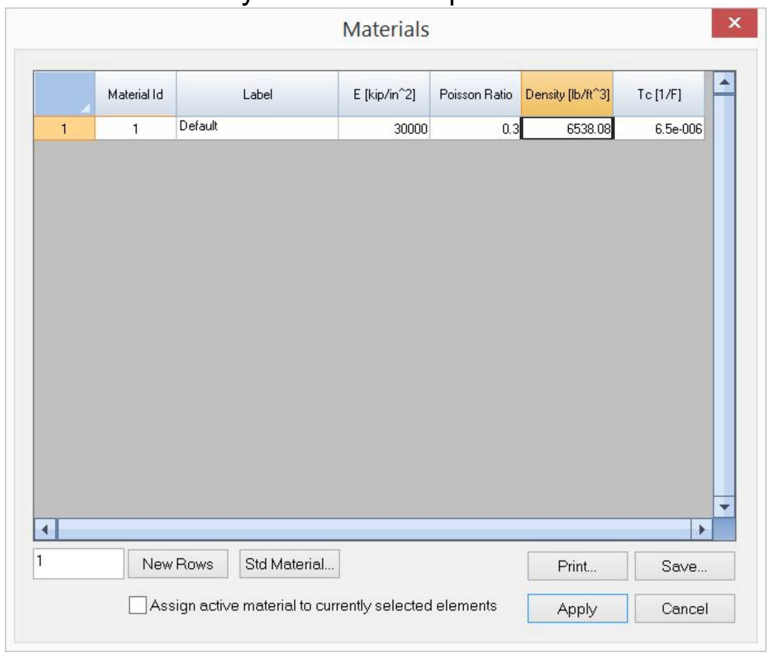
Period (sec)	Spectral Acceleration (g)
0.125	1.45300
0.143	1.42860
0.164	1.63990
0.167	1.66670
0.200	2.00000

Suggested Modeling Steps

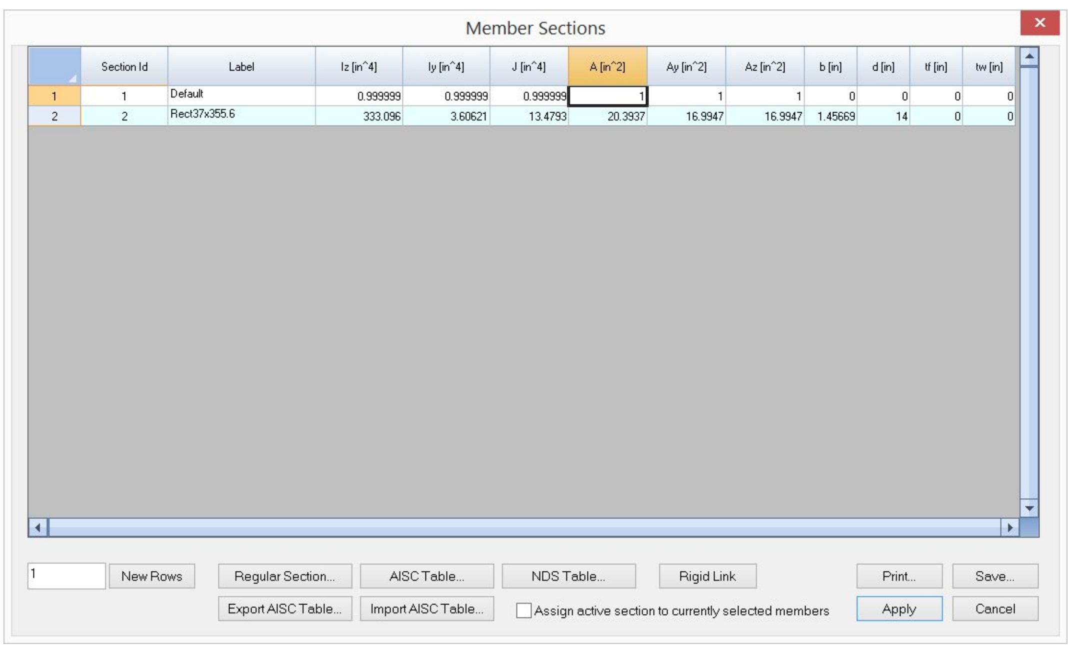
- Set proper units from Settings and Tools > Units & Precisions.



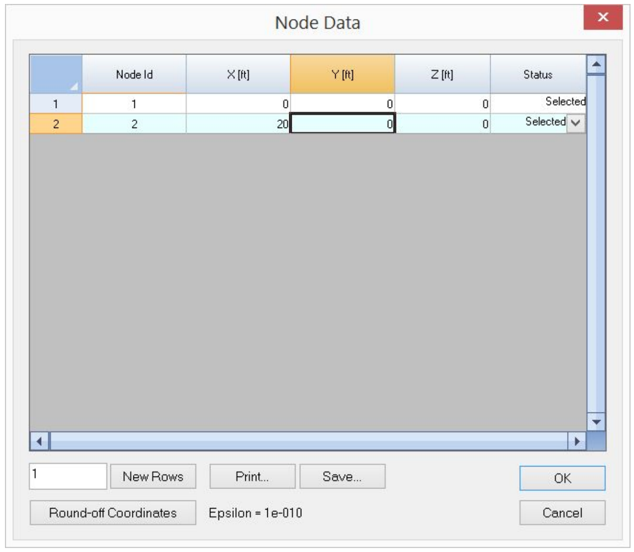
- Define the material from Modify > Member Properties > Materials



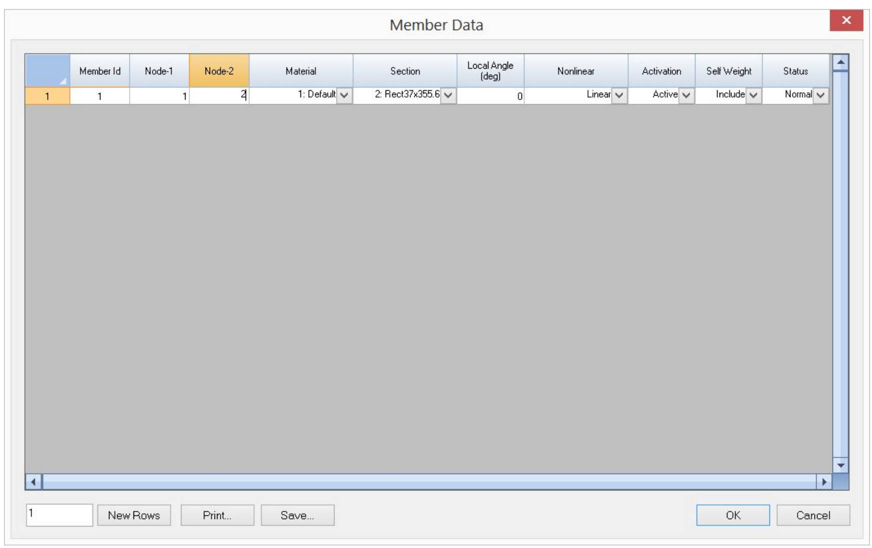
- Define the section from Modify > Member Properties > Sections using the Regular Section button.



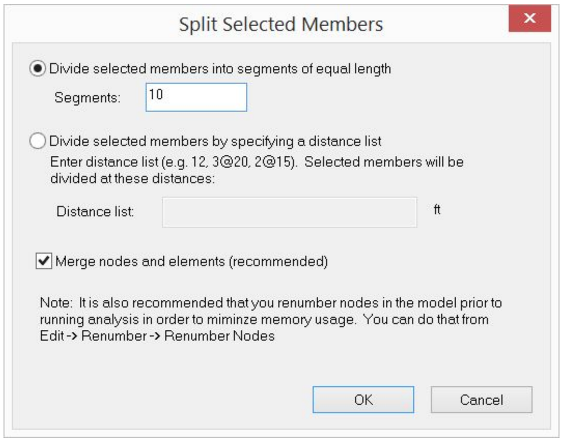
- Define the two nodes from Tables > Nodes.



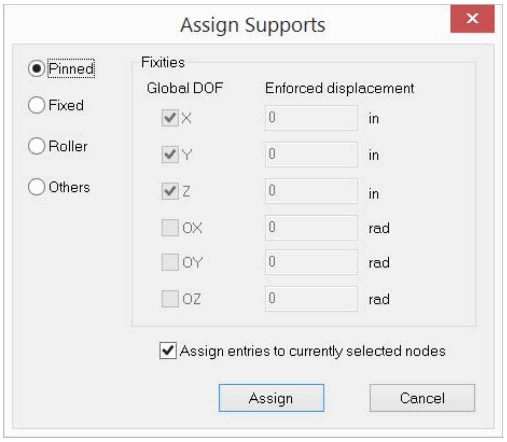
- Define the one beam from Tables > Members. Make sure the correct material and section are used for this beam.



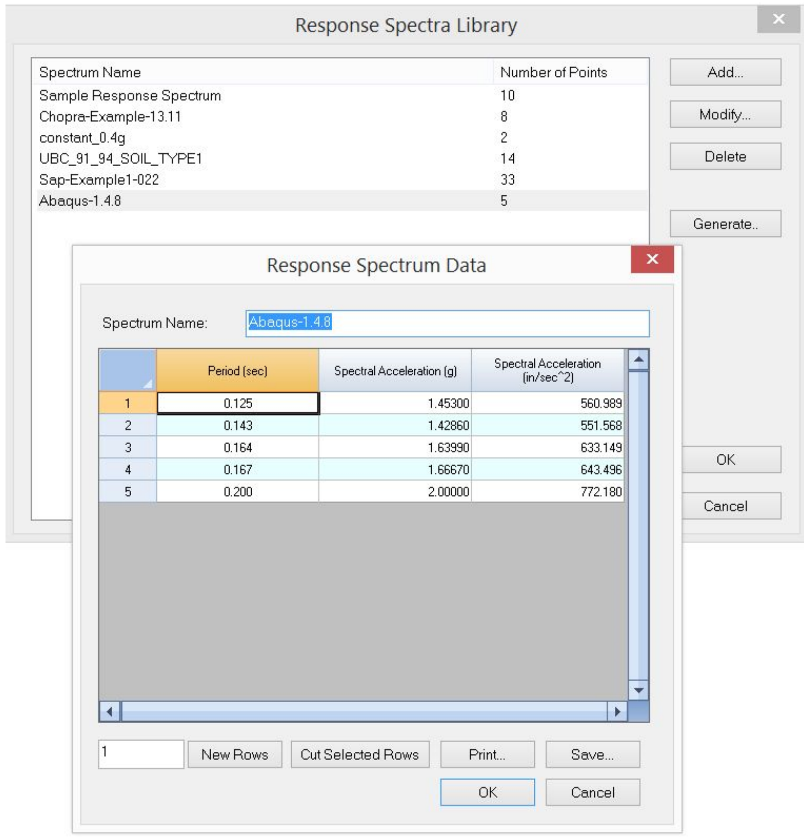
- Select the beam we just created. Use Modify > Split > Split Members to split it to 10 elements of equal lengths.



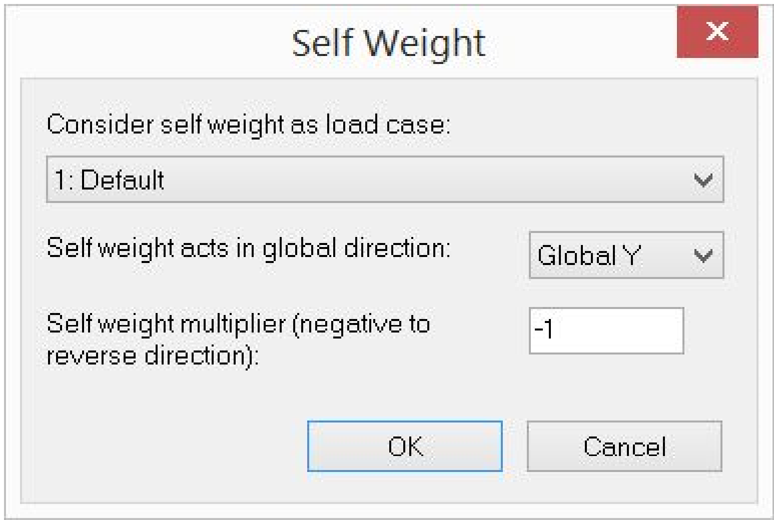
- Use Create > Boundary Conditions > Support to assign supports to node 1 and 2



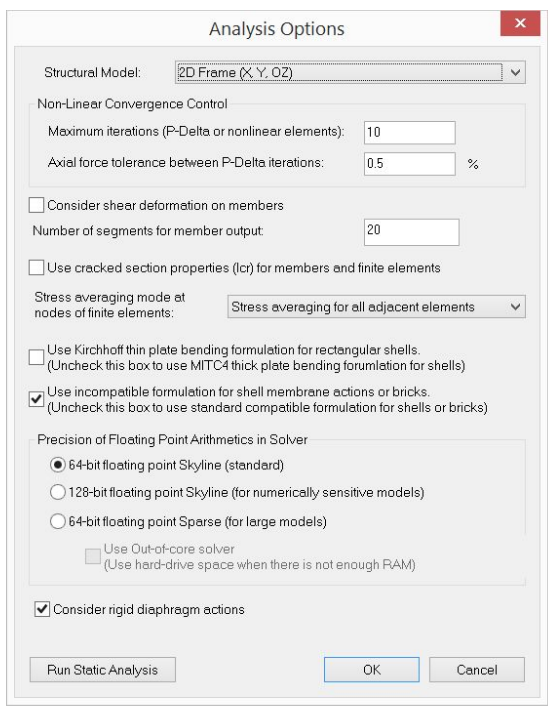
- From Create > Generate Loads > Response Spectra Library > Add, define the spectrum.



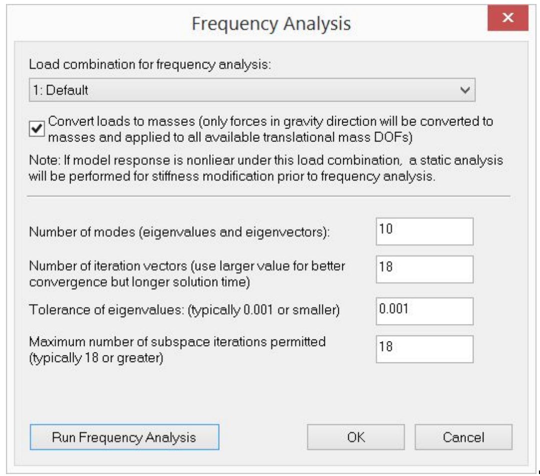
- We will convert self-weight to calculate masses. So from Create > Draw Loads > Self Weights, define the self-weight multiplier as -1 in Global Y direction. By default, self weight will be of load case "Default", which is included in the default load combination "Default".



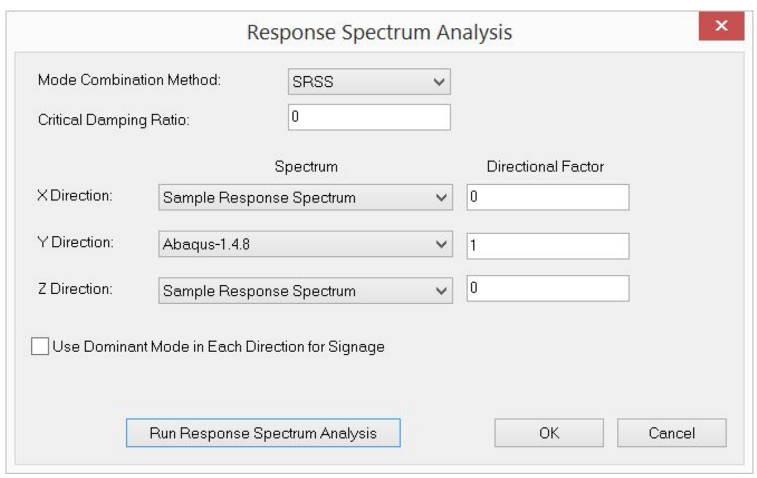
- Set structural model as 2D Frame from Analysis > Analysis Options. We will not consider shear deformation on members.



- Frequency analysis must be run prior to response spectrum analysis. So run frequency analysis from Analysis > Frequency Analysis. We will compute the first 10 modes.



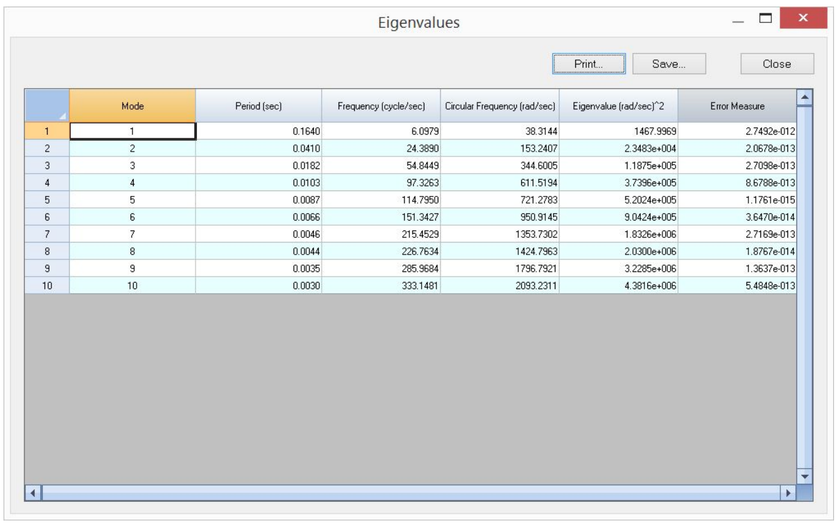
- Run response spectrum analysis from Analysis > Response Spectrum Analysis. Make sure we use the correct spectrum and apply directional factor only in Y direction.



- After the response spectrum analysis is done, we can then exam results such as Analysis Results > Dynamics > Eigenvalues, Analysis Results > Dynamics > Modal Nodal Displacements, Analysis Results > Dynamics > Modal Member End Forces & Moments, etc.

The following is a result comparison between ENERCALC 3D and the reference [Ref 24].

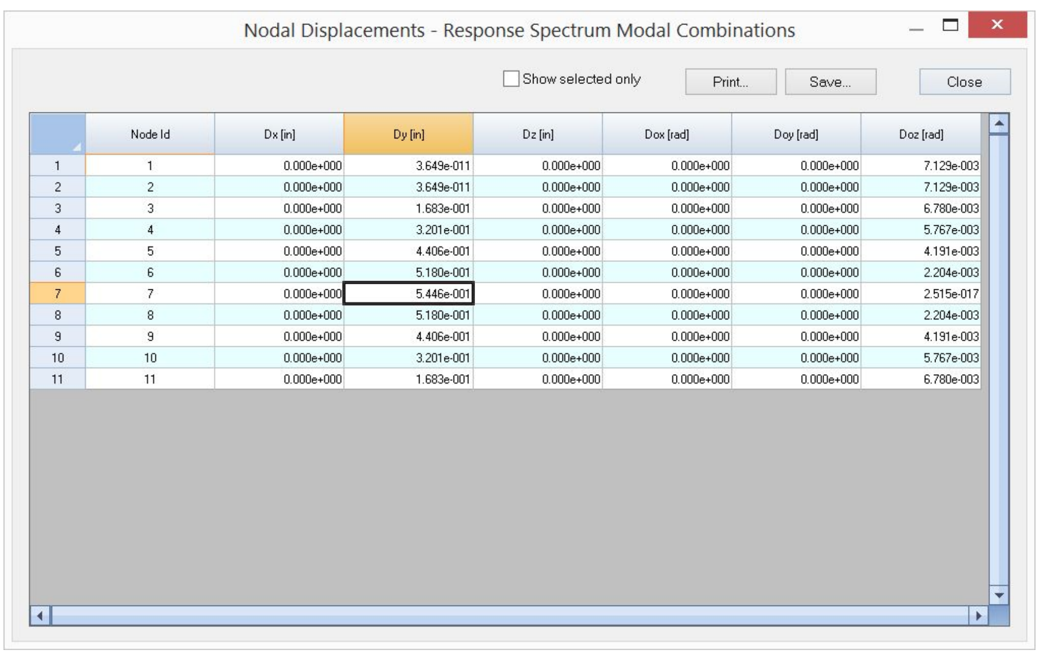
	ENERCALC 3D	Reference
First Mode Frequency	6.0979 Hz	6.098 Hz
Midspan Displacement Dy	0.5446 in	0.549 in
Midspan moment	9.40764e5 lb-in (78.397 kip-ft)	9.493e5 lb-in



Eigenvalues

Print... Save... Close

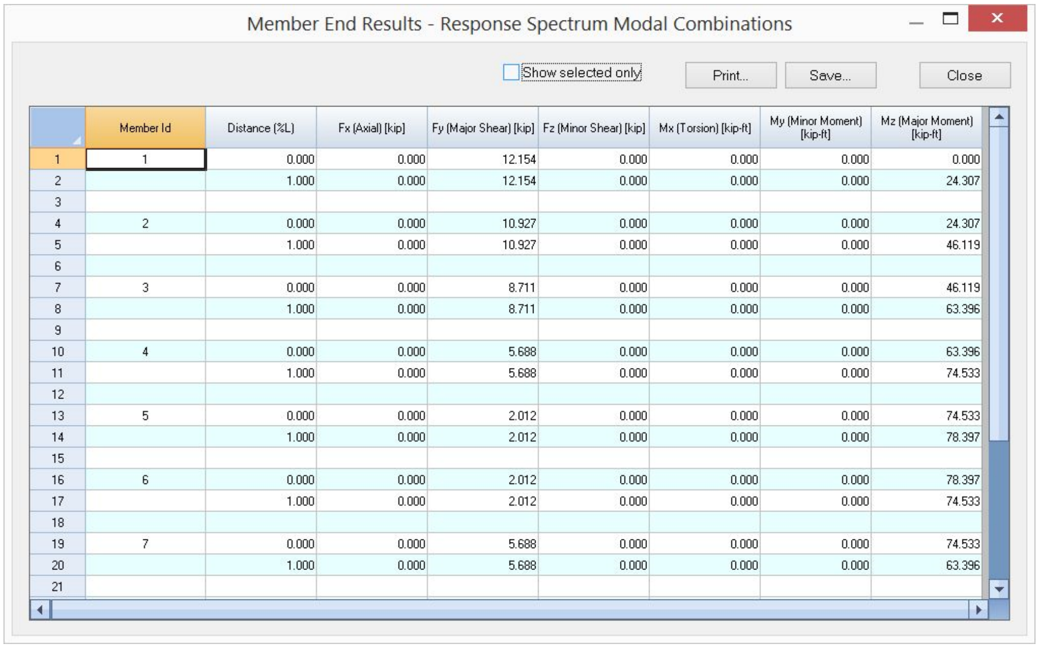
	Mode	Period (sec)	Frequency (cycle/sec)	Circular Frequency (rad/sec)	Eigenvalue (rad/sec)^2	Error Measure
1	1	0.1640	6.0979	38.3144	1467.9969	2.7492e-012
2	2	0.0410	24.3890	153.2407	2.3483e+004	2.0678e-013
3	3	0.0182	54.8449	344.6005	1.1875e+005	2.7098e-013
4	4	0.0103	97.3263	611.5194	3.7396e+005	8.6788e-013
5	5	0.0087	114.7950	721.2783	5.2024e+005	1.1761e-015
6	6	0.0066	151.3427	950.9145	9.0424e+005	3.6470e-014
7	7	0.0046	215.4529	1353.7302	1.8326e+006	2.7169e-013
8	8	0.0044	226.7634	1424.7963	2.0300e+006	1.8767e-014
9	9	0.0035	285.9684	1796.7921	3.2285e+006	1.3637e-013
10	10	0.0030	333.1481	2093.2311	4.3816e+006	5.4848e-013



Nodal Displacements - Response Spectrum Modal Combinations

Show selected only Print... Save... Close

	Node Id	Dx [in]	Dy [in]	Dz [in]	Dox [rad]	Doy [rad]	Doz [rad]
1	1	0.000e+000	3.649e-011	0.000e+000	0.000e+000	0.000e+000	7.129e-003
2	2	0.000e+000	3.649e-011	0.000e+000	0.000e+000	0.000e+000	7.129e-003
3	3	0.000e+000	1.683e-001	0.000e+000	0.000e+000	0.000e+000	6.780e-003
4	4	0.000e+000	3.201e-001	0.000e+000	0.000e+000	0.000e+000	5.767e-003
5	5	0.000e+000	4.406e-001	0.000e+000	0.000e+000	0.000e+000	4.191e-003
6	6	0.000e+000	5.180e-001	0.000e+000	0.000e+000	0.000e+000	2.204e-003
7	7	0.000e+000	5.446e-001	0.000e+000	0.000e+000	0.000e+000	2.515e-017
8	8	0.000e+000	5.180e-001	0.000e+000	0.000e+000	0.000e+000	2.204e-003
9	9	0.000e+000	4.406e-001	0.000e+000	0.000e+000	0.000e+000	4.191e-003
10	10	0.000e+000	3.201e-001	0.000e+000	0.000e+000	0.000e+000	5.767e-003
11	11	0.000e+000	1.683e-001	0.000e+000	0.000e+000	0.000e+000	6.780e-003


Member End Results - Response Spectrum Modal Combinations

Show selected only | Print... | Save... | Close

	Member Id	Distance (%L)	Fx (Axial) [kip]	Fy (Major Shear) [kip]	Fz (Minor Shear) [kip]	Mx (Torsion) [kip-ft]	My (Minor Moment) [kip-ft]	Mz (Major Moment) [kip-ft]
1	1	0.000	0.000	12.154	0.000	0.000	0.000	0.000
2		1.000	0.000	12.154	0.000	0.000	0.000	24.307
3								
4	2	0.000	0.000	10.927	0.000	0.000	0.000	24.307
5		1.000	0.000	10.927	0.000	0.000	0.000	46.119
6								
7	3	0.000	0.000	8.711	0.000	0.000	0.000	46.119
8		1.000	0.000	8.711	0.000	0.000	0.000	63.396
9								
10	4	0.000	0.000	5.688	0.000	0.000	0.000	63.396
11		1.000	0.000	5.688	0.000	0.000	0.000	74.533
12								
13	5	0.000	0.000	2.012	0.000	0.000	0.000	74.533
14		1.000	0.000	2.012	0.000	0.000	0.000	78.397
15								
16	6	0.000	0.000	2.012	0.000	0.000	0.000	78.397
17		1.000	0.000	2.012	0.000	0.000	0.000	74.533
18								
19	7	0.000	0.000	5.688	0.000	0.000	0.000	74.533
20		1.000	0.000	5.688	0.000	0.000	0.000	63.396
21								

Part XI

11 References

1. K.J. Bathe, “Finite Element Procedures” Prentice-Hall, Inc., 1996
2. O.C. Zinenkiewicz, “The Finite Element Method”, 3rd ed., McGraw-Hill Book Company (UK) Limited, 1983
3. R.C. Cook & D.S. Malkus & M.E. Plesha, “Concepts and Applications of Finite Element Analysis”, 3rd ed., John Wiley & Sons, Inc., 1989
4. R.H. Macneal & R.L. Harder, “A proposed Standard Set of Problems to Test Finite Element Accuracy”, pp3-20 of “Finite Elements in Analysis and Design”, North-Holland, 1985
5. J.P. Hartog, “Advanced Strength of Materials”, McGraw-Hill Book Company, 1952.
6. R.J. Roark & W.C. Young, “Formulas for Stress and Strain”, 5th ed., McGraw-Hill Book Company, 1975.
7. W. McGuire & R.H. Gallagher & R.D. Ziemian, “Matrix Structural Analysis”, 2nd ed., John Wiley & Sons, Inc., 2000
8. J.S. Przemieniecki, “Theory of Matrix Structural Analysis”, McGraw-Hill, 1968
9. D. Breyer, “Design of Wood Structures”, 3rd ed, McGraw-Hill, 1993
10. J. MaCormac, “Structural Steel Design LRFD Method”, 2nd ed, HarperCollins College Publishers, 1995
11. A. Tamboli, “Steel Design Handbook LRFD Method”, McGraw-Hill, 1997
12. ACI, “Building Code Requirements for Reinforced Concrete (ACI 318-89) (Revised 1992)”, American Concrete Institute, Detroit, Michigan.
13. K. Leet & D. Bernal, “Reinforced Concrete Design”, 3rd ed, McGraw-Hill Book Company, 1997.
14. A. K. Chopra, “Dynamic of Structures – Theory and Applications to Earthquake Engineering”, 2nd ed, Prentice-Hall, 2001
15. Ansel Ugural, “Stresses in Plates and Shells” 2nd Edition, The GcGraw-Hill Companies, Inc., 1999
16. James G. MacGregor & James K. Wight, “Reinforced Concrete – Mechanics and Design” 4th Edition, Pearson Prentice Hall, 2005
17. “Notes on ACI 318-02 Building Code Requirements for Structural Concrete”, 8th Edition, Portland Cement Association, 2002
18. R. Park and W.L. Gamble “Reinforced Concrete Slabs”, John Wiley & Sons, 1980

19. ACI, “Building Code Requirements for Structural Concrete (ACI 318-05) and Commentary (ACI 318R-05)”, American Concrete Institute, Detroit, Michigan, 2004

20. Arthur H. Nilson, David Darwin, Charles W. Dolan, “Design of Concrete Structures”, 13th Edition, McGraw-Hill Higher Education, 2004

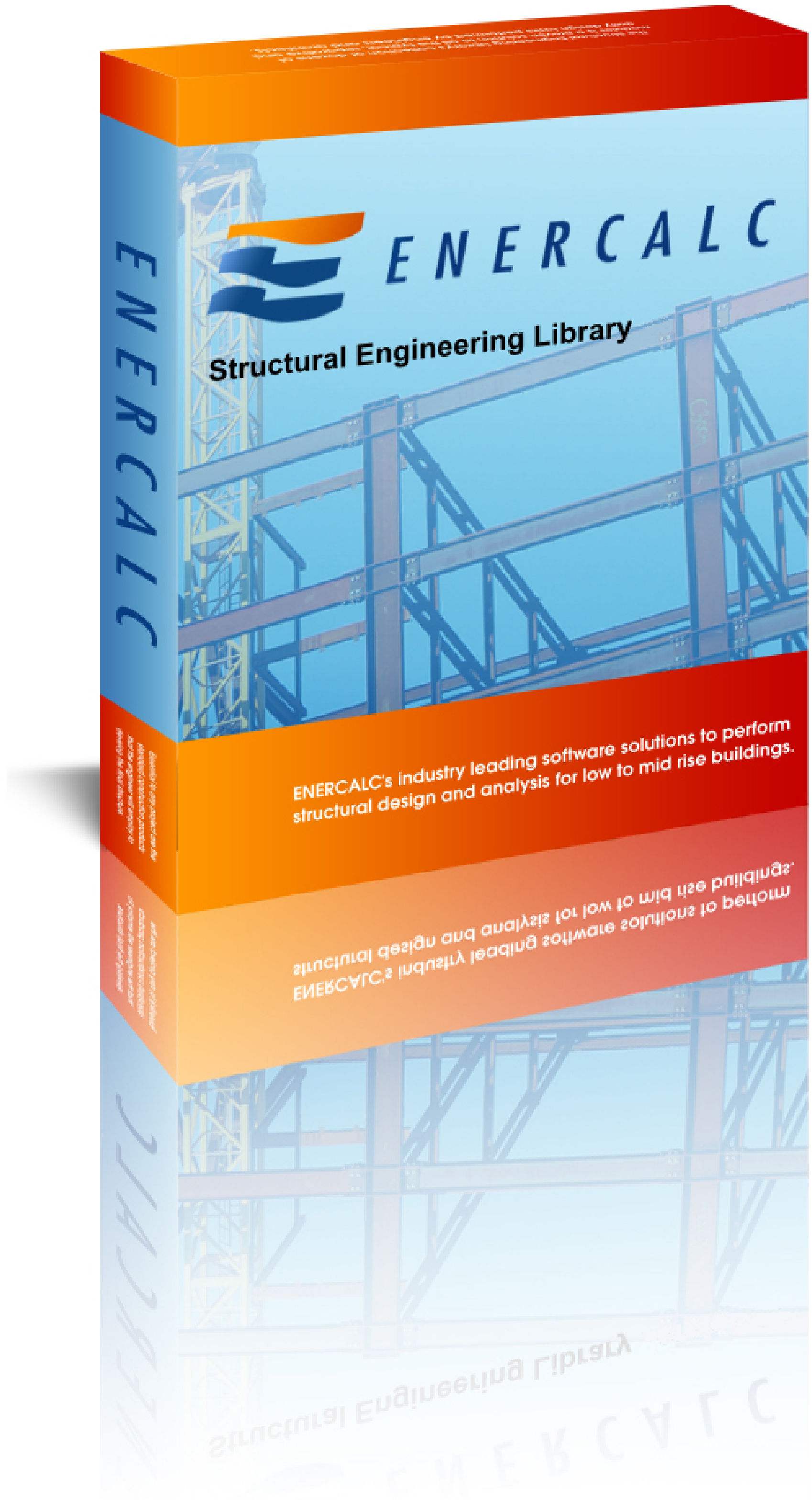
21. AISC “Steel Construction Manual”, 14th Edition

22. Charles Salmon, John Johnson and Faris Malhas, “Steel Structures” 5th Edition, Pearson Prentice Hall, 2009

23. AISC “Design Examples”, Version 14.1

24. Abaqus Benchmarks Guide, Dassault Systemes, 2013

25. ADINA Verification Manual, ADINA R & D Inc., June 2001


ENERCALC
Structural Engineering Library
ENERCALC's industry leading software solutions to perform structural design and analysis for low to mid rise buildings.