

Software Verification Examples



ETABS®

Integrated Software for
Structural Analysis and Design

SOFTWARE VERIFICATION EXAMPLES



Computers and Structures, Inc.
Berkeley, California, USA

Version 8
June 2003

Copyright

The computer program ETABS and all associated documentation are proprietary and copyrighted products. Worldwide rights of ownership rest with Computers and Structures, Inc. Unlicensed use of the program or reproduction of the documentation in any form, without prior written authorization from Computers and Structures, Inc., is explicitly prohibited.

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA

Phone: (510) 845-2177

FAX: (510) 845-4096

e-mail: info@csiberkeley.com (for general questions)

e-mail: support@csiberkeley.com (for technical support questions)

web: www.csiberkeley.com

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF ETABS. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.



Contents

Introduction

Examples

- 1 Plane Frame with Beam Span Loads, Static Gravity Load Analysis
- 2 Three-Story Plane Frame, Dynamic Response Spectrum Analysis
- 3 Three-Story Plane Frame, Code-Specific Static Lateral Load Analysis
- 4 Single Story Three-Dimensional Frame, Dynamic Response Spectrum Analysis
- 5 Three-Story Three-Dimensional Braced Frame, Dynamic Response Spectrum Analysis
- 6 Nine-Story Ten-Bay Plane Frame, Eigenvalue Analysis
- 7 Seven-Story Plane Frame, Gravity and Lateral Loads Analysis
- 8 Two-Story Three-Dimensional Frame, Dynamic Response Spectrum Analysis
- 9 Two-Story Three-Dimensional Unsymmetrical Building Frame, Dynamic Response Spectrum Analysis
- 10 Three-Story Plane Frame with ADAS Elements, Nonlinear Time History Analysis

-
- 11 Three-Story Plane Frame with Viscous Damper Elements, Nonlinear Time History Analysis
 - 12 Pounding of Two Planar Frames, Nonlinear Time History Analysis
 - 13 Base Isolated Two-Story 3D Frame, Nonlinear Time History Analysis
 - 14 Friction Pendulum Base-Isolated 3D Frame, Nonlinear Time History Analysis
 - 15 Wall Area Object Behavior, Static Lateral Loads Analysis



Introduction

This manual presents a set of simple building systems that have been analyzed using ETABS Version 8. The examples demonstrate some of the analytical capabilities of the ETABS system. For purposes of verification, key results produced by ETABS are compared to independent sources, such as hand calculated results, theoretical or published results, or results obtained from other structural/finite element programs that are verified not using ETABS. The examples cover each type of element, static and dynamic analysis and linear and nonlinear options.

For each example, this manual contains a short description of the problem; a list of significant ETABS options activated; and a comparison of key results with theoretical results or results from other computer programs. The input data file for each example is provided on the ETABS CD.

Plane Frame with Beam Span Loads Static Gravity Load Analysis

Description

This is a one-story, two-dimensional frame subjected to vertical static loading.

The frame geometry and loading patterns are shown in Figure 1-1.

Significant Options of ETABS Activated

- Two-dimensional frame analysis
- Vertical beam span loading
- No rigid joint offsets on beams and columns
- Column pinned end connections

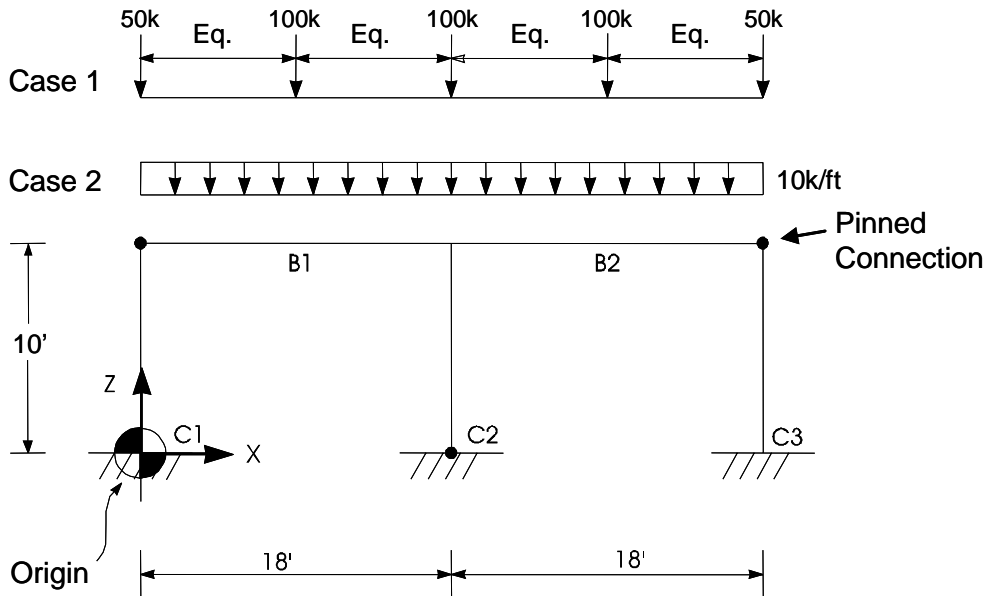


Figure 1-1 Plane Frame with Beam Span Loads

Computer Model

The frame is a three-column line, two-bay system. Kip-inch-second units are used. The modulus of elasticity is 3000 ksi. All columns are 12"x24"; all beams are 12"x30".

To be able to compare ETABS results with theoretical results using prismatic members and elementary beam theory, rigid joint offsets on columns and beams are not modeled, and axial and shear deformations are neglected. Thus, the automatic property generation feature of ETABS is not used; instead, the axial area and moment of inertia for each member are explicitly input.

The input data file for this example is EX1.EDB. This file is available to the user on the program CD.

Comparison of Results

The theoretical results for bending moments and shear forces on beams B1 and B2 are easily obtained from tabulated values for propped cantilevers (American

Institute of Steel Construction 1989). These values for beam B1 are compared with ETABS results in Table 1-1. The results are identical.

Table 1-1 Comparison of Results for Beam B1

Quantity	Location	Load Case I (Concentrated Load)	
		ETABS	Theoretical
Bending Moments	End I	0.00	0.00
	¼ Point	1,687.50	1,687.50
	½ point	3,375.00	3,375.00
	¾ point	-337.50	-337.50
	End J	-4,050.00	-4,050.00
Shear Forces	End I	-31.25	-31.25
	¼ Point	-31.25	-31.25
	½ point	68.75	68.75
	¾ point	68.75	68.75
	End J	68.75	68.75
Quantity	Location	Load Case II (Uniformly Distributed Load)	
		ETABS	Theoretical
Bending Moments	End I	0.00	0.00
	¼ Point	2,430.00	2,430.00
	½ point	2,430.00	2,430.00
	¾ point	0.00	0.00
	End J	-4,860.00	-4,860.00
Shear Forces	End I	-67.50	-67.50
	¼ Point	-22.50	-22.50
	½ point	22.50	22.50
	¾ point	67.50	67.50
	End J	112.50	112.50

Reference

American Institute of Steel Construction 1989. *Manual of Steel Construction-Allowable Stress Design*. Chicago, Illinois.

Three-Story Plane Frame Dynamic Response Spectrum Analysis

Description

This is a three-story plane frame subjected to the El Centro 1940 seismic response spectra, N-S component, 5 percent damping.

The frame geometry is shown in Figure 2-1.

Significant Options of ETABS Activated

- Two-dimensional frame analysis
- Rigid joint offsets on beams and columns automatically calculated
- Dynamic response spectrum analysis

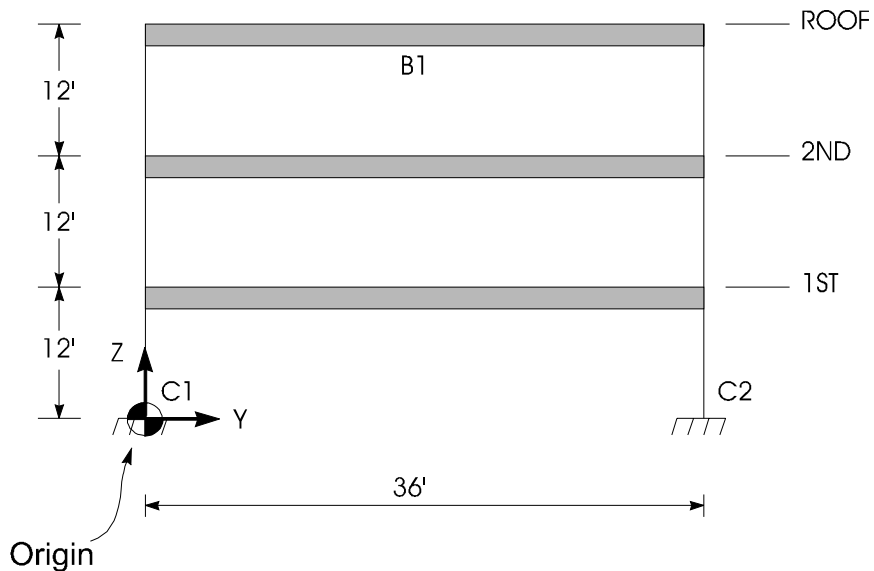


Figure 2-1 Three-Story Plane Frame

Computer Model

The frame is modeled as a two-column line, single bay system. Kip-inch-second units are used. Other parameters associated with the structure are as follows:

All columns are W14X90

All beams are infinitely rigid and 24" deep

Modulus of elasticity = 29500 ksi

Typical story mass = 0.4 kip-sec²/in

The column is modeled to have infinite axial area, so that axial deformation is neglected. Also, zero column shear area is input to trigger the ETABS option of neglecting shear deformations. These deformations are neglected to be consistent with the hand-calculated model with which the results are compared.

The input data file for this example is EX2.EDB. The response spectrum file is ELCN-RS1. These files are available on the CD.

Comparison of Results

Assuming the beams to be rigid and a rigid offset at the column top ends of 24 inches (i.e. equal to the depth of the beams), and neglecting both shear deformations and axial deformations, the story lateral stiffness for this example can be calculated (Przemieniecki 1968).

The example then reduces to a three-spring, three-mass system with equal stiffnesses and masses. This can be analyzed using any exact method (Paz 1985) to obtain the three natural periods and mass normalized mode shapes of the system.

The three theoretical natural periods and mass normalized mode shapes are compared in Table 2-1 with ETABS results. The results are identical.

Table 2-1 Comparison of Results for Periods and Mode Shapes

Mode	Period, secs.	Mode Shape	ETABS	Theoretical
1	0.4414	Roof Level	1.165	1.165
		2 nd Level	0.934	0.934
		1 st Level	0.519	0.519
2	0.1575	Roof Level	0.934	0.934
		2 nd Level	-0.519	-0.519
		1 st Level	-1.165	-1.165
3	0.1090	Roof Level	0.519	0.519
		2 nd Level	-1.165	-1.165
		1 st Level	0.934	0.934

The spectral accelerations at the three natural periods can then be linearly interpolated from the response spectrum used.

The spectral accelerations can in turn be used with the mode shapes and story mass information to obtain the modal responses (Paz 1985). The modal responses for story displacements and column moments can then be combined using the complete quadratic combination procedure (Wilson, et al. 1981).

The story displacements and column moments thus obtained are compared in Table 2-2 with ETABS results. The results are identical.

Table 2-2 Comparison of Displacements and Column Moments

Quantity	ETABS	Theoretical
Displacement at		
Roof	2.139	2.139
2 nd	1.716	1.716
1 st	0.955	0.955
Moment, Column C1, at Base	11,730	11,730

References

Paz, M. 1985. *Structural Dynamics, Theory and Computations*, Van Nostrand Reinhold.

Przemieniecki, J.S. 1968. *Theory of Matrix Structural Analysis*. Mc-Graw-Hill.

Wilson, E.L., A.D. Kiureghian and E.P. Bayo 1981. *A Replacement for the SRSS Method in Seismic Analysis*. Earthquake Engineering and Structural Dynamics, Vol. 9.

Three-Story Plane Frame, Code-Specified Static Lateral Load Analysis

Description

This is a three-story plane frame subjected to the following three code-specified lateral load cases:

- ❑ UBC 1997 specified seismic loads (International Conference of Building Officials 1997)
- ❑ IBC 2000 specified seismic loads (International Code Council, Inc. 2000)
- ❑ UBC 1997 specified wind loads (International Conference of Building Officials 1997)

The frame geometry is shown in Figure 3-1.

Significant Options of ETABS Activated

- ❑ Two-dimensional frame analysis

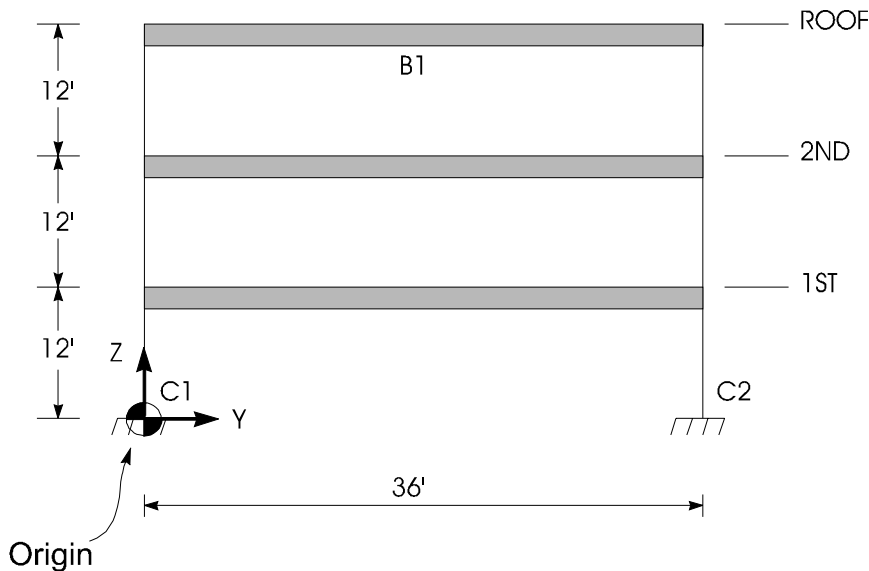


Figure 3-1 Three-Story Plane Frame

- ❑ Section properties automatically recovered from AISC database
- ❑ Automatic generation of UBC 1997 seismic loads
- ❑ Automatic generation of IBC 2000 seismic loads
- ❑ Automatic generation of UBC 1997 wind loads

Computer Model

The frame is modeled as a two-column line, single bay system. Kip-inch-second units are used. Other parameters associated with the structure are as follows:

All columns are W14X90	
All beams are infinitely rigid and 24" deep	
Modulus of elasticity	= 29500 ksi
Poisson's ratio	= 0.3
Typical story mass	= 0.4 kip-sec ² /in

The input data file for this example is EX3.EDB. This file is available on the CD.

For the UBC97 seismic load analysis, the code parameters associated with the analysis are as follows:

UBC Seismic zone factor, Z	= 0.40
UBC Soil Profile Type	= SC
UBC Importance factor, I	= 1.25
UBC Overstrength Factor	= 8.5
UBC coefficient C_t	= 0.035
UBC Seismic Source Type	= B
Distance to Source	= 15 km

For the IBC2000 seismic load analysis, the code parameters associated with the analysis are as follows:

IBC Site Class	= C
IBC Response Accel, S_s	= 1
IBC Response Accel, S_1	= 0.4
IBC Response Modification, R	= 8
IBC Coefficient C_t	= 0.035
IBC Seismic Group	= I

For the UBC97 wind load analysis, the exposure and code parameters associated with the analysis are as follows:

Width of structure supported by frame	= 20 ft
UBC Basic wind speed	= 100 mph
UBC Exposure type	= B
UBC Importance factor, I	= 1
UBC Windward coefficient, C_q	= 0.8
UBC Leeward coefficient, C_q	= 0.5

Comparison of Results

For each of the static lateral load analyses, the story shears can be computed using the formulae given in the applicable references. For the seismic loads, the fundamental period computed by ETABS can be used in the formulae. From ETABS results, this fundamental period is 0.5204 second. (Note the difference between the calculated fundamental period for this example and Example 2, which neglects shear and axial deformations.)

Hand-calculated story shears are compared with story shears produced by the ETABS program in Table 3-1 for UBC seismic loads, Table 3-2 for IBC seismic loads and Table 3-3 for UBC wind loads. The values are identical.

Table 3-1 Comparison of Results for Story Shears-UBC 1997 Seismic

Level	ETABS (kips)	Theoretical (kips)
Roof	34.09	34.09
2 nd	56.82	56.82
1 st	68.19	68.19

Table 3-2 Comparison of Results for Story Shears-IBC 2000 Seismic

Level	ETABS (kips)	Theoretical (kips)
Roof	19.38	19.38
2 nd	32.25	32.25
1 st	38.64	38.64

Figure 3-3 Comparison of Results for Story Shears-UBC 1997 Wind

Level	ETABS (kips)	Theoretical (kips)
Roof	3.30	3.30
2 nd	9.49	9.49
1 st	15.21	15.21

References

International Code Council, Inc. 2000. *International Building Code*. Falls Church, Virginia.

International Conference of Building Officials 1997. *Uniform Building Code*. Whittier, California.

Single-Story, Three-Dimensional Frame Dynamic Response Spectrum Analysis

Description

This is a one-story, four-bay, three-dimensional frame. The frame is subjected to the El Centro 1940 N-S component seismic response spectrum, for 5 percent damping, in two orthogonal directions.

The frame geometry is shown in Figure 4-1.

Significant Options of ETABS Activated

- Three-dimensional frame analysis
- Automatic story mass calculation
- Dynamic response spectrum analysis

Comparison of Results

The example is a three-degree-of-freedom system. From the individual column lateral stiffnesses, assuming rigid beams and rigid offsets at column top ends equal to 36 inches (i.e., the depth of the beams) and neglecting both shear deformations and column axial deformations, the structural stiffness matrix can be assembled (Przemieniecki 1968).

From the stiffness and mass matrices of the system, the three natural periods and mass normalized mode shapes of the system can be obtained (Paz 1985). These are compared in Table 4-1 with ETABS results. The results are identical.

Table 4-1 Comparison of Results for Periods and Mode Shapes

Mode	Quantity	ETABS	Theoretical
1	Period, sec.	0.1389	0.1389
	Mode Shape		
	X-translation	-1.6244	-1.6244
	Y-translation	0.0000	0.000
	Z-rotation	0.0032	0.0032
2	Period, sec.	0.1254	0.1254
	Mode Shape		
	X-translation	0.000	0.000
	Y-translation	1.6918	1.6918
	Z-rotation	0.000	0.000
3	Period,sec.	0.0702	0.070
	Mode Shape		
	X-translation	0.4728	0.4728
	Y-translation	0.000	0.000
	Z-rotation	0.0111	0.0111

References

Paz, M 1985. *Structural Dynamics, Theory and Computations*. Van Nostrand Reinhold.

Przemieniecki, J.S. 1968. *Theory of Matrix Structural Analysis*. McGraw-Hill.

Three-Story Three-Dimensional Braced Frame Dynamic Response Spectrum Analysis

Description

This is an L-shaped building structure with four identical braced frames. All members (columns and braces) carry only axial loads.

The structure is subject to the El Centro 1940 N-S component seismic response spectrum in the X-direction. The structural damping is 5 percent. The geometry of the structure and a typical frame are shown in Figure 5-1.

Significant Options of ETABS Activated

- Three-dimensional structure analysis using planar frames
- Brace (diagonal) and column members with no bending stiffness
- Dynamic response spectrum analysis

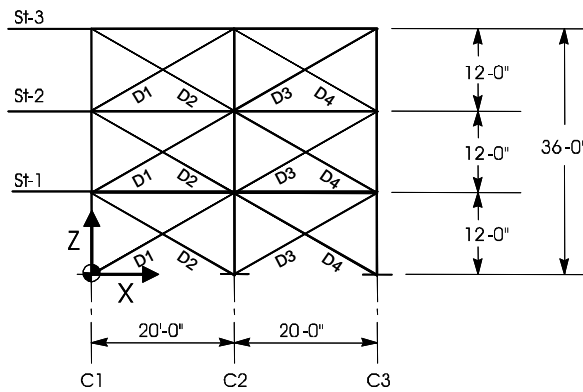
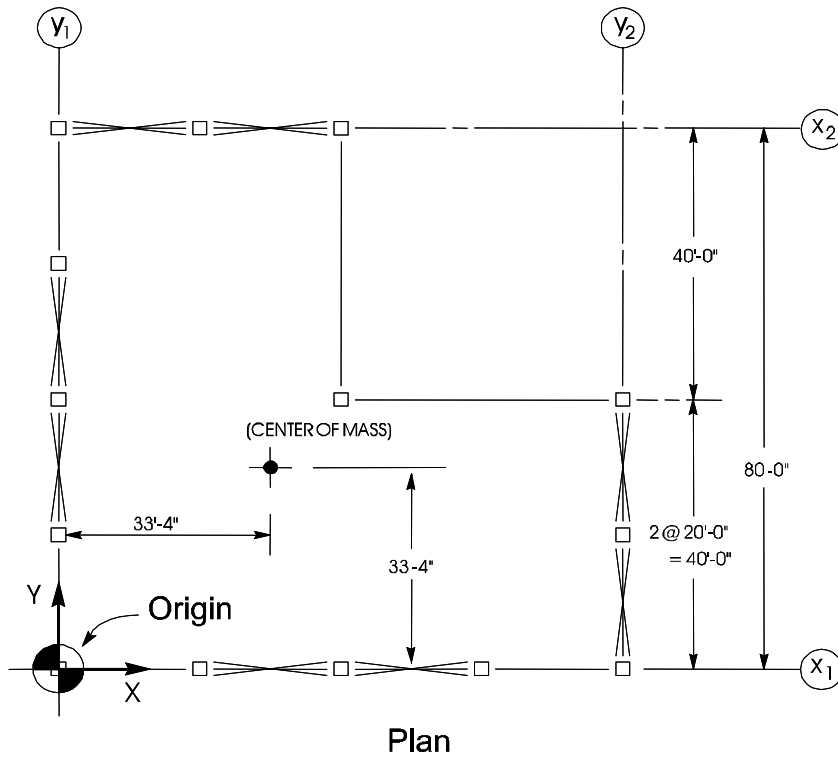


Figure 5-1 Three-Story, Three-Dimensional Braced Frame Building

Computer Model

The structure is modeled by appropriately placing four identical planar frames. Each frame is modeled using three column lines. Kip-inch-second units are used. The modulus of elasticity is taken as 29500 ksi and the typical member axial area as 6 in². A story mass of 1.242 kip-sec²/in and a mass moment of inertia of 174907.4 kip-sec²-in are used.

The input data file is EX5.EDB, and is available on the CD.

Comparison of Results

This example has been solved in Wilson and Habibullah (1992) and Peterson (1981). A comparison of ETABS results for natural periods and key member forces for one frame with these references is given in Table 5-1. The agreement is excellent.

Table 5-1 Comparison of Results

Quantity	ETABS	Wilson and Habibullah	Peterson
Period, Mode 1	0.32686	0.32689	0.32689
Period, Mode 2	0.32061	0.32064	0.32064
Axial Force Column C1, Story 1	279.39	279.47	279.48
Axial Force Brace D1, Story 1	194.44	194.51	194.50
Axial Force Brace D3, Story 1	120.49	120.53	120.52

References

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

Wilson, E.L. and A. Habibullah 1992. *SAP90, Sample Example and Verification Manual*. Computers and Structures, Inc. Berkeley, California.

Nine-Story, Ten-Bay Plane Frame Eigenvalue Analysis

Description

This is a nine-story, ten-bay plane frame, as shown in Figure 6-1.

An eigenvalue analysis is made.

Significant Options of ETABS Activated

- Two-dimensional frame analysis
- Eigenvalue analysis

Computer Model

The frame is modeled with eleven column lines and ten bays. Kip-ft-second units are used.

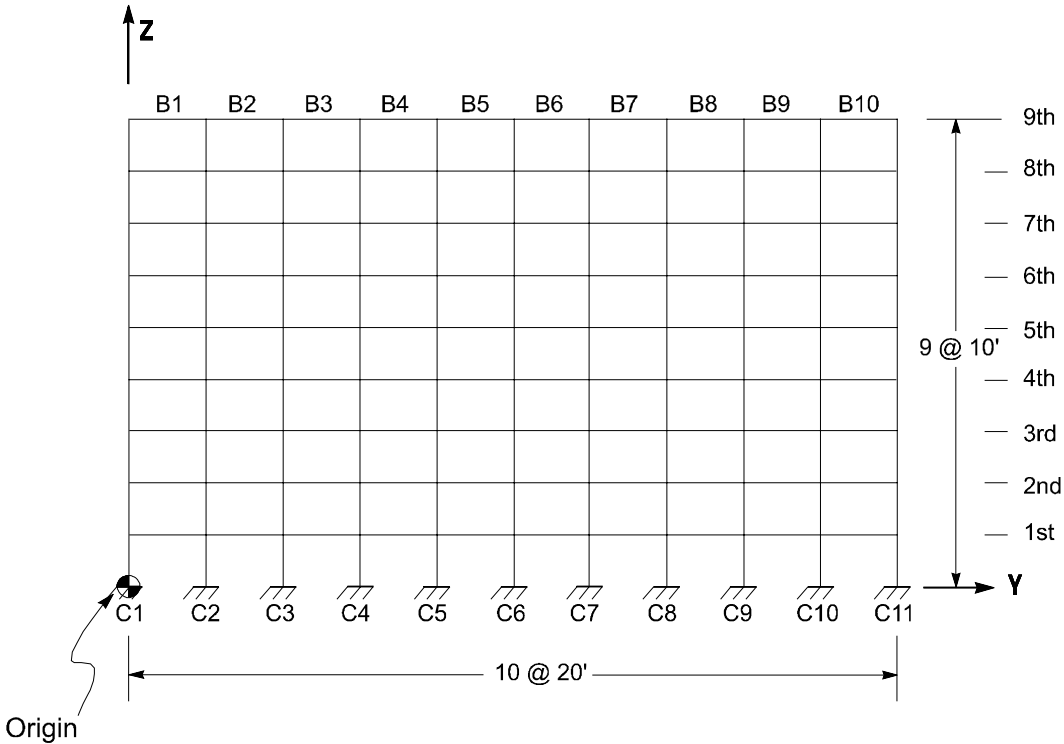


Figure 6-1 Nine-Story, Ten-Bay Plane Frame

A modulus of elasticity of 432000 ksf is used. A typical member axial area of 3ft^2 and moment of inertia of 1ft^4 are used. A mass of $3\text{kip}\cdot\text{sec}^2/\text{ft}/\text{ft}$ of member length is converted to story mass using tributary lengths and used for the analysis.

The input data filename for this example is EX6.EDB. This file is available on the CD.

Comparison of Results

This example is also analyzed in Wilson and Habibullah (1992) and Bathe and Wilson (1972). There are two differences between the ETABS analysis and the analyses of the references. The models of the references assign vertical and horizontal mass degrees of freedom to each joint in the structure. However, the ETABS model only assigns horizontal masses and additionally, only one horizontal mass is assigned for all the joints associated with any one floor level.

The eigenvalues obtained from ETABS are compared in Table 6-1 with results from Wilson and Habibullah (1992) and Bathe and Wilson (1972). Considering the differences in modeling enumerated herein, the comparison is excellent.

Table 6-1 Comparison of Results for Eigenvalues

Quantity	ETABS	Wilson and Habibullah	Bathe and Wilson
1	0.58964	0.58954	0.58954
2	5.53195	5.52696	5.52695
3	16.5962	16.5879	16.5878

References

- Bathe, K.J. and E.L. Wilson 1972. Large Eigenvalue Problems in Dynamic Analysis. Journal of the Eng. Mech. Div. ASCE. Vol. 98, No. EM6, Proc. Paper 9433. December.
- Wilson, E.L. and A. Habibullah 1992. *SAP90, Sample Example and Verification Manual*, Computers and Structures, Inc. Berkeley, California.

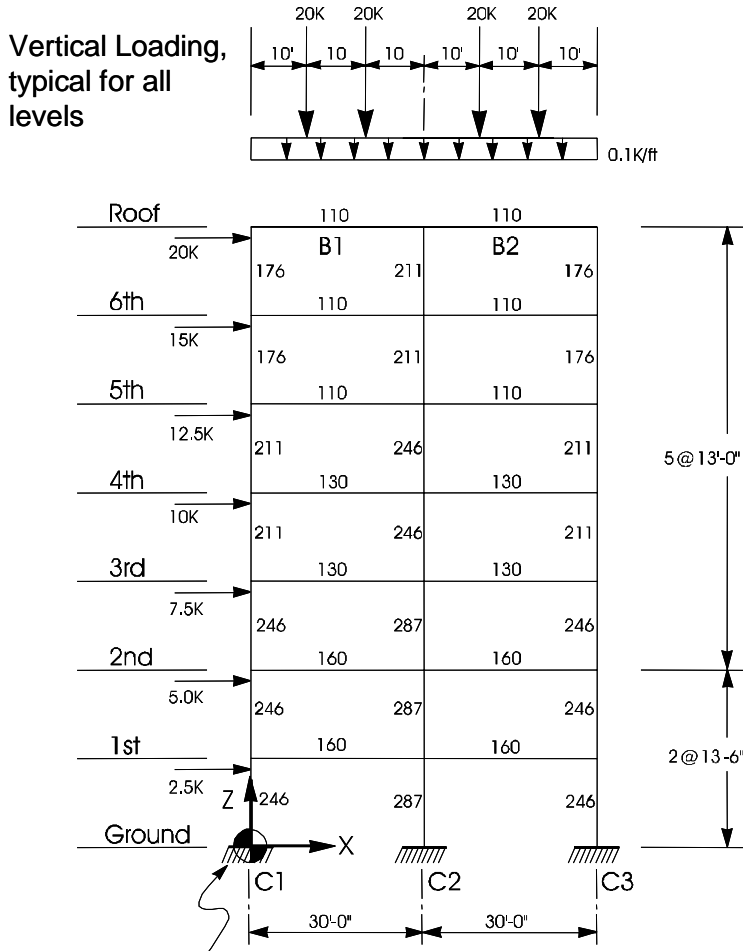
Seven-Story Plane Frame, Gravity and Lateral Loads Analysis

Description

This is a seven-story plane frame. The gravity loads and the geometry of the frame are shown in Figure 7-1.

The frame is subjected to the following lateral loads:

- Static lateral loads, shown in Figure 7-1
- Lateral loads resulting from the El Centro 1940 N-S component seismic response spectra, 5 percent damping
- Lateral loads resulting from the El Centro 1940 N-S component acceleration time history



All columns are W14s
 All beams are W24s
 Member weights are indicated
 Typical story mass = 0.49 kip-sec²/in

Figure 7-1 Seven-Story Plane Frame

Significant Options of ETABS Activated

- ❑ Two-dimensional frame analysis
- ❑ User-specified section properties
- ❑ User-specified lateral loads
- ❑ Dynamic response spectrum analysis
- ❑ Dynamic time history analysis

Computer Model

The frame is modeled with three column lines and two bays. Kip-inch-second units are used. Because the wide flange members used in the frame are older sections, their properties are not available in the AISC section property database included with the ETABS program, and the required properties therefore need to be explicitly provided in the input data.

The input data file is EX7.EDB. The input history is ELCN-THU. Time history results are obtained for the first eight seconds of the excitation. This is consistent with DYNAMIC/EASE2, with which the results are compared.

Comparison of Results

The example frame is analyzed in Wilson and Habibullah (1992) for gravity loads, static lateral loads and dynamic response spectrum loads. DYNAMIC/EASE2 analyzes the example frame under static lateral loads and dynamic response spectrum and time history loads. A comparison of key ETABS results with Wilson and Habibullah (1992) and DYNAMIC/EASE2 results is presented in Tables 7-1, 7-2, 7-3 and 7-4. Note the difference in modal combination techniques between ETABS and Wilson and Habibullah, which uses complete quadratic combination (CQC), and DYNAMIC/EASE2, which uses square root of the sum of the squares combination (SRSS). The comparison of the results for all three analyses is excellent.

Table 7-1 Comparison of Results for Static Lateral Loads

Quantity	ETABS	Wilson and Habibullah	DYNAMIC/EASE2
Lateral Displacement at Roof	1.4508	1.4508	1.4508
Axial Force Column C1, at ground	69.99	69.99	69.99
Moment Column C1, at ground	2324.68	2324.68	2324.68

Table 7-2 Comparison of Results for Periods of Vibration

Mode	ETABS	Wilson and Habibullah	DYNAMIC/EASE2
1	1.27321	1.27321	1.27321
2	0.43128	0.43128	0.43128
3	0.24204	0.24204	0.24204
4	0.16018	0.16018	0.16018
5	0.11899	0.11899	0.11899
6	0.09506	0.09506	0.09506
7	0.07951	0.07951	0.07951

Table 7-3 Comparison of Results for Response Spectrum Analysis

Quantity	ETABS CQC Combination	Wilson and Habibullah CQC Combination	DYNAMIC/EASE2 SRSS Combination
Lateral Displacement at Roof	5.4314	5.4314	5.4378
Axial Force Column C1 at ground	261.52	261.50	261.76
Moment Column C1 at ground	9916.12	9916.11	9868.25

Table 7-4 Comparison of Results for Time History Analysis

Quantity	ETABS	Wilson and Habibullah
Maximum Roof Displacement	5.49	5.48
Maximum Base Shear	285	284
Maximum Axial Force, Column C1 at ground	263	258
Maximum Moment, Column C1 at ground	9104	8740

References

DYNAMIC/EASE2. *Static and Dynamic Analysis of Multistory Frame Structures Using DYNAMIC/EASE2*, Engineering Analysis Corporation and Computers and Structures, Inc. Berkeley, California.

Wilson, E.L. and A. Habibullah 1992. *SAP90, Sample Example and Verification Manual*. Computers and Structures, Inc. Berkeley, California.

Two-Story, Three-Dimensional Frame Dynamic Response Spectrum Analysis

Description

This is a two-story, three-dimensional building frame subjected to a response spectrum of constant amplitude.

The geometry of the structure is shown in Figure 8-1.

Significant Options of ETABS Activated

- Three-dimensional frame analysis
- User-specified section properties
- Dynamic response spectrum analysis

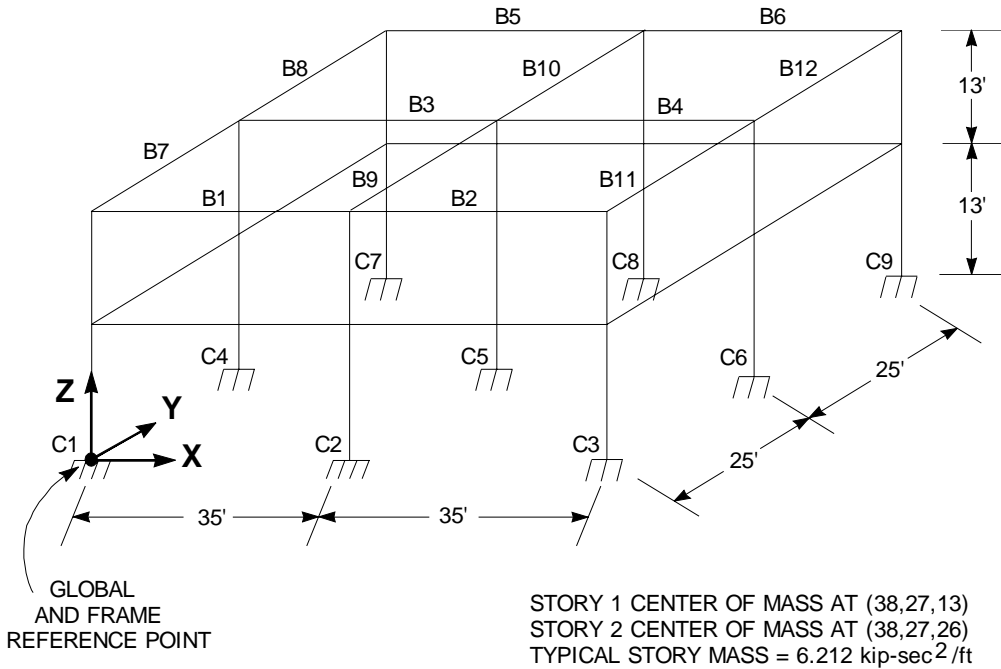


Figure 8-1 Two-Story Three-Dimensional Frame

Computer Model

The three-dimensional structure is modeled as a single frame with nine column lines and twelve bays. Kip-foot-second units are used.

For consistency with the models documented in other computer programs with which the ETABS results are compared (see Table 8-1), no story mass moments of inertia are assigned in the ETABS model. A response spectrum with a constant value of 0.4g is used. Other parameters associated with the structure are as follows:

	Columns	Beams
Axial area	4 ft ²	5 ft ²
Minor moment of inertia	1.25 ft ⁴	1.67 ft ⁴
Major moment of inertia	1.25 ft ⁴	2.61 ft ⁴
Modulus of elasticity	350000 ksf	500000 ksf

The input data file is EX8.EDB. This file is available on the CD.

Comparison of Results

This example is also analyzed in Wilson and Habibullah (1992) and Peterson (1981). A comparison of the key ETABS results with Wilson and Habibullah and Peterson is shown in Table 8-1. The agreement is excellent.

Table 8-1 Comparison of Results

Quantity	ETABS	Reference 1	Reference 2
Period, Mode 1	0.22708	0.22706	0.22706
Period, Mode 2	0.21565	0.21563	0.21563
Period, Mode 3	0.07335	0.07335	0.07335
Period, Mode 4	0.07201	0.07201	0.07201
X-Displacement Center of mass, 2 nd Story	0.0201	0.0201	0.0201

References

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

Wilson, E.L. and A. Habibullah 1992. *SAP90, Sample Example and Verification Manual*, Computers and Structures, Inc. Berkeley, California.

Two-Story, Three-Dimensional Unsymmetrical Building Frame Dynamic Response Spectrum Analysis

Description

This is a two-story three-dimensional unsymmetrical building frame. The structure is subjected to a seismic response spectrum along two horizontal axes that are at a 30-degree angle to the building axes.

The geometry of the structure is shown in Figure 9-1.

Significant Options of ETABS Activated

- Three-dimensional frame analysis
- Dynamic response spectrum analysis

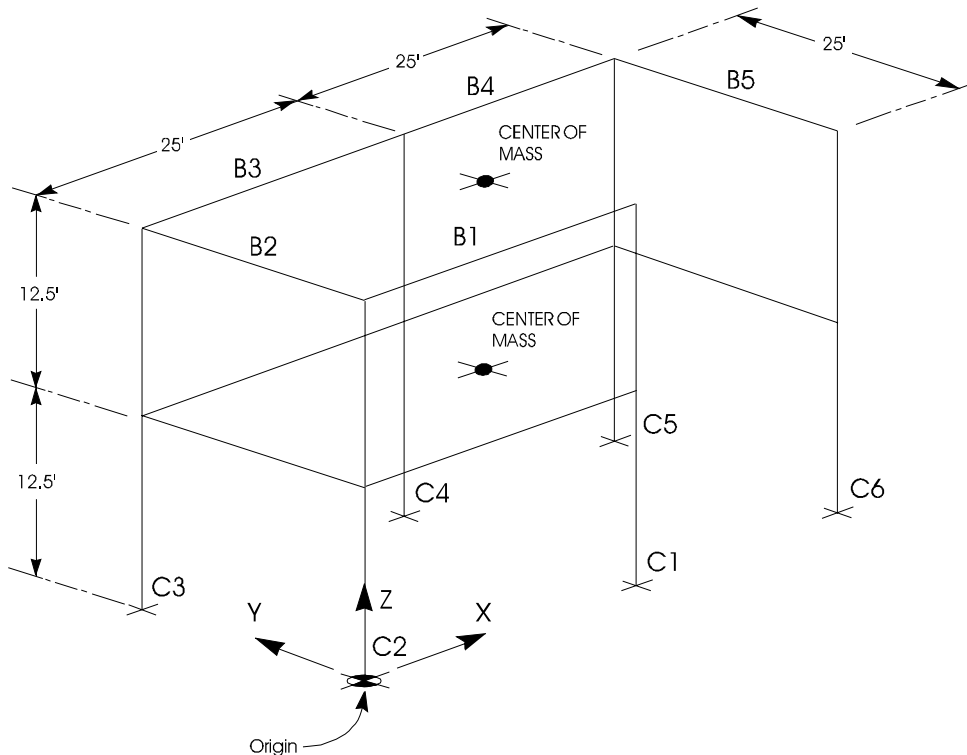


Figure 9-1 Two-Story Three-Dimensional Unsymmetrical Building Frame

Computer Model

The three-dimensional structure is modeled as a single frame with six column lines and five bays. Kip-foot-second units are used. The seismic excitation is identical to the one used in Wilson and Habibullah (1992). Typical columns are 18"x18" and beams are 12"x24". The modulus of elasticity is taken as 432000ksf.

The input data file is EX9.EDB. This file is available on the CD.

Comparison of Results

The structure is also analyzed in Wilson and Habibullah (1992). Key ETABS results are compared in Table 9-1. The comparison is excellent.

Table 9-1 Comparison of Results

Quantity	ETABS	Wilson and Habibullah
Period, Mode 1	0.4146	0.4146
Period, Mode 2	0.3753	0.3753
Period, Mode 3	0.2436	0.2436
Period, Mode 4	0.1148	0.1148
Period, Mode 5	0.1103	0.1103
Period, Mode 6	0.0729	0.0729
X- Displacement Center of Mass at 2 nd Story for:		
Seismic at 30° to X	0.1062	0.1062
Seismic at 120° to X	0.0617	0.0617

Reference

Wilson, E.L. and A. Habibullah 1992. *SAP90, Sample Example and Verification Manual*, Computers and Structures, Inc. Berkeley, California.

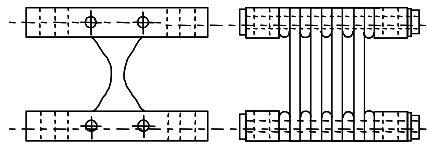
Three-Story Plane Frame with ADAS Elements Nonlinear Time History Analysis

Description

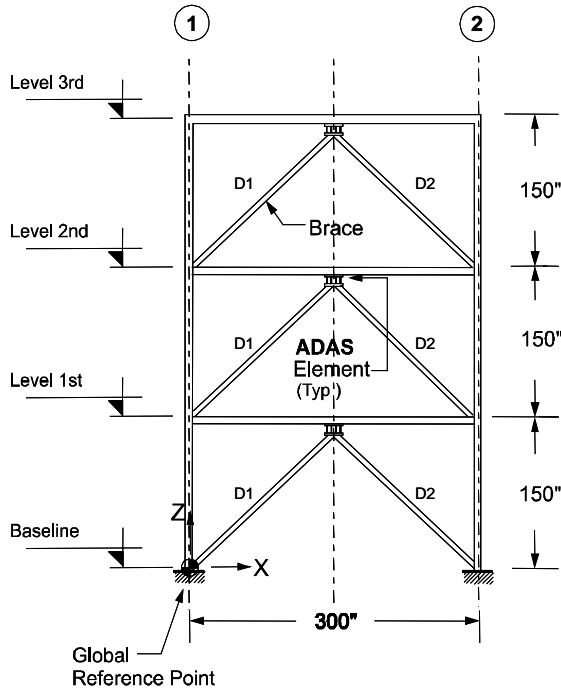
This is a single bay three-story plane frame subjected to ground motion, as shown in Figure 10-1. The El Centro 1940 (N-S) record is used in the nonlinear time history analysis. Three elements that absorb energy through hysteresis (ADAS elements as described in Scholl 1993 and Tsai, et al. 1993) are used to connect the chevron braces to the frame. Two models are investigated. In the first model, the ADAS elements are intended to produce about 5% damping in the fundamental mode. In the second model, damping is increased to 25%. The manufacturer supplied the properties of ADAS elements.

Significant Options of ETABS Activated

- Two-dimensional frame analysis
- Panel zones



Typical ADAS Element



Frame Elevation

Figure 10-1 Planar Frame with ADAS Elements

- Point assignments
- Nonlinear time history analysis
- Ritz vectors

Computer Model

The frame is modeled as a two-column line, one-bay system. Kip-inch-second units are used. The modulus of elasticity is taken as 29000 ksi. Column, beam and brace section properties are user-defined.

The ADAS elements are modeled in ETABS by assigning a panel zone with a nonlinear link property to the mid-span point object where the chevrons intersect the beams at each story. The link properties use the uniaxial hysteretic spring property (PLASTIC1) and provide beam-brace connectivity with nonlinear behavior in the U2 (shear in the 1-2 plane) direction. Under this arrangement, displacements are transferred between the chevrons and the frame via the link elements undergoing shear deformation.

A single rigid diaphragm is allocated to each story level, and connects all three point objects (two column points and one mid-span point) at each story. Because of the rigid diaphragms, no axial force will occur in the beam members. All members are assigned a rigid zone factor of 1.

In both models the value of post yield stiffness ratio is taken as 5% and the time increment for output sampling is specified as .02 second.

The input data files for this example are EX10A.EDB (5% damping) and EX10B.EDB (25% damping). The time history file is ELCN-THE. These files are available on the CD.

Comparison of Results

Sample results are compared in Table 10-1 with results from the nonlinear analysis program DRAIN-2DX (Prakash, et al. 1993) for both 5% and 25% damping cases. The agreement is good to excellent.

Table 10-1 Results Comparison

Level	5% Damping		25% Damping	
	ETABS	DRAIN-2DX	ETABS	DRAIN-2DX
Comparison of Maximum Story Deflections				
3 rd	4.57	4.57	2.10	1.92
2 nd	3.48	3.51	1.68	1.55
1 st	1.82	1.82	0.92	0.86

Table 10-1 Results Comparison

Level	5% Damping		25% Damping	
	ETABS	DRAIN-2DX	ETABS	DRAIN-2DX
Comparison of Maximum Link Shear Force				
3 rd	7.29	7.31	17.75	17.40
2 nd	13.97	13.92	36.70	36.20
1 st	17.98	18.00	47.79	47.10
Comparison of Maximum Brace Axial Force				
3 rd	5.16	5.17	12.55	12.30
2 nd	9.88	9.84	25.95	25.60
1 st	12.71	12.70	33.79	33.28

References

- Prakash, V., G.A. Powell, and S. Campbell 1993. *DRAIN-2DX Base Program Description and User Guide*. Department of Civil Engineering. University of California. Berkeley, California.
- Scholl, Roger E. 1993. *Design Criteria for Yielding and Friction Energy Dissipaters*. Proceedings of ATC-17-1 Seminar on Seismic Isolation, Passive Energy Dissipation, and Active Control. San Francisco, California. Vol. 2, 485-495. Applied Technology Council. Redwood City, California.
- Tsai, K.H., H.W. Chen, C.P. Hong, and Y.F. Su 1993. Design of Steel Triangular Plate Energy Absorbers for Seismic-Resistant Construction. *Earthquake Spectra*. Vol. 9, Number 3, 505-528.

Three-Story Plane Frame with Viscous Damper Elements, Nonlinear Time History Analysis

Description

This is a single-bay, three-story plane frame subjected to ground motion, as shown in Figure 11-1. The El Centro 1940 (N-S) record is used in the nonlinear time history analysis. Three viscous damper elements of the type described in Hanson (1993) are used to connect the chevron braces to the frame. Two models are investigated. In the first model, the damper elements are intended to produce about 5% damping in the fundamental mode. In the second model, damping is increased to 25%.

The ETABS viscous damper element (DAMPER) is a uniaxial damping device with a linear or nonlinear force-velocity relationship given by $F = CV^\alpha$.

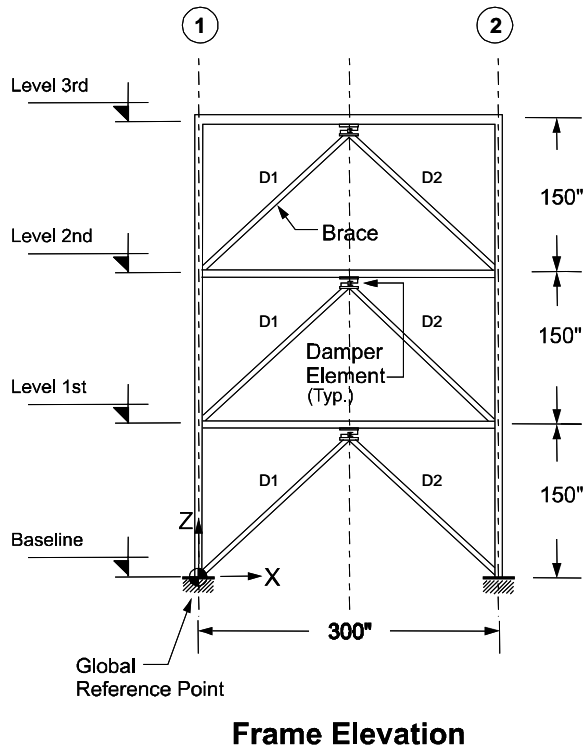


Figure 11-1 Planar Frame with Damper Elements

Significant Options of ETABS Activated

- Two-dimensional frame analysis
- Use of panel zones
- Use of uniaxial damper elements
- Point assignments
- Nonlinear time history analysis
- Ritz vectors

Computer Model

The frame is modeled as a two-column line, one-bay system. Kip-inch-second units are used. The modulus of elasticity is taken as 29000 ksi. Column, beam and brace section properties are user-defined.

The damper elements are modeled in ETABS by assigning a panel zone with a nonlinear link property to the mid-span point object where the chevrons intersect the beams at each story. The link properties use the uniaxial damper property (DAMPER), and provide beam-brace connectivity with nonlinear behavior in the U2 (shear in the 1-2 plane) direction. Under this arrangement, displacements are transferred between the chevrons and the frame via the link elements (dampers) undergoing shear deformation.

A single rigid diaphragm is allocated to each story level and connects all three point objects (two column points and one mid-span point) at each story. Because of the rigid diaphragms, no axial force will occur in the beam members. All members are assigned a rigid zone factor of 1.

The time increment for output sampling is specified as .02 second.

The input data files for this example are EX11A.EDB (5% damping) and EX11B.EDB (25% damping). The time history file is ELCN-THE. These files are available to the user on the CD.

Comparison of Results

Sample results for $\alpha = 1$ are compared in Table 11-1 with results from the nonlinear analysis program DRAIN-2DX (Prakash, et al. 1993) for both 5% and 25% damping cases. The agreement is excellent.

Table 11-1 Results Comparison

Level	5% Damping		25% Damping	
	ETABS	DRAIN-2DX	ETABS	DRAIN-2DX
Comparison of Maximum Story Deflections				
3 rd	4.09	4.11	2.26	2.24
2 nd	3.13	3.14	1.75	1.71
1 st	1.63	1.63	0.89	0.87

Table 11-1 Results Comparison

Level	5% Damping		25% Damping	
	ETABS	DRAIN-2DX	ETABS	DRAIN-2DX
Comparison of Maximum Link Shear Force				
3 rd	6.16	5.98	14.75	14.75
2 nd	10.79	10.80	32.82	32.84
1 st	15.15	15.02	44.90	44.97
Comparison of Maximum Brace Axial Force				
3 rd	4.36	4.23	10.43	10.43
2 nd	7.63	7.63	23.21	23.22
1 st	10.71	10.62	31.75	31.80

References

- Hanson, R.D. 1993. *Supplemental Damping for Improved Seismic Performance*. Earthquake Spectra, Vol. 9, Number 3, 319-334.
- Prakash, V., G.A. Powell and S. Campbell. DRAIN-2DX 1993. *Base Program Description and User Guide*. Department of Civil Engineering. University of California., Berkeley, California.

Pounding of Two Planar Frames, Nonlinear Time History Analysis

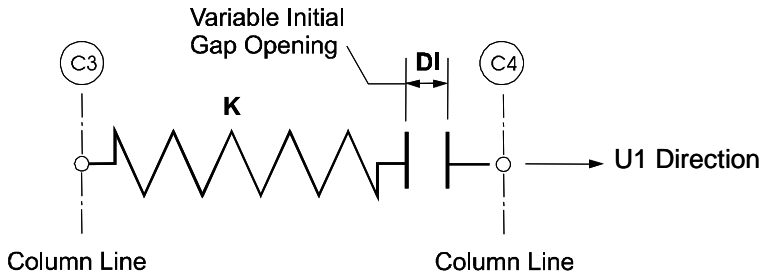
Description

A two-bay seven-story plane frame is linked to a one-bay four-story plane frame using ETABS GAP elements. The structure experiences pounding because of ground motion. The El Centro 1940 (N-S) record is used in the nonlinear time history analysis.

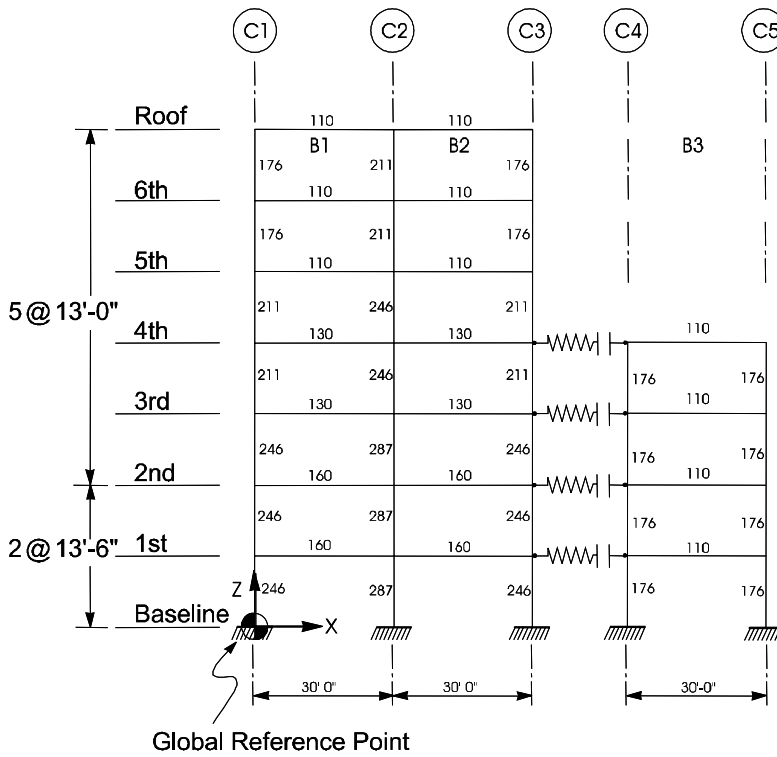
The geometry of the structure is shown in Figure 12-1.

Significant Options of ETABS Activated

- Two-dimensional frame analysis
- Use of uniaxial gap elements
- Point assignments



Gap Element Used as a Link



- All columns are W14s
- All Beams are W24s

Frame Elevation

Figure 12-1 Planar Frame with Gap Elements

- Nonlinear time history analysis
- Use of multiple diaphragms

Computer Model

This example illustrates the use of gap elements to model pounding between buildings. The combined structure is modeled as a single frame with five column lines and three beam bays. Kip-inch-second units are used. The modulus of elasticity is taken as 29500 ksi. Column and beam section properties are user-defined.

Through the joint assignment option, Column lines 4 and 5 are connected to Diaphragm 2. Column lines 1 to 3 remain connected to Diaphragm 1 by default. This arrangement physically divides the structure into two parts. The interaction is provided via the gap elements, which are used as links spanning Column lines 3 and 4. The local axis 1 of the links is in the global X-direction.

The input data for this example is EX12.EDB. The time history file is ELCN-THU. Both of the files are available to the user on the CD.

Comparison of Results

The example frame analyzed using ETABS is also analyzed using SAP2000 (Computers and Structures 2002) for time history loads (SAP2000 has been verified independently). A comparison of key ETABS results with SAP2000 is presented in Table 12-1. The agreement of the results is excellent.

Table 12-1 Comparison of Results for Time History Analysis

Quantity	ETABS	SAP2000
Maximum Lateral Displacement at Roof	5.5521	5.5521
Maximum Axial Force, Column C1 at ground	266.89	266.88

A typical output produced by the program is shown in Figure 12-2. It shows the variations of the displacement of Column lines 3 and 4 and the link force at Story 4. It is clearly evident that the link force is generated whenever the two column lines move in phase and their separation is less than the specified initial opening, or if they move towards each other out of phase. For display purposes, the link forces are scaled down by a factor of 0.01.

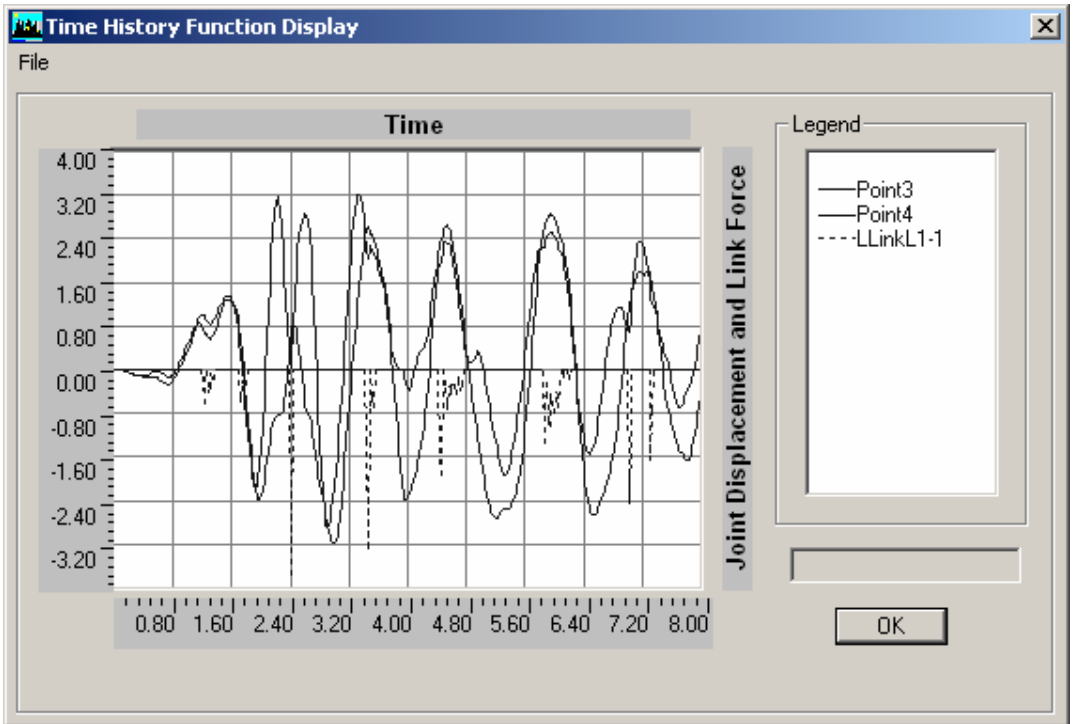


Figure 12-2 Variations of Displacement of Column Lines 3 and 4 and Link Force at Story 4

References

Computers and Structures, Inc. 2002. *SAP2000, Analysis Reference Manual, Version 8*. Computers and Structures, Inc. Berkeley, California.

Base-Isolated Two-Story 3D Frame Nonlinear Time History Analysis

Description

This is a two-story three-dimensional frame with base isolation. The structure is subjected to earthquake motion in two perpendicular directions using the Loma Prieta acceleration records. The geometry of the structure is shown in Figure 13-1.

Significant Options of ETABS Activated

- Three-dimensional frame analysis
- Use of area (floor) objects
- Use of biaxial hysteretic elements
- Point assignments
- Nonlinear time history analysis using ritz vectors

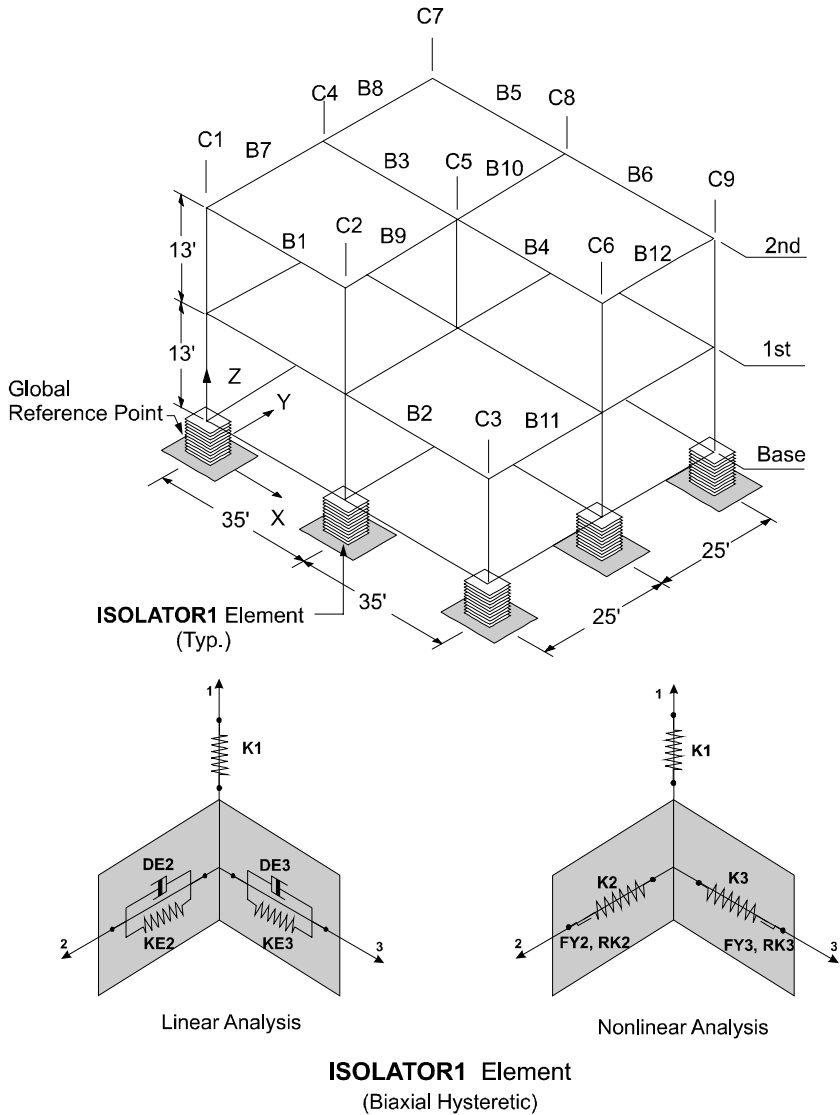


Figure 13-1 Base-Isolated Three-Dimensional Frame

Computer Model

The structure is modeled as a single reinforced concrete frame with nine column lines and twelve bays. The floor slab is taken to be 8 inches thick, covering all of the specified floor bays at the base and the 1st story level. At the sec-

ond story level the corner column as well as the two edge beams are eliminated, together with the floor slab, to render this particular level unsymmetric, as depicted in Figure 13-1.

Hysteretic base isolators of the type described in Nagarajaiah et al. (1991) are modeled using the ETABS ISOLATOR1 elements, which show biaxial hysteretic characteristics.

A modulus of elasticity of 3000 ksi is used. The self-weight of concrete is taken as 150 pcf. Kip-inch-second units are used.

The input data file for this example is EX13.EDB. The time history files are LP-TH0 and LP-TH90. All of these files are available to the user on the CD as part of the ETABS package.

Comparison of Results

The example frame analyzed using ETABS is also analyzed using SAP2000 (Computer and Structure 2002) for time history loads (SAP2000 has been verified independently). A comparison of key ETABS results with SAP2000 is presented in Table 13-1. The agreement of the results is excellent.

Table 13-1 Comparison of Results for Time History Analysis

Quantity	ETABS	SAP2000
Maximum Uy Displacement, Column C9 at 2 nd Floor	3.4735	3.4736
Maximum Axial Force, Column C1 at base	13.56	13.55

A typical output produced by the program is shown in Figure 13-2. It shows the load-deformation relationship in the major direction for a typical isolator member.

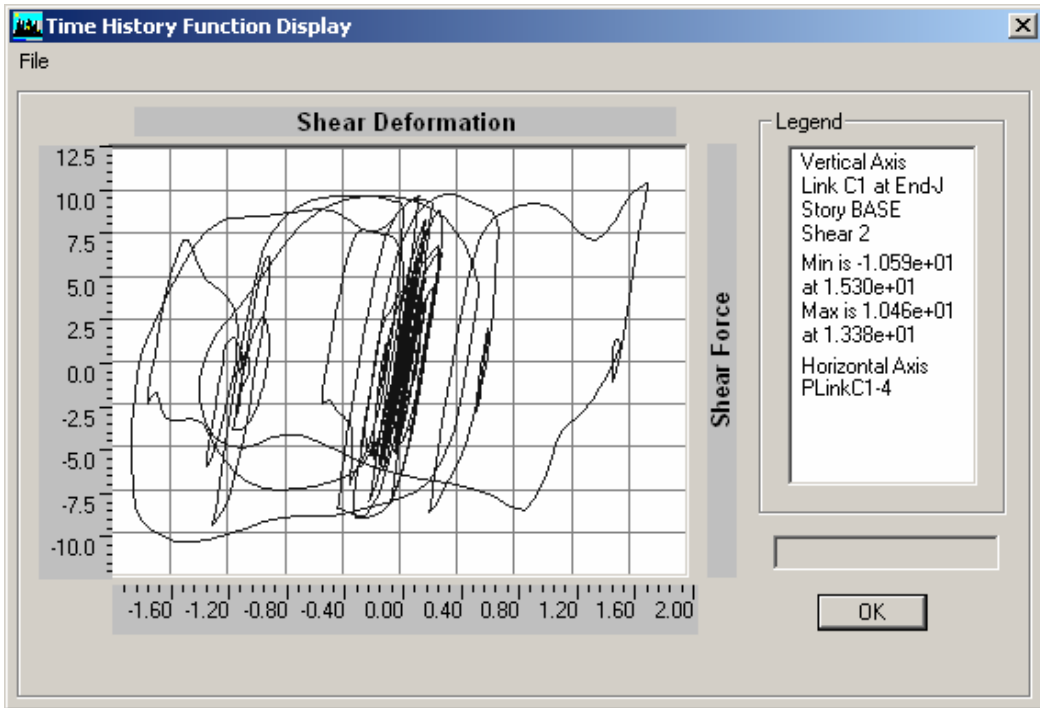


Figure 13-2 Load Deformation Diagram

References

- Computers and Structures, Inc. 2002. *SAP2000, Analysis Reference Manual, Version 8*. Computers and Structures, Inc. Berkeley, California.
- Nagarajaiah, S., A.M. Reinhorn and M.C. Constantinou 1991. *3D-Basis: Nonlinear Dynamic Analysis of Three-Dimensional Base Isolated Structures: Part II, Technical Report NCEER-91-0005*. National Center for Earthquake Engineering Research. State University of New York at Buffalo. Buffalo, New York.

Friction Pendulum Base-Isolated 3D Frame Nonlinear Time History Analysis

Description

This is a two-story three-dimensional frame with base isolation using friction pendulum base isolators. The structure is subjected to earthquake motion in two perpendicular directions using the Loma Prieta acceleration records. The geometry of the structure is shown in Figure 14-1.

Significant Options of ETABS Activated

- Three-dimensional frame analysis
- Use of area (floor) objects
- Use of biaxial friction pendulum elements
- Point assignments
- Nonlinear time history analysis using ritz vectors

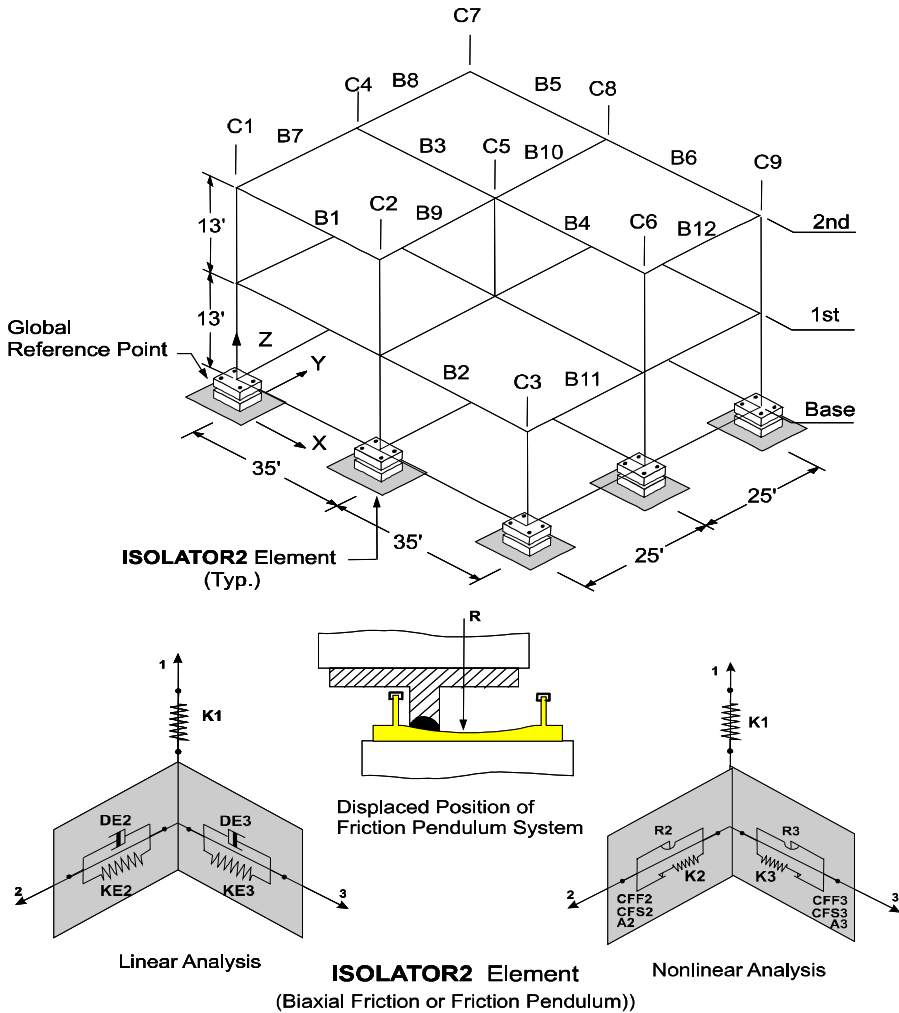


Figure 14-1 Base-Isolated Three-Dimensional Frame

Computer Model

The structure is modeled as a single reinforced concrete frame with nine column lines and twelve bays. The floor slab is taken to be 8 inches thick, covering all of the specified floor bays at the base and the 1st story level. At the second story level, the corner column and the two edge beams are eliminated, together with the floor slab, to render this particular level anti-symmetric, as depicted in Figure 14-1.

Friction pendulum type base isolators of the type described in Zayas and Low (1990) are modeled using the ETABS ISOLATOR2 elements.

The isolator properties are defined as follows:

Stiffness in direction 1	1E3
Stiffness in directions 2 and 3	1E2
Coefficient of friction at fast speed	.04
Coefficient of friction at slow speed	.03
Parameter determining the variation of the coefficient of friction with velocity	20
Radius of contact surface in directions 2 and 3	60

A modulus of elasticity of 3000 ksi is used. The self-weight of concrete is taken as 150 pcf. Kip-inch-second units are used.

It is important for these isolator elements that the axial load from other loads be modeled before starting the nonlinear analysis. This is achieved by using a factor of unity on the dead load (self weight) on the structure in the nonlinear analysis initial conditions data. The input data file for this example is EX14.EDB. The time history files are LP-TH0 and LP-TH90. All of the files are available to the user on the CD as part of the ETABS package.

Comparison of Results

The example frame analyzed using ETABS is also analyzed using SAP2000 (Computers and Structures 2002) for time history loads (SAP2000 has been verified independently). A comparison of key ETABS results with SAP2000 is presented in Table 14-1. The agreement of the results is excellent.

Table 14-1 Comparison of Result for Time History Analysis

Quantity	ETABS	SAP2000
Maximum Uy Displacement, Column C9 at 2 nd Floor	4.2077	4.2069
Maximum Axial Force, Column C1 at base	38.25	38.25

A typical output produced by the program is shown in Figure 14-2. It shows the variation of the displacement of the second story at column line 1.

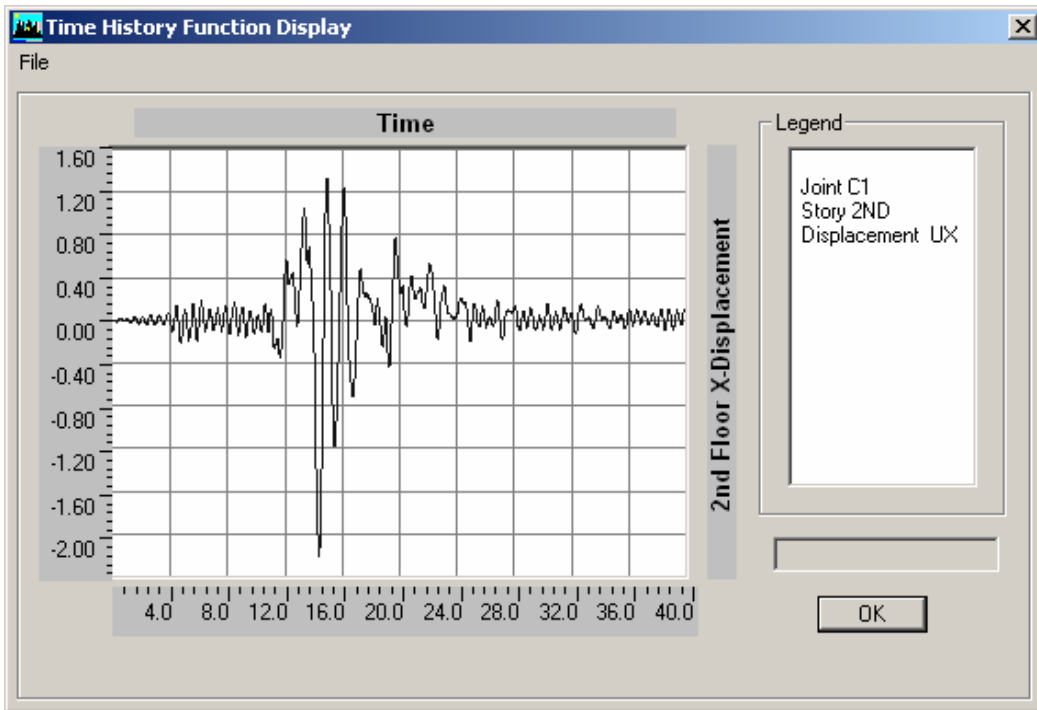


Figure 14-2 Variation of Displacement

References

- Computers and Structures, Inc. 2002. *SAP2000, Analysis Reference Manual, Version 8*. Computers and Structures, Inc. Berkeley, California.
- Zayas, V. and S. Low, 1990. *A Simple Pendulum Technique for Achieving Seismic Isolation*. Earthquake Spectra, Vol. 6, No. 2. Earthquake Engineering Research Institute. Oakland, California.

Wall Area Object Behavior Static Lateral Loads Analysis

Description

This example analyzes a series of wall configurations to evaluate the behavior of the ETABS area object with wall section assignments. All walls are subjected to a static lateral load applied at the top of the wall.

The following walls are included:

- ❑ Planar shear wall, shown in Figure 15-1
- ❑ Wall supported on columns, shown in Figure 15-2
- ❑ Wall-spandrel system, shown in Figure 15-3
- ❑ C-shaped wall section, shown in Figure 15-4
- ❑ Wall with edges thickened, shown in Figure 15-5
- ❑ E-shaped wall section, shown in Figure 15-6

Significant Options of ETABS Activated

- ❑ Use of area objects
- ❑ Two-dimensional and three-dimensional shear wall systems
- ❑ Static lateral loads analysis

Computer Model

A modulus of elasticity of 3000 ksi and a Poisson's ratio of 0.2 are used for all walls. Kip-inch-second units are used throughout. The following sections describe the models for the different walls.

Planar Shear Wall , Example 15a

This shear wall is modeled with one panel per story. Three different wall lengths of 120", 360" and 720" are analyzed. Also, one-story and three-story walls are analyzed, together with the six-story wall shown in Figure 15-1. A wall thickness of 12" is used.

The input data files for these walls are included as files EX15A1.EDB through EX15A9.EDB. These and the following input data files are available on the CD.

Wall Supported on Columns, Example 15b

This wall is modeled with two column lines. Columns are used for the first story, and the top two stories have a single area object with end piers, as shown in Figure 15-2. End piers are 40" by 12" in cross section and panels are 12" thick. Columns are 40" by 20" in cross section.

The input data for this wall is EX15B.EDB.

Wall-Spandrel System, Example 15c

This wall is modeled with four column lines. The spandrels are modeled as beams. Two different spandrel lengths of 60" and 240" are analyzed. Each wall is modeled with two area objects per story. Three-story walls are also analyzed together with the six-story wall shown in Figure 15-3. A wall and spandrel thickness of 12" is used.

The input data files for this wall are 15C1.EDB through 15C4.EDB.

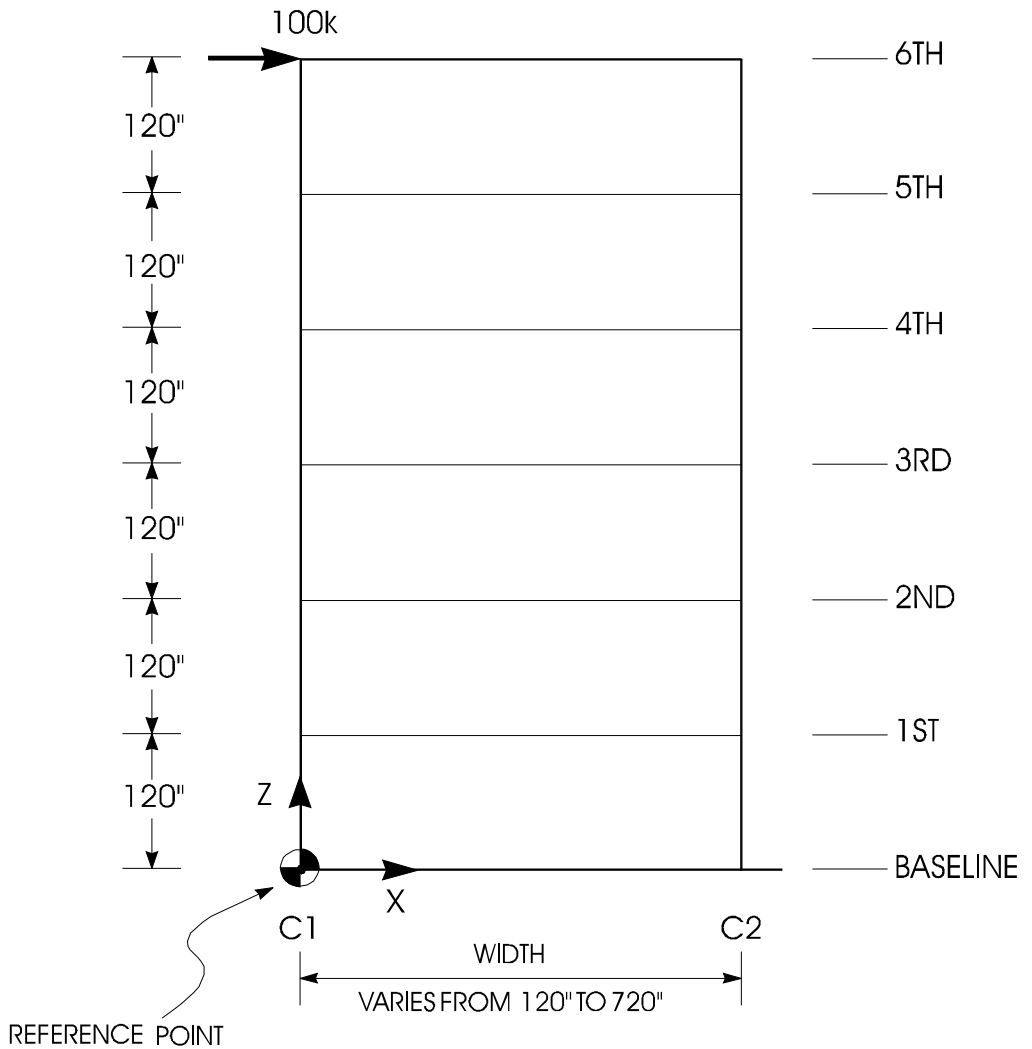
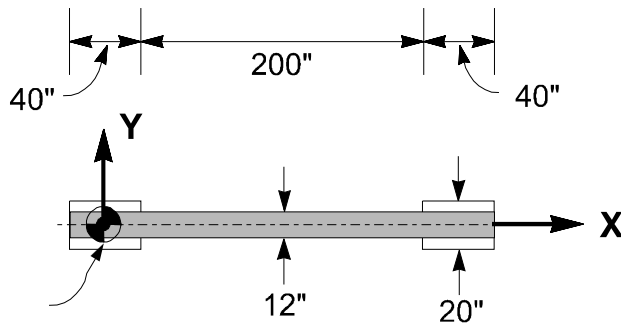
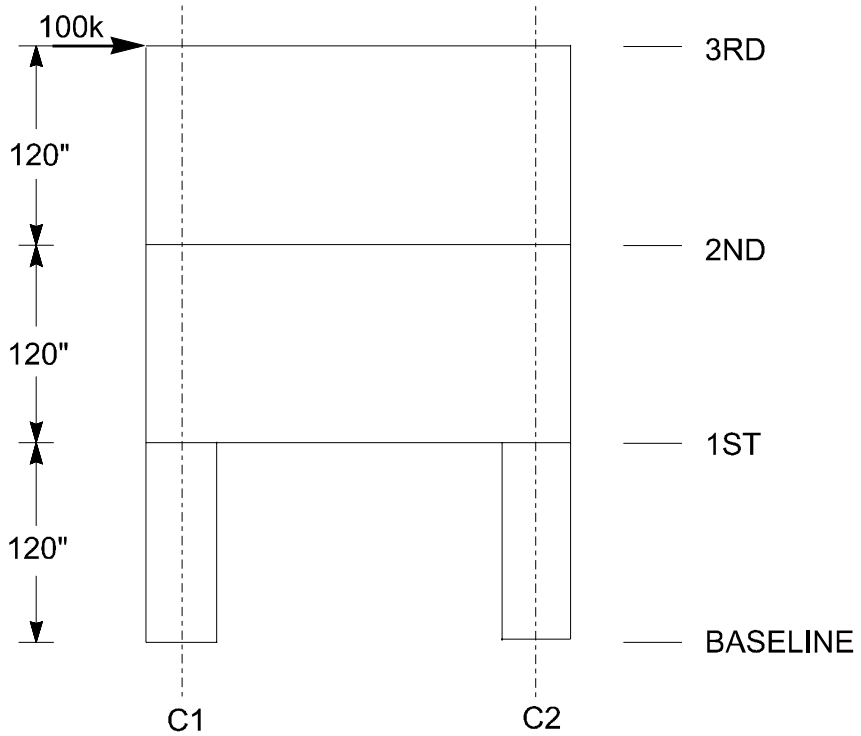


Figure 15-1 Planar Shear Wall, Example 15a



GLOBAL REFERENCE POINT

Figure 15-2 Wall Supported on Columns, Example 15b

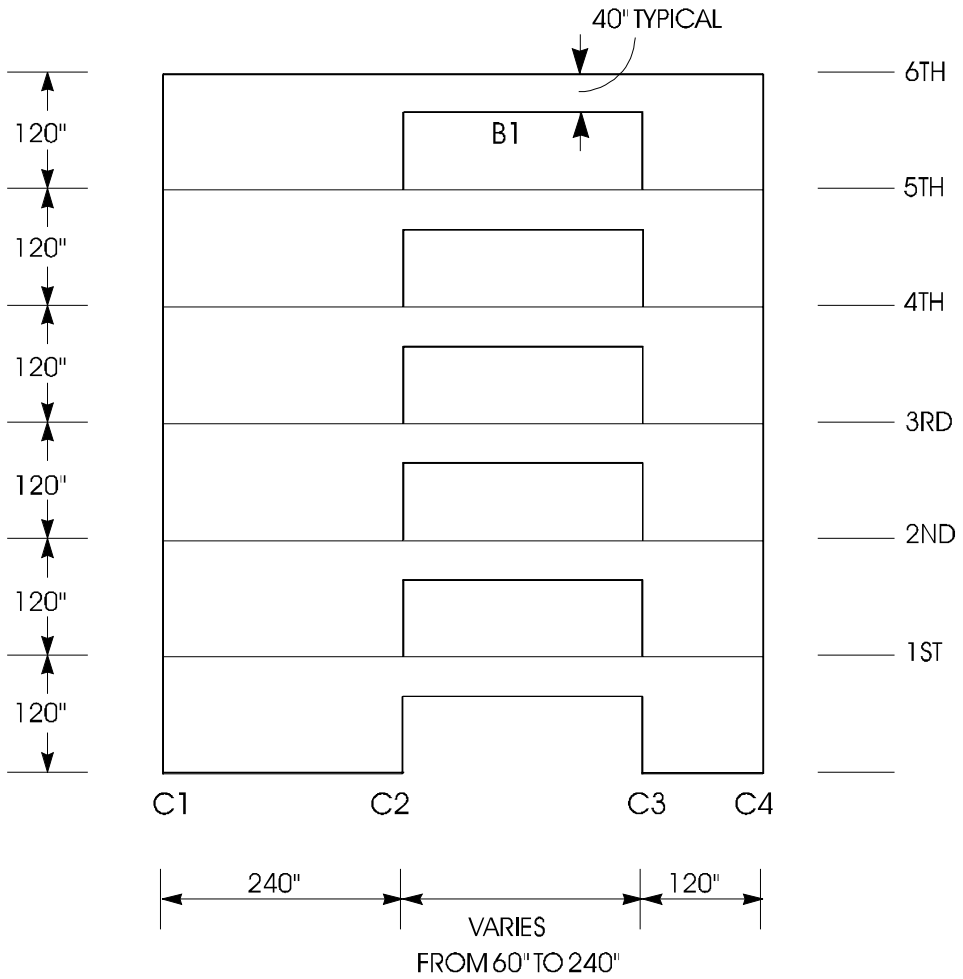


Figure 15-3 Wall-Spandrel System, Example 15c

C-Shaped Wall Section, Example 15d

This wall is modeled with six column lines and five area objects per story, to model the shape of the wall. A three-story wall was also analyzed together with the six-story wall, as shown in Figure 15-4. A wall thickness of 6" is used.

The input data files for these walls are included as files EX15D1.EDB and EX15D2.EDB.

Wall with Edges Thickened, Example 15e

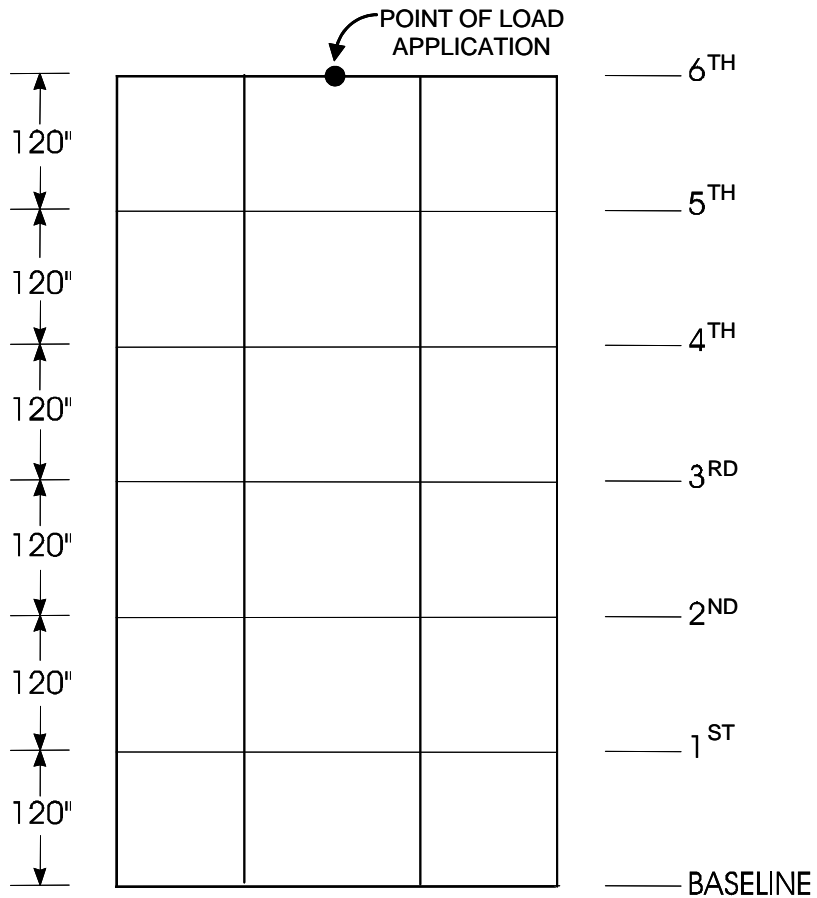
This wall is modeled with two column lines and one area object, with end piers, per story as shown in Figure 15-5. A three-story wall was also analyzed together with the six-story wall shown in Figure 15-5.

The input data for these walls are included as files EX15E1.EDB and EX15E2.EDB.

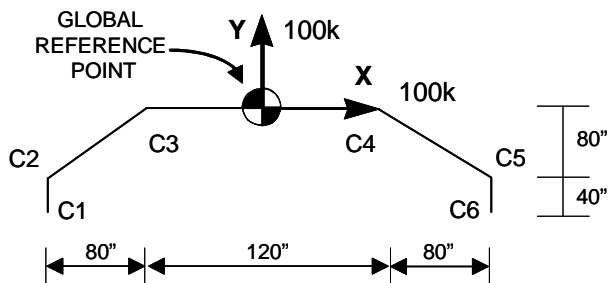
E-Shaped Wall Section, Example 15f

This wall is modeled with six column lines and five area objects per story to model the shape of the wall. A three-story wall was also analyzed together with the six-story wall, as shown in Figure 15-6. A wall thickness of 6" is used.

The input data for these walls are included as files EX15F1.EDB and EX15F2.EDB.



ELEVATION



PLAN

Figure 15-4 C-Shaped Wall Section, Example 15d

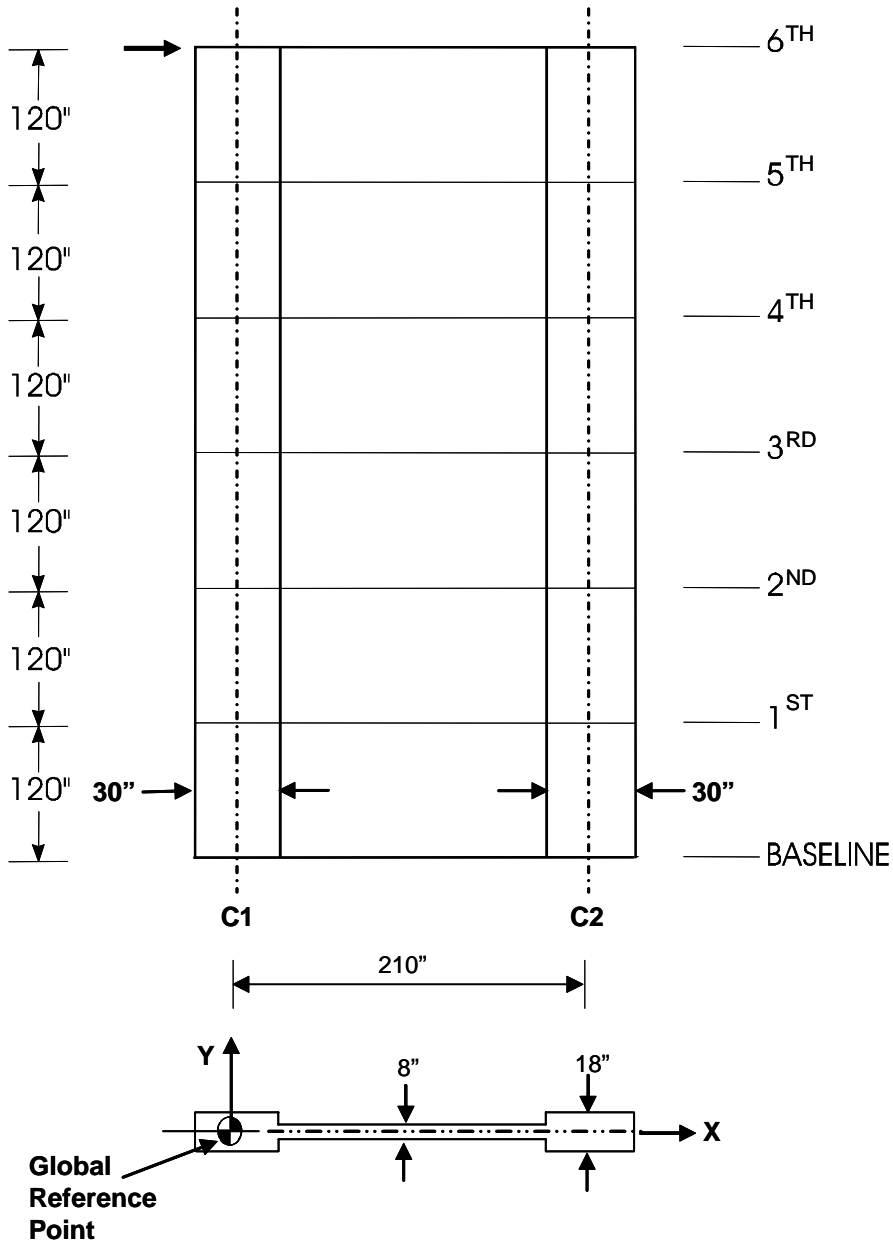


Figure 15-5 Wall with Thickened Edges, Example 15e

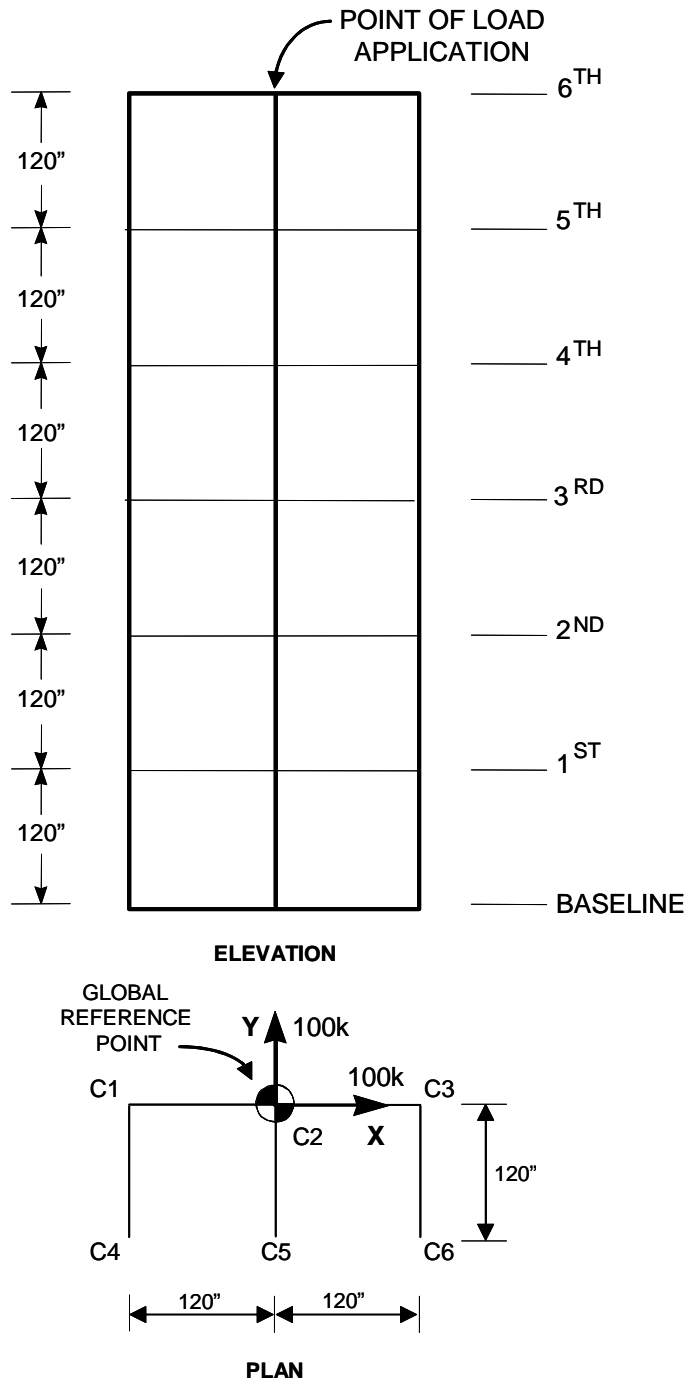


Figure 15-6 E-Shaped Wall Section, Example 15f

Comparison of Results

All walls analyzed in this example using ETABS were also analyzed using the general structural analysis program SAP2000 (Computers and Structure 2002), using refined meshes of the membrane/shell element of that program. The SAP2000 meshes used are shown in Figures 15-7, 15-8, 15-9, 15-10, 15-11 and 15-12. For the SAP2000 analysis, the rigid diaphragms at the floor levels were modeled by constraining all wall nodes at the floor to have the same lateral displacement for planar walls, or by adding rigid members in the plane of the floor for three-dimensional walls.

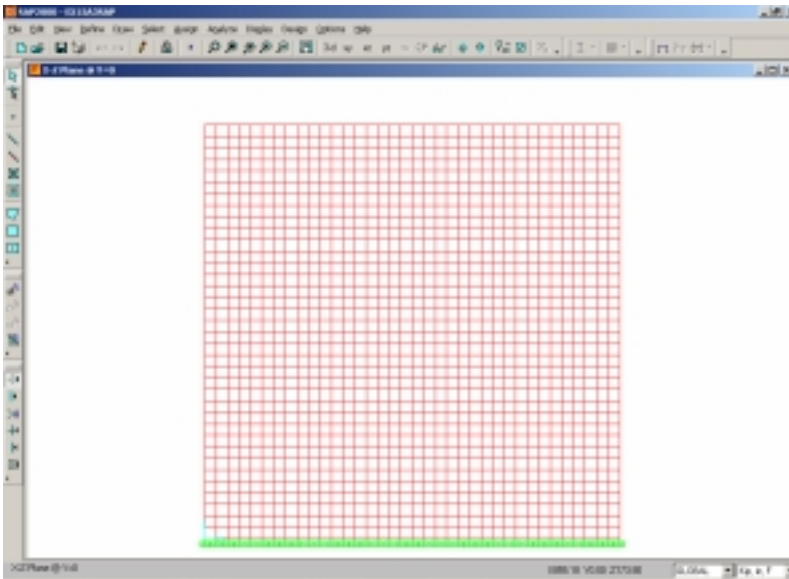


Figure 15-7 SAP2000 Mesh, Example 15a

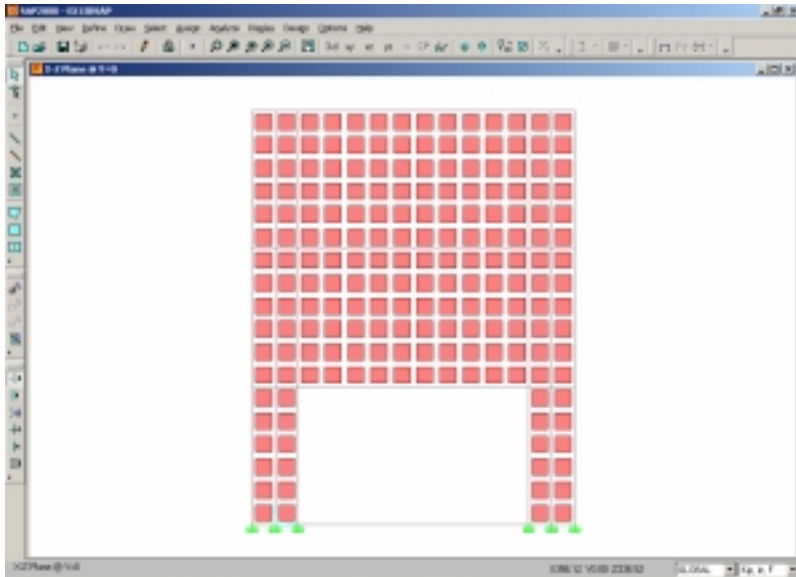


Figure 15-8 SAP2000 Mesh, Example 15b

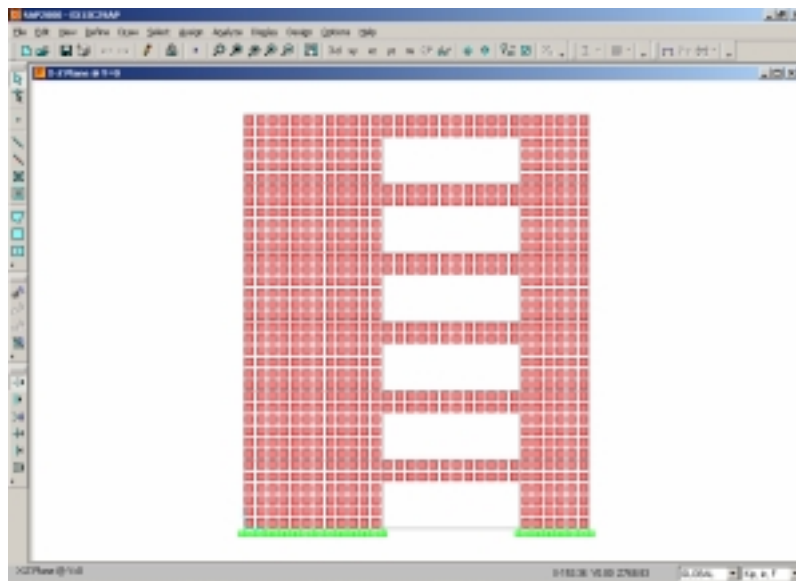


Figure 15-9 SAP2000 Mesh, Example 15c

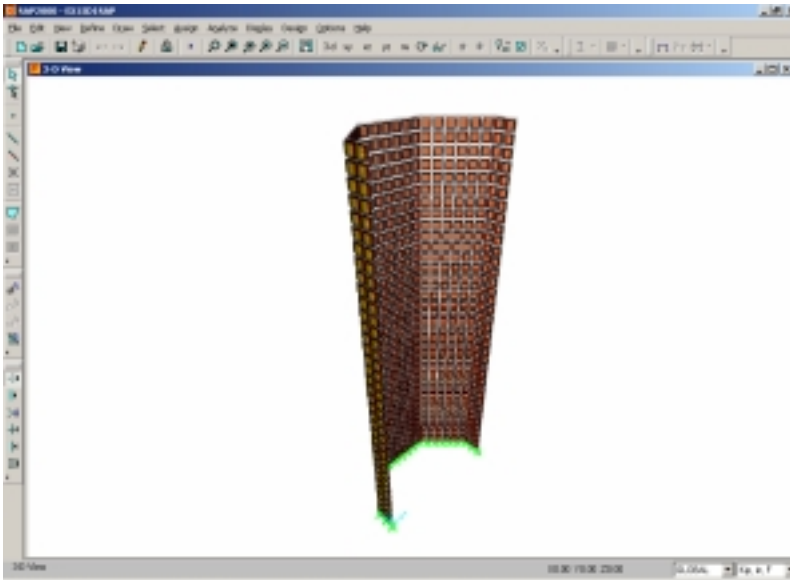


Figure 15-10 SAP2000 Mesh, Example 15d

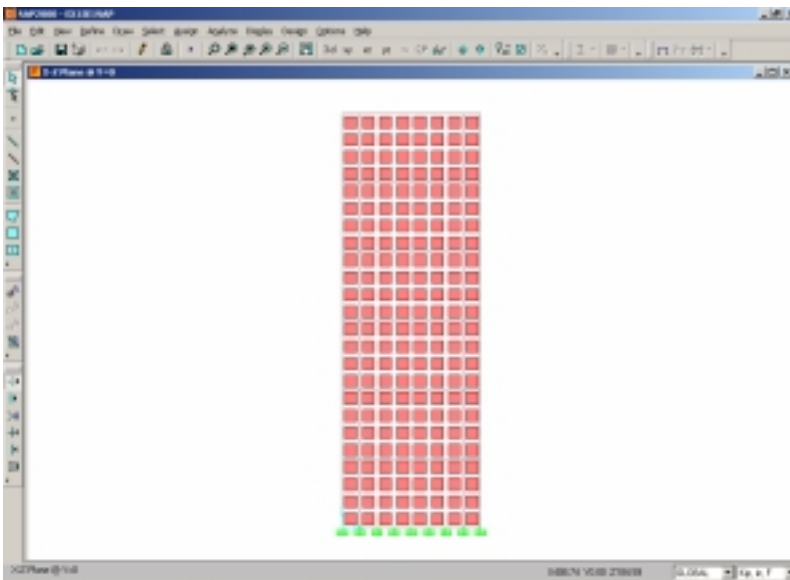


Figure 15-11 SAP2000 Mesh, Example 15e

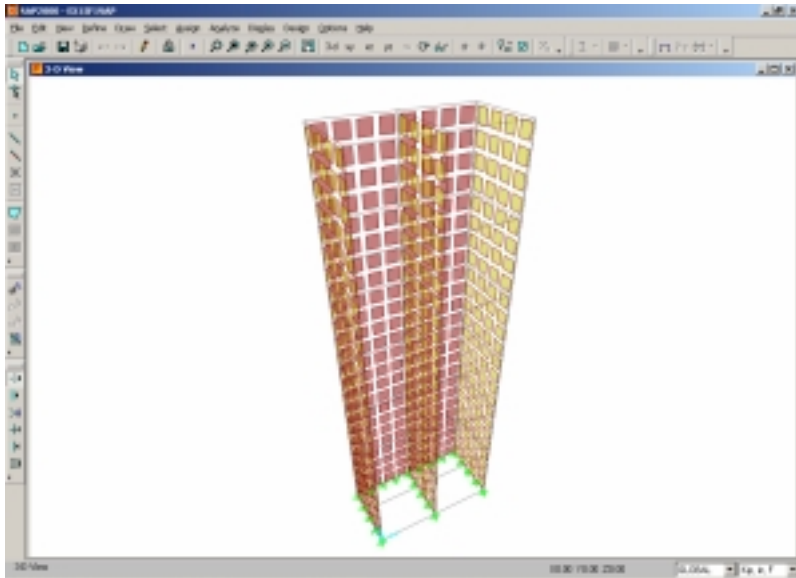


Figure 15-12 SAP2000 Mesh, Example 15f

The lateral displacements from the ETABS and SAP2000 analyses are compared in Tables 15-1, 15-2, 15-3, 15-4, 15-5 and 15-6 for the various walls. The agreement is good. In general, the comparisons become better as the number of stories is increased.

Table 15-1 Results Comparison for Top Displacements (Inches), Example 15a

Number of Stories	Wall Height (inches)	Wall Length (inches)	ETABS	SAP2000
6	720	120	2.3926	2.4287
		360	0.0985	0.1031
		720	0.0172	0.0186
3	360	120	0.3068	0.3205
		360	0.0169	0.0187
		720	0.0046	0.0052
1	120	120	0.0144	0.0185
		360	0.0024	0.0029
		720	0.0011	0.0013

Table 15-2 Results Comparison for Displacements (Inches), Example 15b

Location	ETABS	SAP2000
Story 3	0.0690	0.0671
Story 2	0.0524	0.0530
Story 1	0.0390	0.0412

**Table 15-3 Results Comparison for Top Displacements (inches)
Example 15c (1-4)**

Number of Stories	Beam Length (inches)	ETABS	SAP2000
6	60	0.0841	0.0869
	240	0.1443	0.1505
3	60	0.0186	0.0200
	240	0.0311	0.0332

Table 15-4 Results Comparison for Top Displacements (Inches) at Load Application Point, Example 15d (1-2)

Number of Stories	Load Direction	Displacement Direction	ETABS	SAP2000
6	X	X	0.8621	0.8936
	X	Z-Rotation	0.0185	0.0191
	Y	Y	1.1427	1.1882
3	X	X	0.1242	0.1337
	X	Z-Rotation	0.0024	0.0025
	Y	Y	0.1614	0.1733

Table 15-5 Results Comparison for Top Displacements (Inches), Example 15e(1-2)

Number of Stories	ETABS	SAP2000
6	0.2798	0.2899
3	0.0451	0.0480

Table 15-6 Results Comparison for Displacements at Load Application, Example 15f (1-2)

Number of Stories	Load Direction	Displacement Direction	ETABS	SAP2000
6	X	X	0.3695	0.3655
	X	Z-Rotation	0.0042	0.0039
	Y	Y	0.7286	0.7490
3	X	X	0.0596	0.0628
	X	Z-Rotation	0.0005	0.0005
	Y	Y	0.0989	0.1058

References

Computers and Structures, Inc. 2002. *SAP2000, Analysis Reference Manual, Version 8.* Computers and Structures, Inc. Berkeley, California.