SAP2000®

Integrated
Finite Element Analysis
and
Design of Structures

VERIFICATION MANUAL



Computers and Structures, Inc.
Berkeley, California, USA

Version 6.1
Revised July 1997

COPYRIGHT

The computer program SAP2000 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

tel: (510) 845-2177
fax: (510) 845-4096
e-mail: *info@csiberkeley.com*
web: *www.csiberkeley.com*

© Copyright Computers and Structures, Inc., 1978–1997.
The CSI Logo is a registered trademark of Computers and Structures, Inc.
SAP2000 is a registered trademark of Computers and Structures, Inc.
Windows is a registered trademark of Microsoft Corporation

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAP2000. 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.

Table of Contents

	Introduction	1
Example 1	Two-Dimensional Frame — Static and Dynamic Loads	3
Example 2	Bathe and Wilson Frame — Eigenvalue Problem	13
Example 3	Three-dimensional Frame — Dynamic Loads	17
Example 4	ASME Frame — Eigenvalue Problem	23
Example 5	Three-dimensional Braced Frame — Dynamic Loads	29
Example 6	Beam — Steady-State Harmonic Loads	35
Example 7	Two-dimensional Truss — Static Loads	39
Example 8	Three-dimensional Building — Dynamic Loads	43
Example 9	Patch Tests — Prescribed Displacements	47
Example 10	Straight Beam — Static Loads	51
Example 11	Curved Beam — Static Loads	59
Example 12	Twisted Beam — Static Loads	63

Example 13	Beam On Elastic Foundation — Static Loads	67
Example 14	Rectangular Plate — Static Loads	71
Example 15	Cantilever Plate — Eigenvalue Problem	75
Example 16	Scordelis-Lo Roof — Static Loads	79
Example 17	Hemispherical Shell — Static Loads	83
Example 18	Portal with P-delta	87
Example 19	Pounding of Two Planar Frames — Nonlinear Time-History Analysis	91
Example 20	Friction-Pendulum Base-Isolated 3D Frame — Nonlinear Time-History Analysis	95

Introduction

This manual presents a set of example problems solved using the SAP2000 structural analysis program. These examples demonstrate many of the capabilities of the SAP2000 program. For purposes of verification, key results from these examples are compared with theoretical or published results from other computer programs, where such are available. The verification problems cover each type of element and include both static and dynamic examples.

For each example, this manual contains:

- A short description of the problem
- A list of significant SAP2000 options activated
- A description of the input data used to create the model
- A comparison of key results with theoretical results or results from other computer programs, if available

Some examples are solved using several different elements, mesh sizes and/or boundary conditions. Key results from these different solutions are presented for comparative purposes.

The data files and selected output files associated with the example problems are provided in subdirectory EXAMPLES of the SAP2000 directory. For each example, the following files may be provided:

- The SAP2000 model file, with extension .SDB. This file can be accessed in SAP2000 using the Open command under the File menu
- The input data text file, with extension .S2K. This is an alternative to the .SDB file and can be accessed in SAP2000 using the Import command under the File menu
- Binary results files, with various extensions. If these files are provided, analysis results can be viewed in SAP2000 without running the analysis

Some of the examples have more than one model file corresponding to the use of different element types or mesh sizes in the creation of the model. The filenames associated with a particular problem are identified (without extension) in each example.

Note that not all examples can be run with every version of the program. For examples, time-history analyses require the PLUS or Nonlinear version, and the nonlinear examples require the Nonlinear version.

Example 1

Two-Dimensional Frame — Static and Dynamic Loads

Description

This is a seven-story two-dimensional frame structure, subjected to static lateral and vertical loads and dynamic lateral loads due to earthquakes. The structure is analyzed once using earthquake loads specified as a response spectrum, and once using earthquake loads specified as a base acceleration time history. A solution to this problem using another computer program is documented in Reference [1]. The framing and the static loads are shown in Figure 1-1. The earthquake excitation is shown in Figure 1-8. It is the N-S component of the 1940 El Centro earthquake. The framing and the loads are all identical to the ones used in Reference [1].

Significant Options of SAP2000 Activated

- Two-dimensional frame analysis
- Diaphragm constraints
- Lateral joint loads
- Vertical span loads
- Response-spectrum analysis
- Time-history response to base excitation

Input Data

The computer model used is shown in Figures 1-1 and 1-2. Kip-inch units are used. Two different input files are used. The first file is to analyze the structure for static vertical and lateral loads and response spectrum dynamic loads. The input data file for this analysis is FRAME. The second file is to repeat the dynamic analysis but using base acceleration time-history as a loading. The input data file for this analysis is FRAMETH and the digitized base acceleration is given in file ELCENTRO. This is shown in Figure 1-8.

The file FRAME is described first. Vertical loads input as Load Case 1 are specified as span loading on beams. Static lateral loads input as Load Case 2 are specified as joint loads. The lateral (Y) displacements of the columns at each story level are constrained together using a separate Diaphragm Constraint for each floor. Also, masses are specified only in the lateral (Y) direction at each story level. These are common modeling techniques used to reduce the size of the equation system and are also utilized in the analysis reported in Reference [1]. The Diaphragm Constraints eliminate all axial deformations in the beam. This, and the absence of mass specification in the vertical direction reduces the dynamic problem to seven natural modes of vibration. All seven modes are included in the analysis.

It should also be noted that the AISC section properties in the database file SECTIONS.PRO are not used in this example and the required properties are explicitly entered. This is intentional as most of the sections shown are older sections not in the current AISC database.

The input file FRAMETH is identical to file FRAME for the structural model. However, no static or response spectrum loads are specified. Instead the base acceleration is specified in the Y direction. The acceleration data is discretized in unequal time steps. The output sampling time used is 0.02 seconds and the response is calculated for the first eight seconds. A damping value of 0.05 is used for all modes.

Comparison of Results

Reference [1] presents results only for the static lateral load analysis and the dynamic analysis. A comparison of key results for these analyses is presented in Figures 1-3, 1-4 and 1-5. The static results and the time periods are identical for the two programs. The comparison is excellent for the response spectrum results and good for the time-history results. Explicit time integration, not dependent on the size of the time steps, is used in SAP2000.

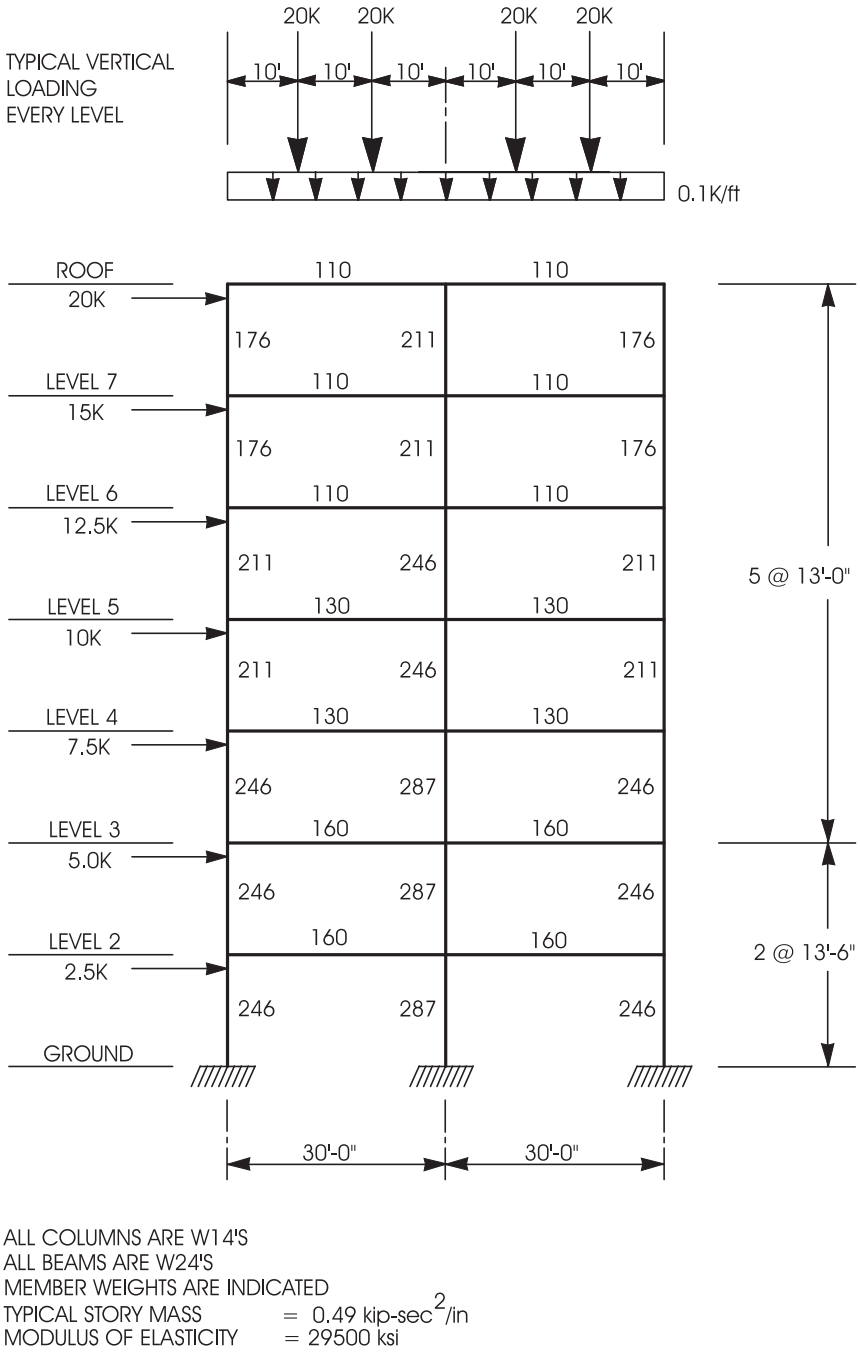
For the response-spectrum results, the program of Reference [1] uses the square root of the sum of the squares (SRSS) method of modal combination. SAP2000

allows both the SRSS method and the complete quadratic combination (CQC) method of modal combination. The CQC method is the default for SAP2000 and is generally recommended. Figure 1-5 presents two sets of results for SAP2000 response spectrum analysis: the default CQC results, and the SRSS results.

Plots of the deformed shape of the structure under the static lateral loads and of the sixth mode shape are shown in Figures 1-6 and 1-7. A plot of time versus displacement at the top of the structure is given in Figure 1-9.

References

1. *Static and Dynamic Analysis of Multistory Frame Structure Using DYNAMIC/EASE2*, Engineering Analysis Corporation and Computers/Structures International.



ALL COLUMNS ARE W14'S
ALL BEAMS ARE W24'S
MEMBER WEIGHTS ARE INDICATED
TYPICAL STORY MASS = 0.49 kip-sec^2/in
MODULUS OF ELASTICITY = 29500 ksi

Figure 1-1
Two-dimensional Frame Example

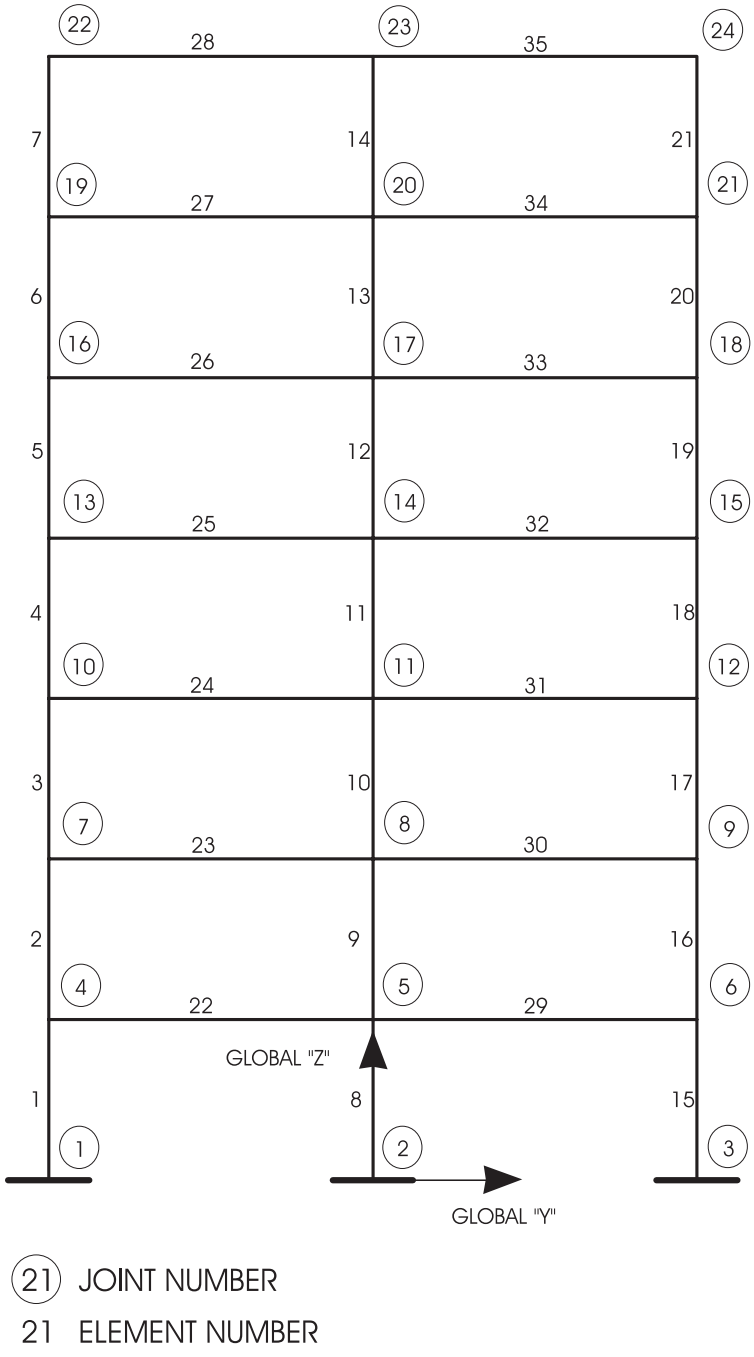


Figure 1-2
Two-dimensional Frame Example Model

Quantity	SAP2000	Reference [1]
Lateral Displacement at Node 22	1.450764	1.450764
Axial Force in Member 1	69.99	69.99
Moment in Member 1 at Node 1	2324.68	2324.68

Figure 1-3
Comparison of Results for Static Lateral Loads

Mode	SAP2000	Reference [1]
1	1.2732	1.2732
2	0.4313	0.4313
3	0.2420	0.2420
4	0.1602	0.1602
5	0.1190	0.1190
6	0.0951	0.0951
7	0.0795	0.0795

Figure 1-4
Comparison of Results for Periods of Vibration

Quantity	SAP2000			Reference [1]	
	Response Spectrum (CQC)	Response Spectrum (SRSS)	Time History	Response Spectrum (SRSS)	Time History
Lateral Displacement at Node 22	5.431	5.437	5.486	5.438	5.46
Axial Force in Member 1	261.5	261.7	263.0	261.8	258.0
Moment in Member 1 at Node 1	9916	9864	9104	9868	8740

Figure 1-5
Comparison of Results for Dynamic Analyses

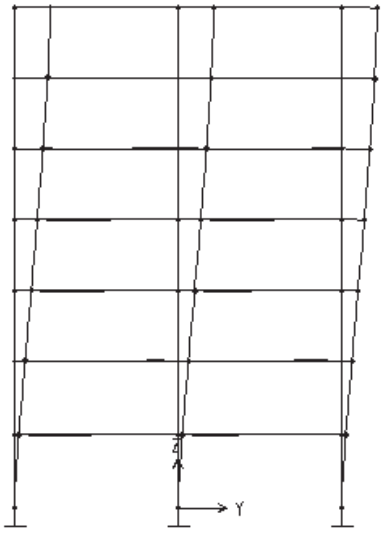


Figure 1-6
Deflection Due to Lateral Loads

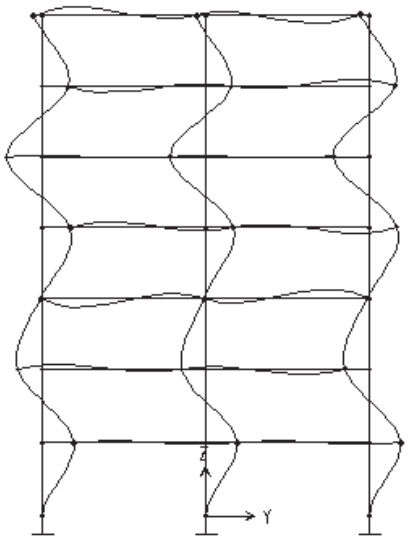


Figure 1-7
Mode Shape 6

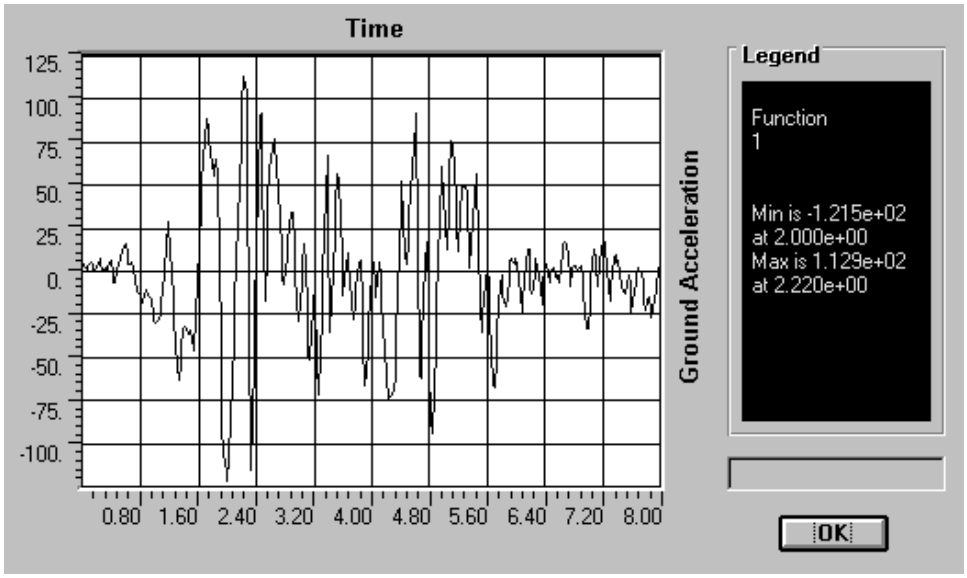


Figure 1-8
El Centro Ground Acceleration Input

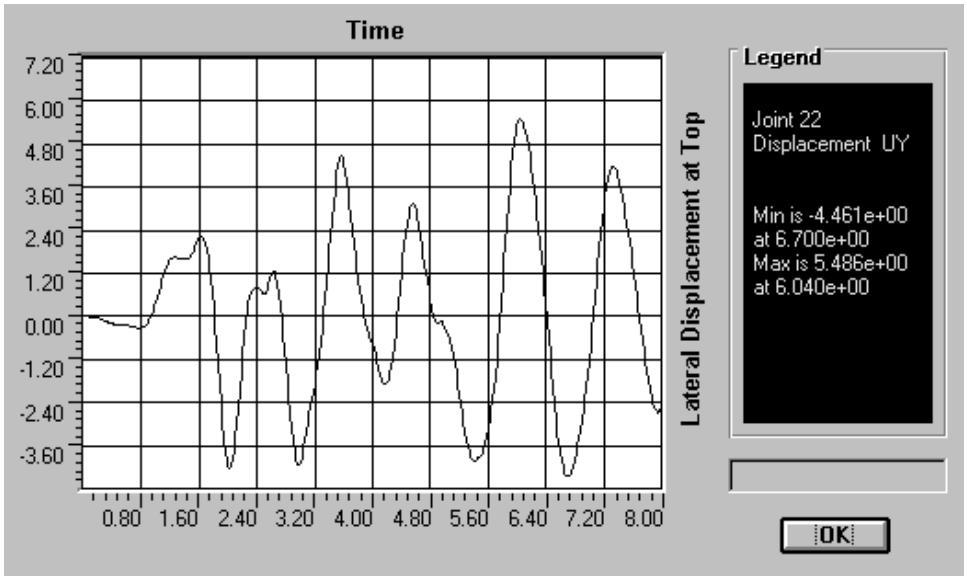


Figure 1-9
Lateral Displacement of Joint 22

Example 2

Bathe and Wilson Frame — Eigenvalue Problem

Description

This is a ten-bay, nine-story, two-dimensional frame structure solved in Reference [1]. The framing and the material and section properties are shown in Figure 2-1. The mass per unit length and other properties used are consistent with References [1] and [2], to which the results are compared. The first three eigenvalues are computed.

Significant Options of SAP2000 Activated

- Two-dimensional frame analysis
- Eigenvalue analysis

Input Data

The computer model used is shown in Figure 2-1. Kip-foot units are used. Mass per unit length of the members is specified. The program automatically computes the joint masses to be used in the eigenvalue analysis.

The input data file for this example is FRAMEBW.

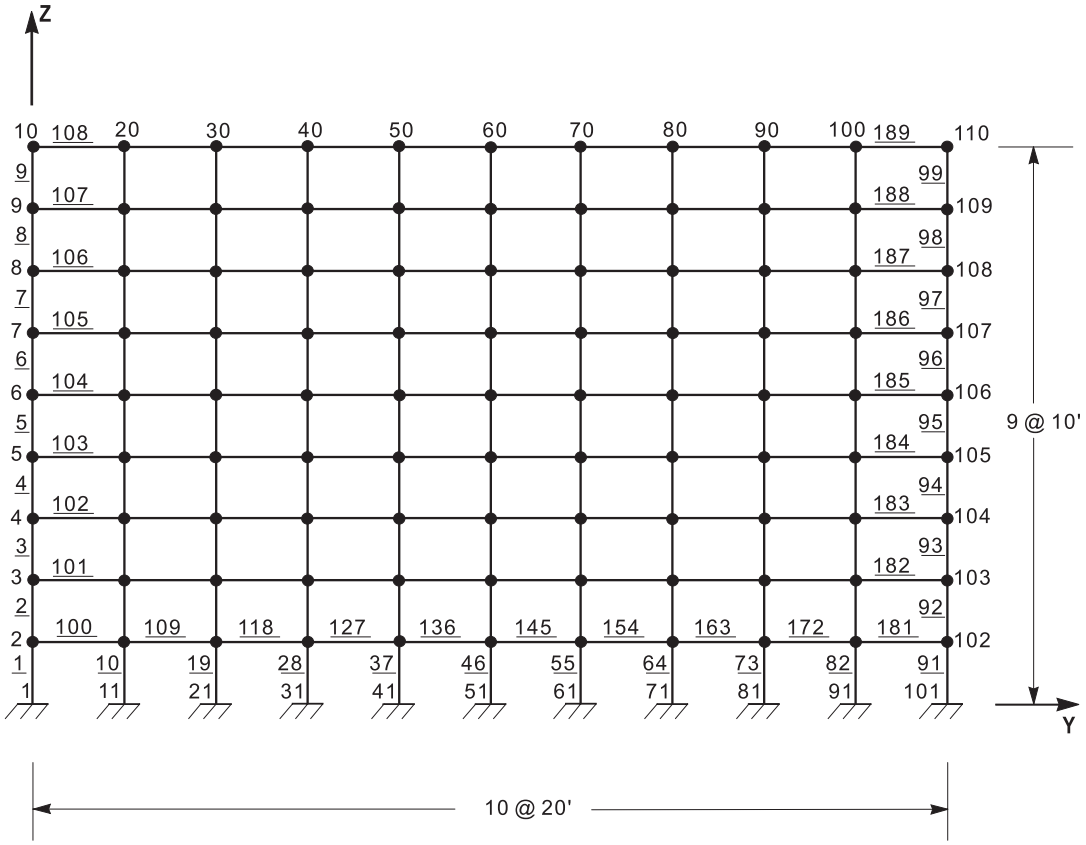
Comparison of Results

A comparison of the first three eigenvalues computed by SAP2000 with results from References [1] and [2] is presented in Figure 2-2. The comparison is excellent.

References

1. **Bathe, K. J. and Wilson, E. L.**
Large Eigenvalue Problems in Dynamic Analysis, Journal of the Eng. Mech. Div., ASCE, Vol. 98, No. EM6, Proc. Paper 9433, Dec. 1972.

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



4 JOINT NUMBER
4 MEMBER NUMBER

TYPICAL PROPERTIES
AREA = 3 ft^2
MOMENT OF INERTIA = 1 ft^4
MODULUS OF ELASTICITY = 4.32x10^5 ksf
MASS PER UNIT LENGTH = 3 kip-sec^2/ft/ft

Figure 2-1
Bathe and Wilson Frame Example

Mode	SAP2000	Reference [1]	Reference [2]
1	0.589541	0.589541	0.589541
2	5.52696	5.52695	5.52696

Figure 2-2
Comparison of Eigenvalues

Example 3

Three-dimensional Frame — Dynamic Loads

Description

This is a two-story, three-dimensional frame structure with rigid diaphragm floors. The problem is the same as the one solved in Reference [1]. The framing is shown in Figures 3-1 and 3-2. The structure is doubly symmetric in plan, except that the center of gravity at each story level is eccentric and is given by coordinates X = 38 feet and Y = 27 feet, represented in Figure 3-1 by joints 28 and 29.

Significant Options of SAP2000 Activated

- Three-dimensional frame analysis
- Rigid diaphragm modeling
- Response spectrum analysis

Input Data

The computer model used is shown in Figures 3-1 and 3-2. Kip-foot units are used. An additional joint is added to each story at the center of gravity, and all story mass is given at these two joints.

Two rigid Diaphragm constraints are defined, one each for Stories 1 and 2. All joints on story 1 are constrained together, including the joint at the center of gravity. Similarly, all joints for story 2 are constrained together. For each story, the X and Y displacements and the Z rotations for all joints are dependent upon each other.

Masses at the centers of gravity are specified in the X and Y directions. No rotational mass inertia is used for consistency with Reference [1]. It should be noted that the problem has only four natural modes. All four modes are used in the analysis.

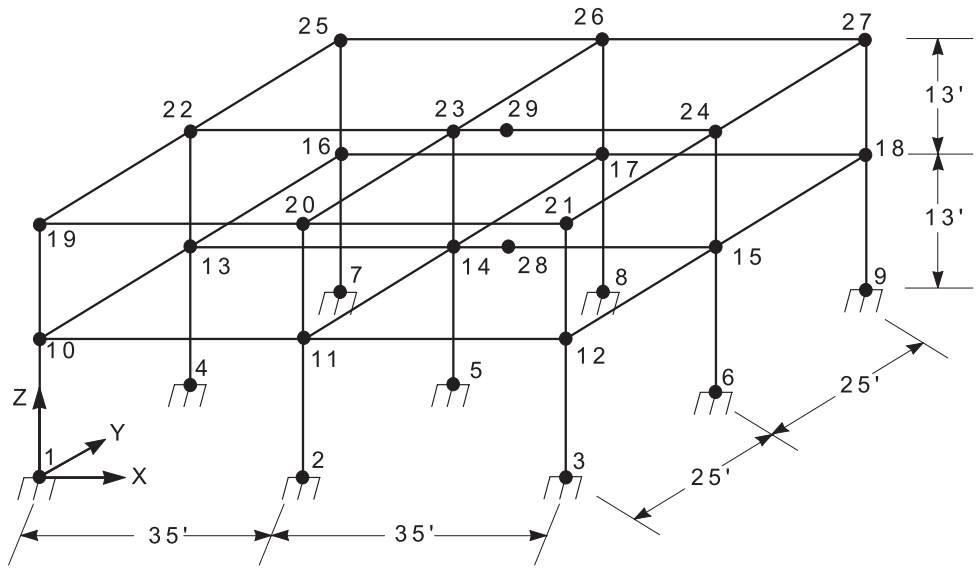
The input data file for this example is FRAME3D.

Comparison of Results

A comparison of the SAP2000 results with Reference [1] results for the four natural periods of vibration and the X-deflection at Joint 29 is presented in Figure 3-3. The comparison is excellent.

Reference

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



STORY 1 C.G. (NODE 28) AT (38,27,13)
STORY 2 C.G. (NODE 29) AT (38,27,26)
TYPICAL STORY MASS = 6.212 kip-sec^2/ft

Figure 3-1
Three-dimensional Frame Example: Dimensions

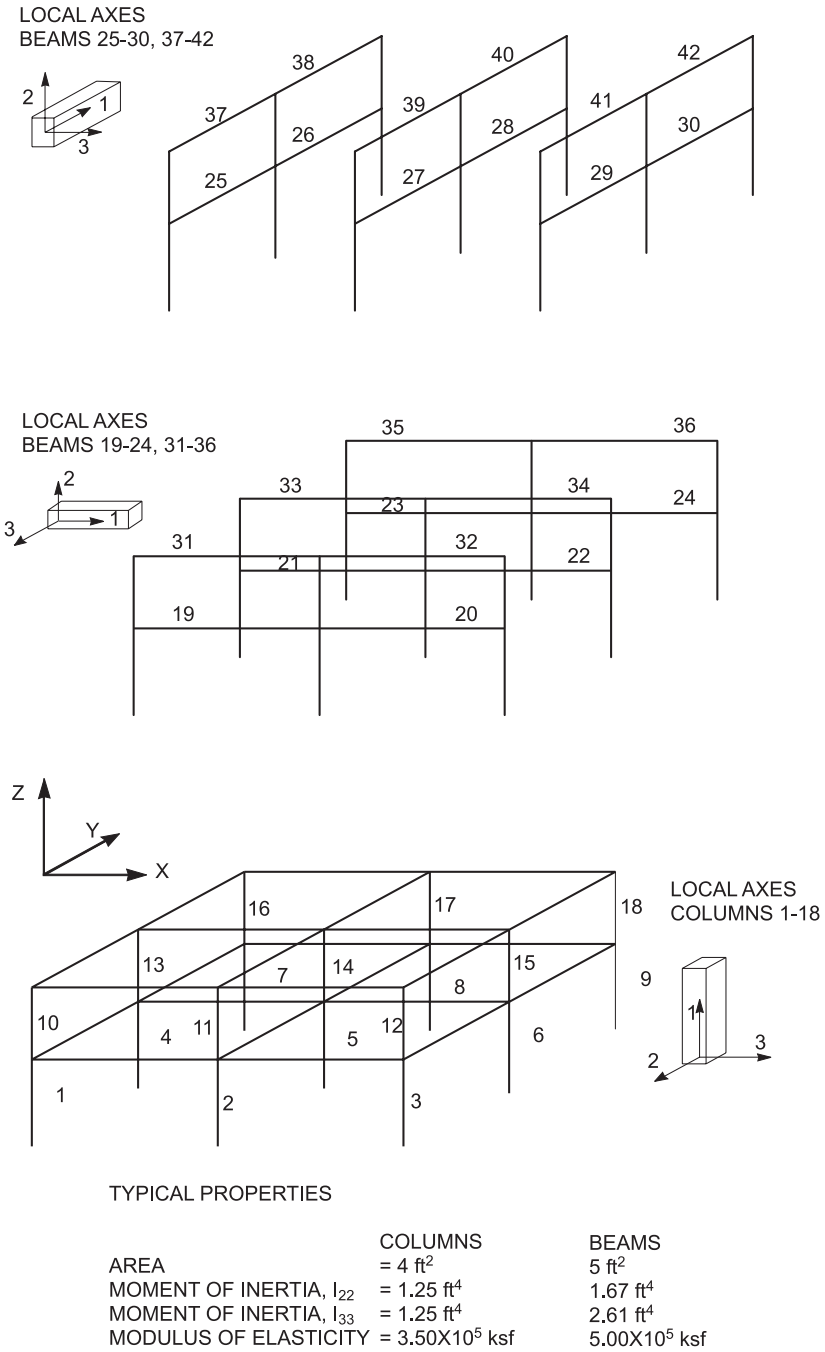


Figure 3-2
Three-dimensional Frame Example: Element Properties

Quantity	SAP2000	Reference [1]
Period, Mode 1	0.2271	0.2271
Period, Mode 2	0.2156	0.2156
Period, Mode3	0.0733	0.0733
Period, Mode 4	0.0720	0.0720
X Deflection, Joint 29	0.0201	0.0201

Figure 3-3
Comparison of Results

Example 4

ASME Frame — Eigenvalue Problem

Description

This is a single-story, single-bay in each direction, three-dimensional frame structure made of 2-inch steel pipe segments and 2.75-inch steel cubes as shown in Figure 4-1. The frame is the same as modeled in References [1] and [2] and is Problem 1 from the ASME 1972 Program Verification and Qualification Library (Reference [3]).

Significant Options of SAP2000 Activated

- Three-dimensional frame analysis
- Use of rigid end offsets on Frame elements
- Eigenvalue analysis
- Ritz vector analysis

Input Data

The computer model used is shown in Figure 4-2. The pipe segments are modeled using Frame elements. For consistency with Reference [1], masses are specified at the nodes instead of using the Frame member mass per unit length specification and additional nodal masses only for the solid cubes. Masses identical to those used in

Reference [1] are specified. Since masses at 14 nodes are specified in three directions, this problem has 3 x 14 = 42 dynamic degrees of freedom. The first 24 modes are calculated using both eigenvectors and Ritz vectors. The Ritz vectors are calculated using ground acceleration in the three global directions as the starting load vectors. The input data file for this example using Ritz vector analysis is FRAMASME. For eigenvector analysis, just change the type of modes requested.

Comparison of Output

A one-to-one comparison of SAP2000 results with References [1] and [2] results for this problem is not possible. This is because both References [1] and [2] use the Guyan reduction method to reduce the 42 dynamic degrees of freedom problem to 24. This introduces approximations into the solution. A comparison of the results is presented in Figure 4-3 for the first 12 natural frequencies. Two sets of results are presented for SAP2000: one using Ritz vector analysis, and one using eigenvector analysis.

The comparison between SAP2000 and References [1] and [2] is good considering the modeling differences between the different solutions. The SAP2000 eigenvector and Ritz-vector results are essentially the same for the first 11 modes, but begin to differ in the higher modes. In general, only the eigenvectors represent the natural modes of the structure. The Ritz vectors are a better basis for response-spectrum and time-history analyses, but may not have the same frequencies and mode shapes as the eigenvectors.

References

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

2. **DeSalvo, G. J. and Swanson, J. A**.
ANSYS, Engineering Analysis System, Examples Manual, Swanson Analysis Systems, Inc., Elizabeth, Pennsylvania, 1977.

3. *Program Verification and Qualification Library*, ASME Pressure Vessel and Piping Division, Committee on Computer Technology, 1972.

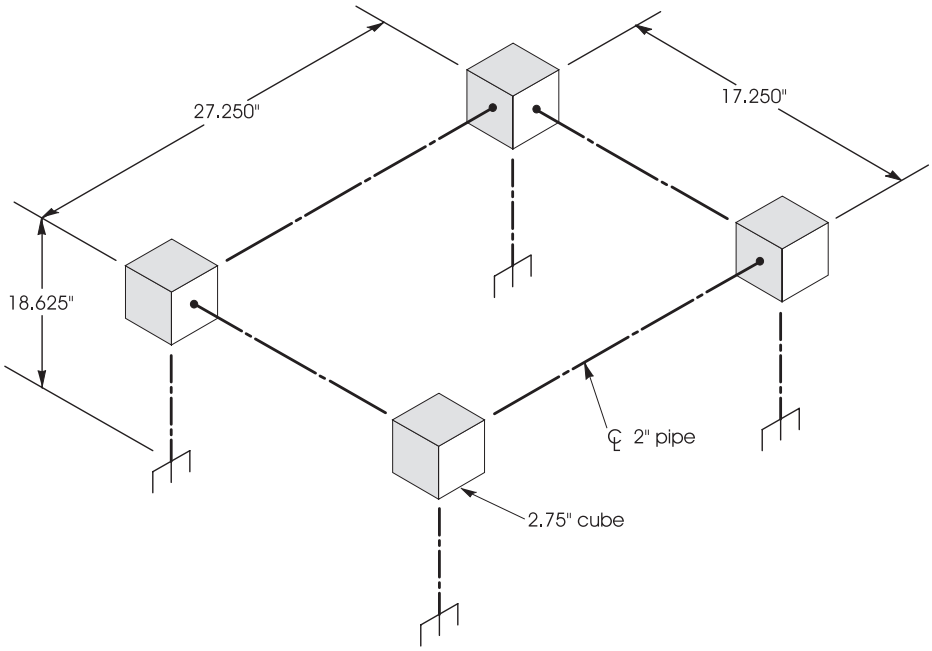
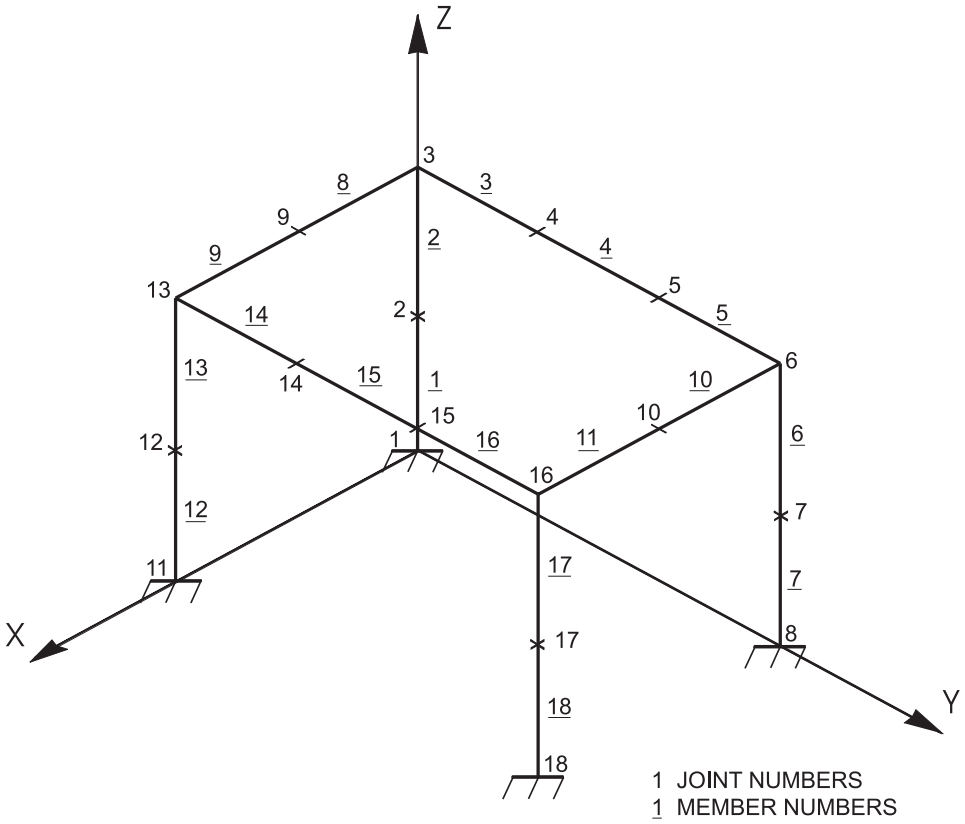


Figure 4-1
ASME Frame Example



TYPICAL PROPERTIES
2.375 in O.D. PIPE 0.154" THICK
MODULUS OF ELASTICITY = 27.9×10^{6} psi
POISSON'S RATIO = 0.3
JOINT MASSES = 8.942×10^{-3} lb-sec^2/in
ADDITIONAL MASSES AT
JOINTS 3,6,13 and 16 = 16.439×10^{-3} lb-sec^2/in

Figure 4-2
ASME Frame Model

Mode	SAP2000 Ritz Analysis	SAP2000 Eigen Analysis	Reference [1] & Reference [2] *
1	114	114	112
2	119	119	116
3	141	141	138
4	222	222	218
5	399	399	404
6	422	422	423
7	450	450	452
8	550	550	554
9	774	774	736
10	800	799	762
11	909	909	853
12	955	946	894

* *Both results are based upon Guyan reduction from 42 to 24 dynamic degrees of freedom.*

Figure 4-3
Comparison of Modal Frequencies

Example 5

Three-dimensional Braced Frame — Dynamic Loads

Description

This is a three-story, L-shaped braced frame structure. The floors act as rigid diaphragms. The framing consists of four identical frames and the frame members carry only axial loads. The problem is identical to the one solved in Reference [1]. The framing is shown in Figure 5-1.

Significant Options of SAP2000 Activated

- Three-dimensional frame analysis
- Axial-load-only frame members
- Rigid diaphragm modeling
- Response spectrum analysis

Input Data

The computer model used for Frame 1 is shown in Figure 5-2. Kip-inch units are used. The models for Frames 2, 3 and 4 are identical except that node numbers are incremented by 12 and member numbers by 21, for each successive frame. It must be noted that the common column between Frames 2 and 3 is modeled twice, once for each frame. This is done for consistency with the modeling in Reference [1].

Joints 49, 50 and 51 are specified at the center of gravity of stories 1, 2 and 3, respectively. X- and Y-direction masses and the mass moment of inertia about the Z-axis are specified at these joints. All joints on a given story are connected together using a Diaphragm constraint.

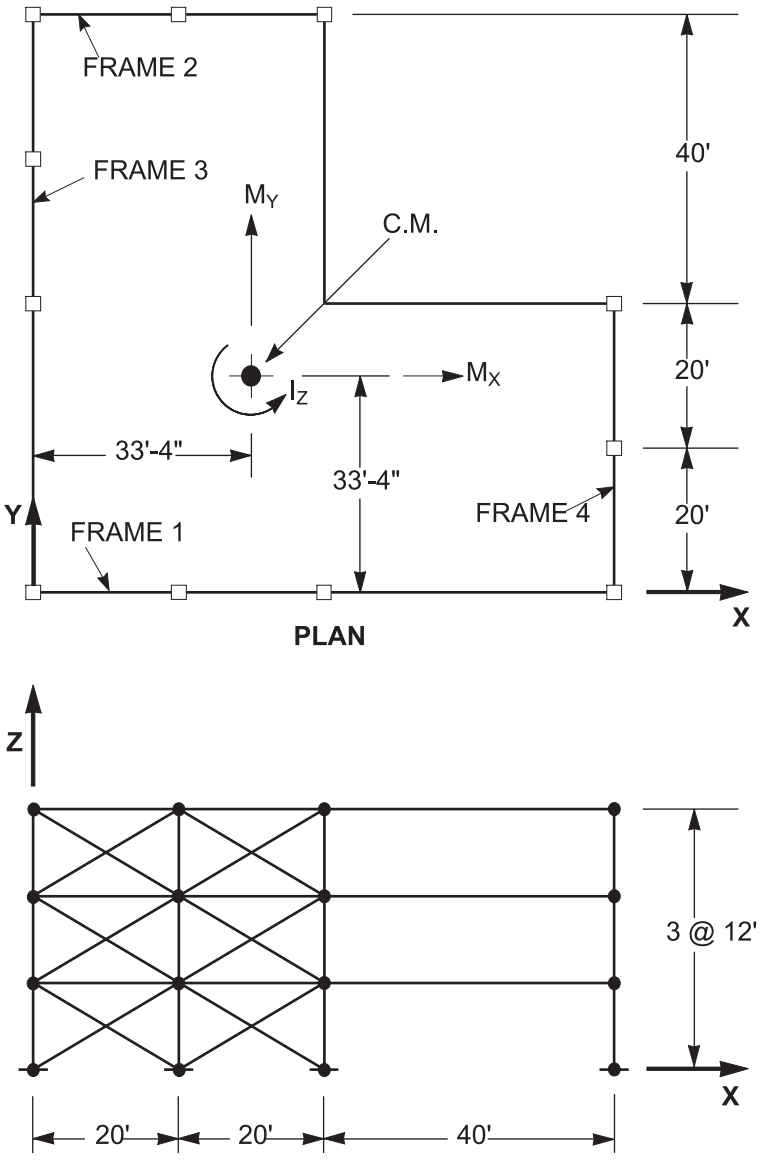
The modeling is identical to that used in Reference [1]. Also, only the first two modes are used in the response spectrum analysis. The input data file for this example is FRAMBRAC.

Comparison of Results

Key results from the SAP2000 analysis are compared with the Reference [1] solutions in Figure 5-3. The comparison is excellent.

Reference

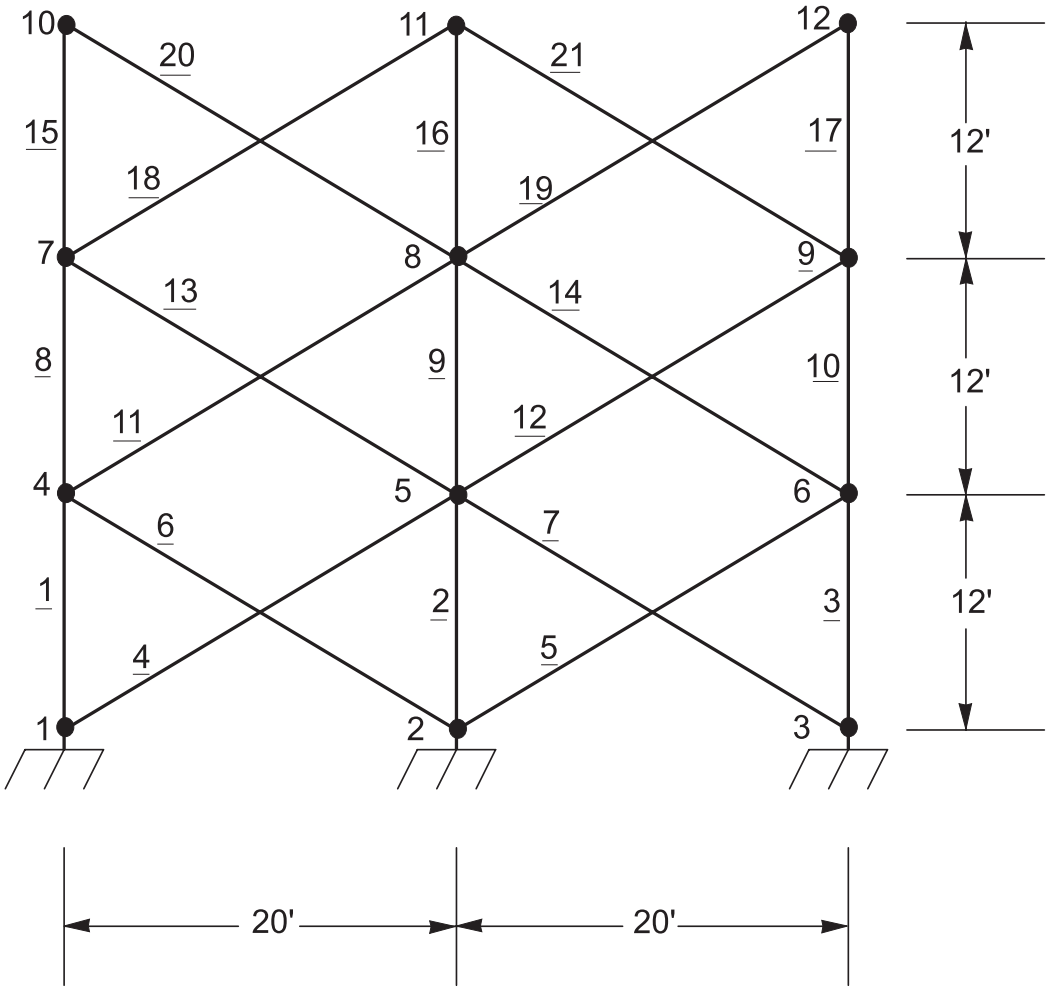
1. **Peterson, F. E.**
EASE2, Elastic Analysis for Structural Engineering, Example Problem Manual, Engineering Analysis Corporation, Berkeley, California, 1981.



TYPICAL STORY MASSES, M_X and M_Y = 1.24224 kip-sec^2/in

TYPICAL STORY MASS MOMENT OF INERTIA, I_Z = 174.907X10^3 kip-sec^2-in

Figure 5-1
Three-dimensional Frame Example



3 Joint Numbers
3 Member Numbers

ELEVATION

TYPICAL PROPERTIES
AREA = 6 in^2
MODULUS OF ELASTICITY = 29500 ksi

Figure 5-2
Three-dimensional Braced Frame Example — Model of Frame 1

Quantity	SAP2000	Reference [1]
Period, Mode 1	0.326887	0.326887
Period, Mode 2	0.320640	0.320640
Axial Force, Member 1	279.47	279.48
Axial Force, Member 4	194.50	194.50
Axial Force, Member 5	120.52	120.52

Figure 5-3
Comparison of Results for Three-dimensional Braced Frame Example

Example 6

Beam — Steady-State Harmonic Loads

Description

This is a fixed-end beam in two dimensions subjected to a uniformly distributed load which varies harmonically with respect to time. The beam is shown in Figure 6-1. The problem is the same as given in Reference [1]. The beam is solved twice, once using the undamped, steady-state analysis option and once using the periodic loading, time-history analysis option.

Significant Options of SAP2000 Activated

- Two-dimensional frame analysis
- Steady-state analysis
- Time-history analysis for periodic loading

Input Data

The computer model used is shown in Figure 6-1. Pound-inch units are used.

For the steady-state option the frequency of the forcing function is provided in cycles per second. The input data file for this option is BEAM.

For the time-history option the time function (sine wave) portion of the loading is discretized at 37 points at equal intervals covering one complete cycle of loading. See Figure 6-2. The input data file for this example is BEAMTH.

Comparison of Results

Reference [1] computes the deflection at the center of the beam to be -0.0541 sin 300t. The SAP2000 result for the amplitude of this deflection for the steady-state option is -0.054535. It should be noted that Reference [1] uses only the first five mode shapes of the beam for its computation. The SAP2000 results for the time-history option are shown in Figure 6-3. The maximum amplitude being reported is -0.05440. The comparison of results for both methods of analysis in SAP2000 with the theoretical results is excellent.

Reference

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

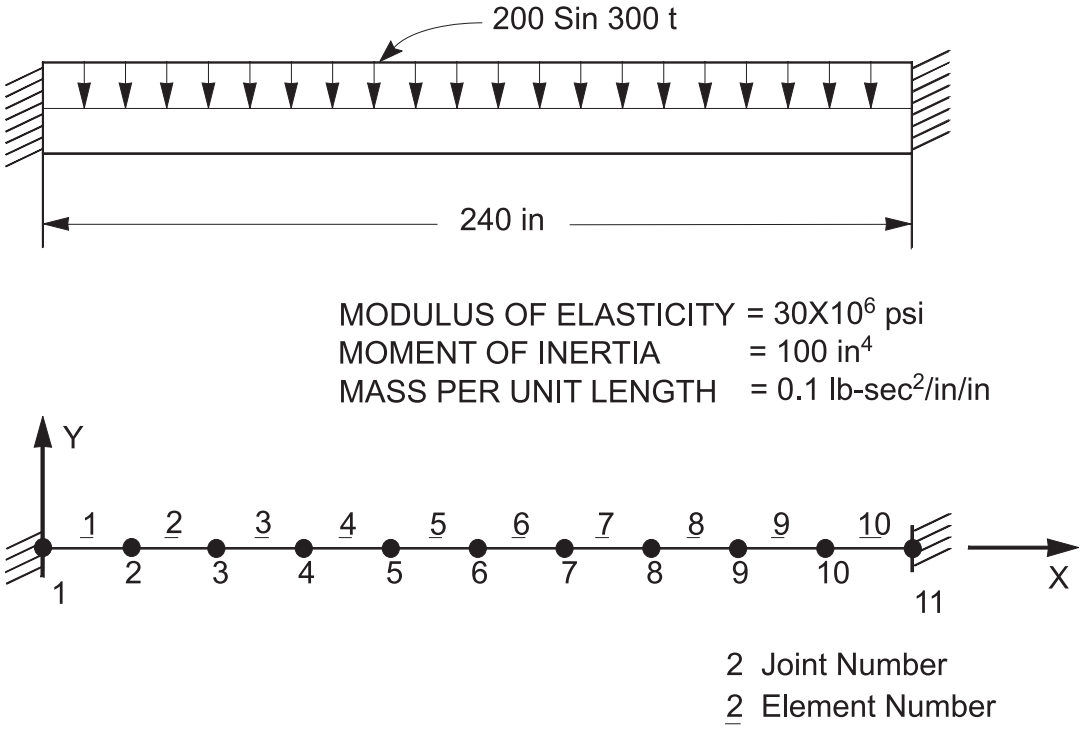


Figure 6-1
Beam Example

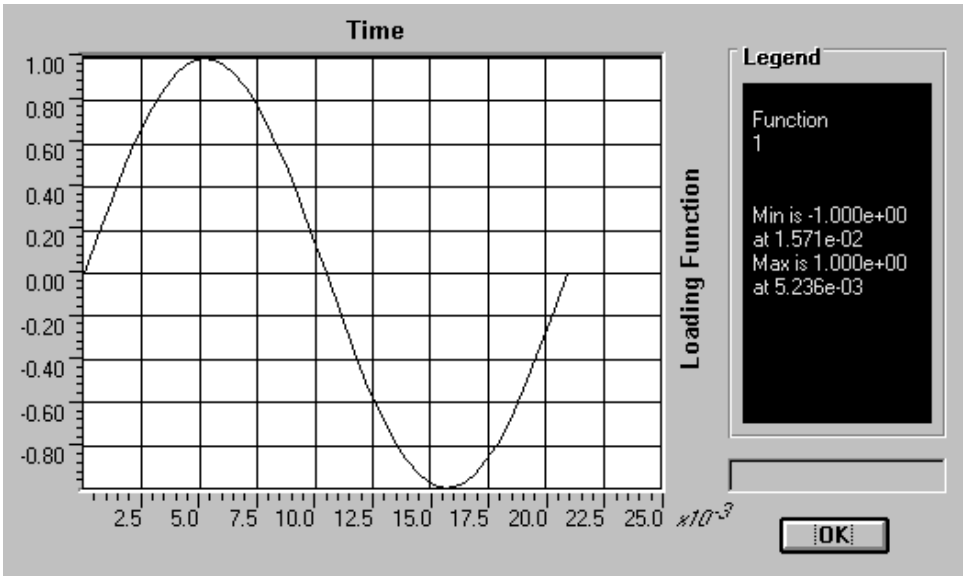


Figure 6-2
Time Variation of Loading

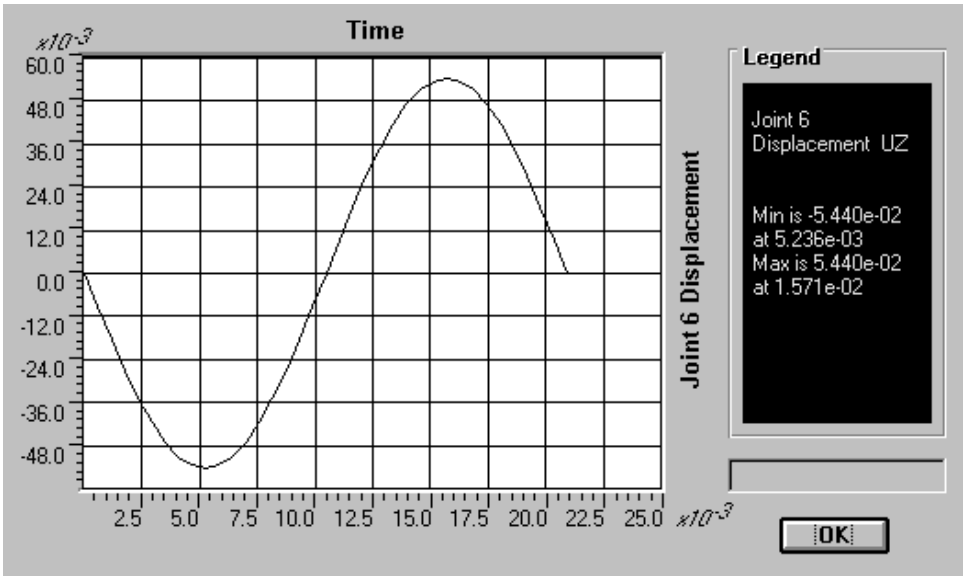


Figure 6-3
Center Span Displacement

Example 7

Two-dimensional Truss — Static Loads

Description

This is a two-dimensional truss structure. All members can carry only axial loads. The truss is shown in Figure 7-1.

Significant Options of SAP2000 Activated

- Two-dimensional truss analysis
- Vertical joint loads

Input Data

The computer model used is shown in Figure 7-1. Truss members are modeled using frame elements with zero moments of inertia. Load case 1 is live load, and Load case 2 is dead load. Load combination 1 is dead load plus live load.

The input data file for this example is TRUSS.

Comparison of Results

This example is included as a sample only. Other results for comparison are not available.

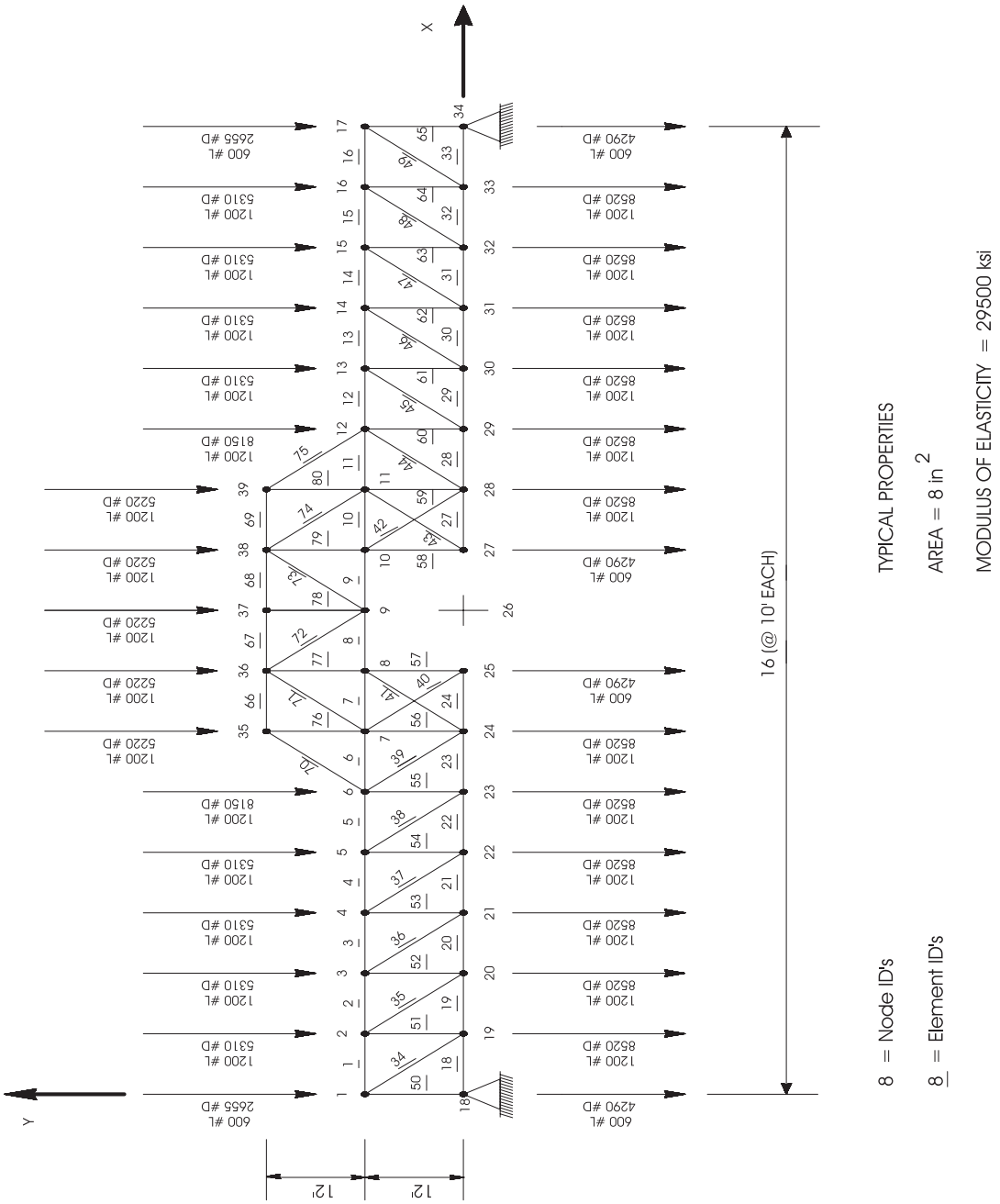


Figure 7-1
Two-dimensional Truss Example

Example 8

Three-dimensional Building — Dynamic Loads

Description

This is a two-story, three-dimensional framed building. The structure is shown in Figure 8-1. The floors act as rigid diaphragms. The building is unsymmetrical and is subjected to lateral dynamic loads along two horizontal axes at a 30^o angle to the building axes.

Significant Options of SAP2000 Activated

- Three-dimensional frame analysis
- Rigid diaphragm modeling
- Response-spectrum analysis in two directions

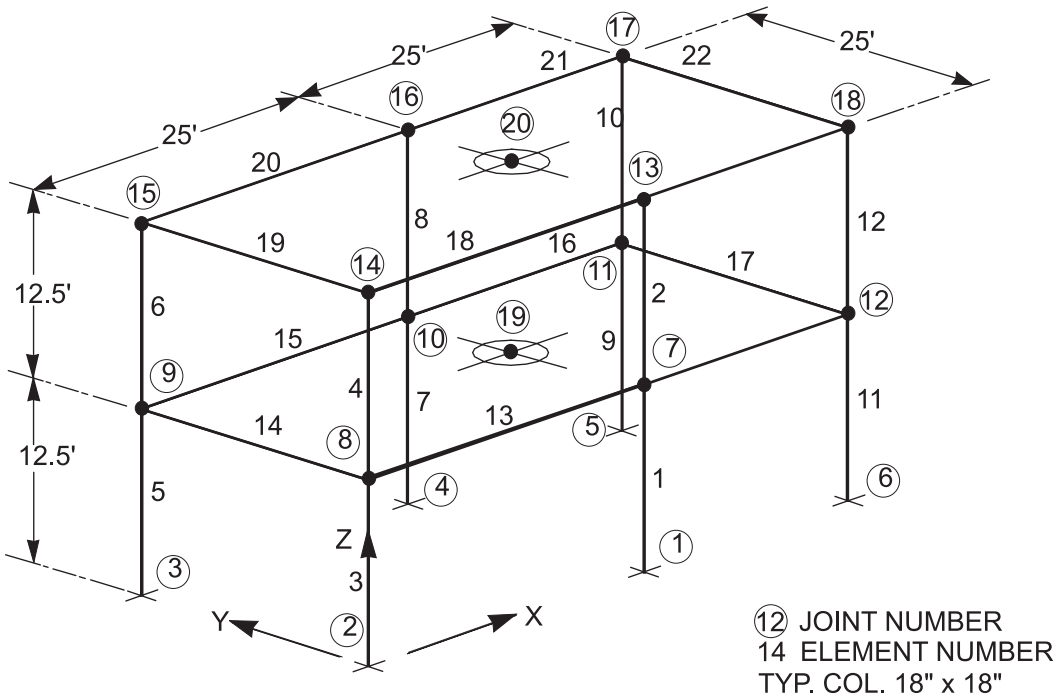
Input Data

The computer model used is shown in Figure 8-1. Kip-foot units are used. Joints 19 and 20 are specified at the center of gravity of Stories 1 and 2, respectively, and the X- and Y-direction masses and the mass moment of inertia about the Z axis is defined at these joints only. All six modes of the structure are used for the dynamic analysis. The response spectra are defined for two horizontal axes at a 30^o angle to the building axes.

The input data file for this example is BUILDING.

Comparison of Results

This example is included as a sample only. Other results for comparison are not available.



MODULUS OF ELASTICITY	= 432000 ksf
TYPICAL STORY MASSES, M_X and M_Y	= 3.88 kip-sec^2/ft
TYPICAL STORY MASS MOMENT OF INERTIA, I_Z	= 1011 kip-sec^2-ft

Figure 8-1
Three-dimensional Building Example

Example 9

Patch Tests — Prescribed Displacements

Description

This is a rectangular plate with irregularly-shaped elements and subjected to prescribed displacements at the edges. The plate is shown in Figure 9-1. The location of the inner nodes and the prescribed displacements are the same as suggested in Reference [1]. The problem is solved using both Plane Stress elements and Shell elements.

Significant Options of SAP2000 Activated

- Plane stress analysis using 4-node Plane elements
- Displacement specification
- Plate bending and membrane analysis using Shell elements

Input Data

The computer model used is shown in Figure 9-1. The prescribed displacements at the edges are calculated as:

$$u = 10^{-3} \left(x + \frac{y}{2} \right)$$

$$v = 10^{-3} \left(y + \frac{x}{2} \right)$$

$$w = 10^{-3} \frac{\left(x^2 + xy + y^2 \right)}{2}$$

$$\Theta_x = 10^{-3} \left(y + \frac{x}{2} \right)$$

$$\Theta_y = 10^{-3} \left(- x - \frac{y}{2} \right)$$

These represent constant stress fields.

For the Plane element solution, only the X- and Y-translations are unrestrained. All other degrees of freedom are restrained. For the Shell element solution, all six degrees of freedom are unrestrained because the membrane formulation of this shell element gives rotational stiffness components about a direction normal to the plane of the shell.

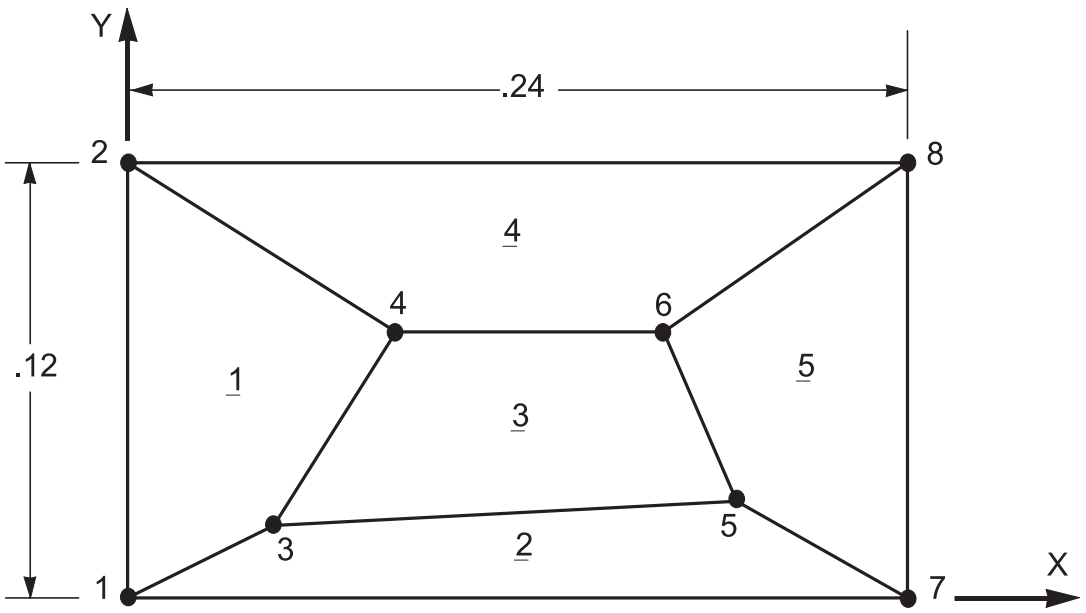
For this example, the input data file using the Plane element is PATCHPLN, and that using the Shell element is PATCHSHL.

Comparison of Results

The theoretical results for the problem are $S_{xx} = S_{yy} = 1333$ and $S_{xy} = 400$ for the membrane components; and $M_{xx} = M_{yy} = 1.111 \times 10^{-7}$ and $M_{xy} = 0.333 \times 10^{-7}$ for the plate bending components. These theoretical results are reproduced by SAP2000.

Reference

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



Location of inner nodes:

	x	y
3	.04	.02
4	.08	.08
5	.18	.03
6	.16	.08

3 Joint Number
3 Element Number

THICKNESS = 0.001
MODULUS OF ELASTICITY = $1.0X10^6$
POISSON'S RATIO = 0.25

Figure 9-1
Patch Test Example

Example 10

Straight Beam — Static Loads

Description

This is a straight cantilever beam of proportions shown in Figure 10-1. The beam has unit forces at the tip in the three orthogonal directions and a unit twist, each modeled as a separate Load case. The beam is modeled using different mesh geometries as suggested in Reference [1]. The problem is solved using Shell elements, nine-node Plane stress elements ,and Solid elements.

Significant Options of SAP2000 Activated

- Shell element analysis
- Plane element analysis with plane-stress option
- Solid element analysis with and without incompatible bending modes

Input Data

Several mesh geometries are used and key results for these are presented. The following data files are provided:

- STRBMSHL for the rectangular mesh of Shell elements.
- STRBMPLN for the rectangular mesh of Plane elements.

- STRBMSOL for the rectangular mesh of Solid elements with incompatible bending modes.

Figures 10-1 to 10-3 represent the models used for the Shell, Plane and Solid elements.

The unit forces at the tip are applied as 1/2 at each node for the Shell element; 1/6 at each corner node and 4/6 at the midside node for the Plane element; and 1/4 at each node for the Solid element. This represents a consistent set of forces for the elements. The unit twist for the Shell and Solid elements is applied as a couple.

For the model using Shell elements the nodes at the fixed end are restrained in the axial direction, in the out-of-plane direction, and for rotations that cause out of plane bending. The transverse, in-plane direction is restrained only at one node so as not to cause local Poisson's effect. The other node is, however, provided with the transverse reaction as a load.

For the model using Plane elements the nodes at the fixed end are restrained in the axial direction and one of them is also restrained in the transverse, in-plane direction for the same reason as for the Shell element. The other nodes are, however, provided with the transverse reactions as loads. All rotations and the out-of-plane direction for all nodes are also restrained.

For the model using Solid elements the nodes at the fixed end are restrained in the axial direction and one of them is also restrained in the two transverse directions for the same reason as for the Shell element. This , however, does not make the model stable, so an additional node is restrained in the vertical direction to prevent rotation about the axial direction. The other nodes are, however, provided with the transverse reactions as loads.

Comparison of Results

The displacements at the tip in the direction of the load is compared for each type of element and the different meshes with the theoretical results in Figures 10-4, 10-5, and 10-6.

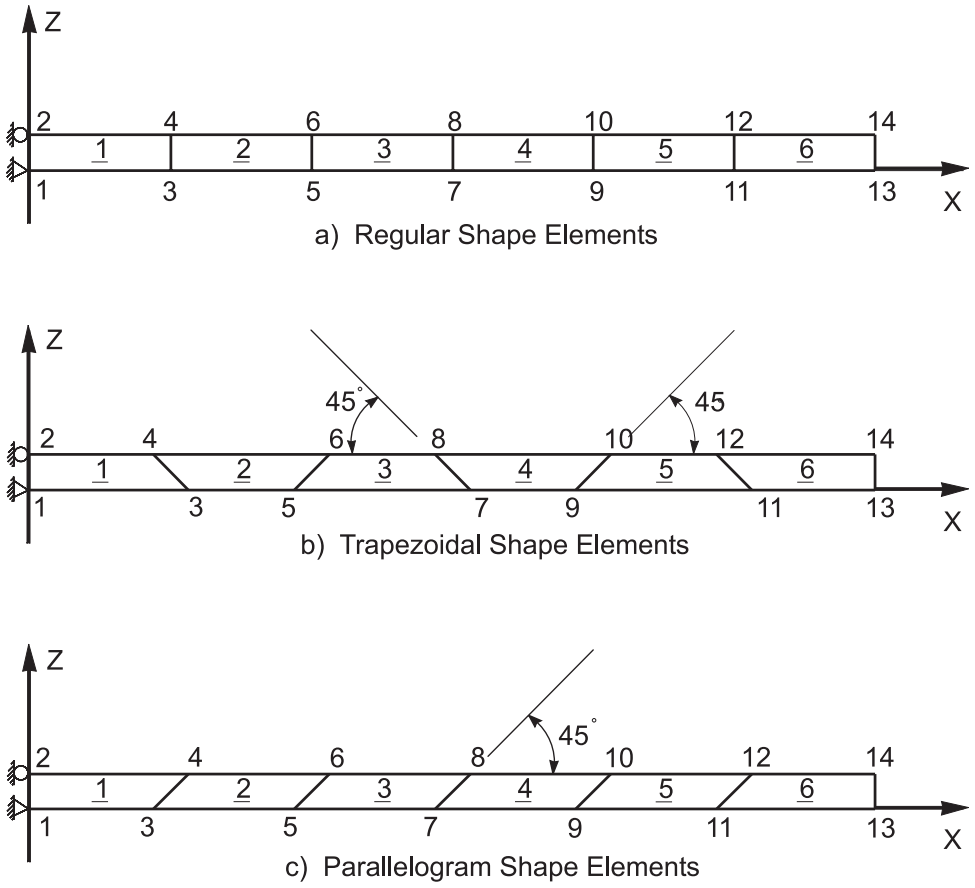
The results for the Shell element are good except for the in-plane shear results for the irregular meshes and the results for the unit twist. The twist results are too stiff. The element is too thick compared to the width for the thin plate twisting behavior the Shell element is capable of modeling. The irregular mesh behavior in in-plane shear can be improved by decreasing the skewness and aspect ratio of the elements and using more elements.

The results for the nine-node Plane element for all mesh geometries is excellent. The four-node Plane element is too stiff in bending and a finer mesh would be required to accurately capture the bending behavior.

The results for the Solid element with a rectangular mesh and using incompatible bending modes are good except for the case with the unit twist. To capture the twisting behavior accurately, more elements are needed across the beam section. The Solid element model without incompatible bending modes is too stiff in the bending mode. A finer mesh along the length of the beam would give better results.

Reference

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



LENGTH	= 6.0
WIDTH	= 0.2
THICKNESS	= 0.1
MODULUS OF ELASTICITY	= $1.0X10^7$
POISSON'S RATIO	= 0.30

LOADING: UNIT FORCES AT FREE END

Figure 10-1
Straight Beam Example - Shell Element Models

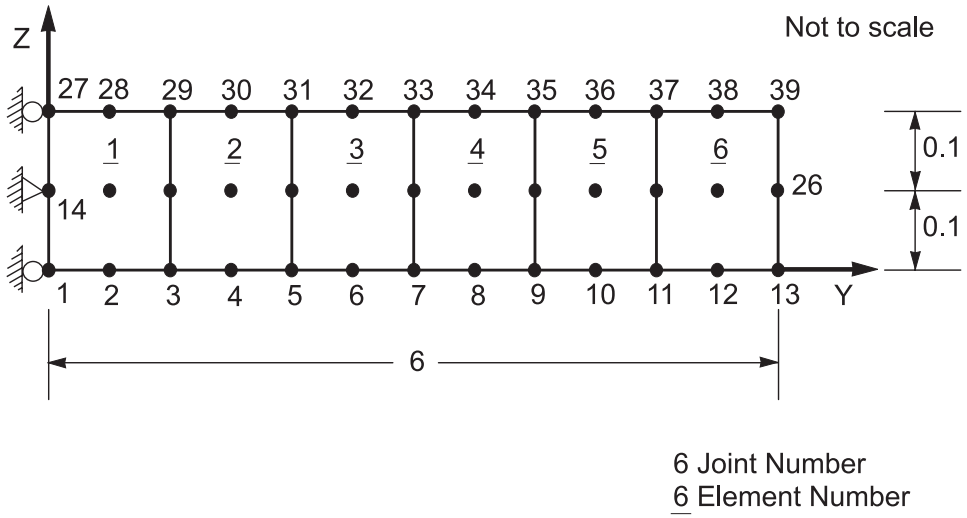


Figure 10-2
Straight Beam Example - Plane Stress Element Model

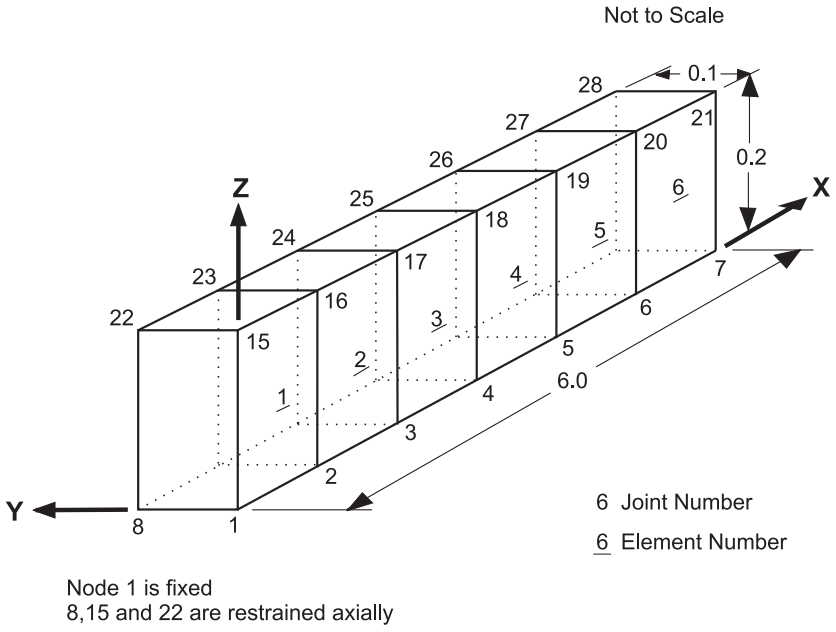


Figure 10-3
Straight Beam Example - Solid Element

Load Direction	SAP2000			Theoretical
	Rectangular Mesh	Trapezoidal Mesh	Parallelogram Mesh	
Extension	$3.000x10^{-5}$	$3.000x10^{-5}$	$3.000x10^{-5}$	$3.000x10^{-5}$
Out-of-plane Shear	0.4263	0.4266	0.4266	0.4321
In-plane Shear	0.1072	0.0221	0.0790	0.1081
Twist	0.00233	0.00233	0.00233	0.00321

Figure 10-4
Comparison of Tip Displacements Using Shell Elements

Load Direction	SAP2000				Theoretical
	Rectangular Mesh 4-Node Element	Rectangular Mesh 9-Node Element	Trapezoidal Mesh 9-Node Element	Parallelo-gram Mesh 9-Node Element	
Extension	$3.000x10^{-5}$	$3.000x10^{-5}$	$3.000x10^{-5}$	$3.000x10^{-5}$	3.000x10-5
In-plane Shear	0.0101	0.1076	0.1063	0.1065	0.1081

Figure 10-5
Comparison of Tip Displacements Using Plane Stress Element

Load Direction	SAP2000		Theoretical
	Rectangular Mesh with Incompatible Bending Modes	Rectangular Mesh without Incompatible Bending Modes	
Extension	3.000×10^{-5}	3.000×10^{-5}	3.000×10^{-5}
Out-of-plane Shear	0.4283	0.0109	0.4321
In-plane Shear	0.1072	0.0101	0.1081
Twist	0.00286	0.00286	0.00321

Figure 10-6
Comparison of Tip Displacements Using Solid Elements

Example 11

Curved Beam — Static Loads

Description

This is a curved cantilever beam of proportions shown in Figure 11-1. The beam is loaded with unit shears at the free end. The problem is the same as suggested in Reference [1]. The problem is solved using Shell elements and Plane elements.

Significant Options of SAP2000 Activated

- Shell element analysis
- Plane element analysis using plane- stress option

Input Data

The input data file for the Shell element example is CRVBMSHL, and for the Plane element example is CRVBMPLN.

Comparison of Results

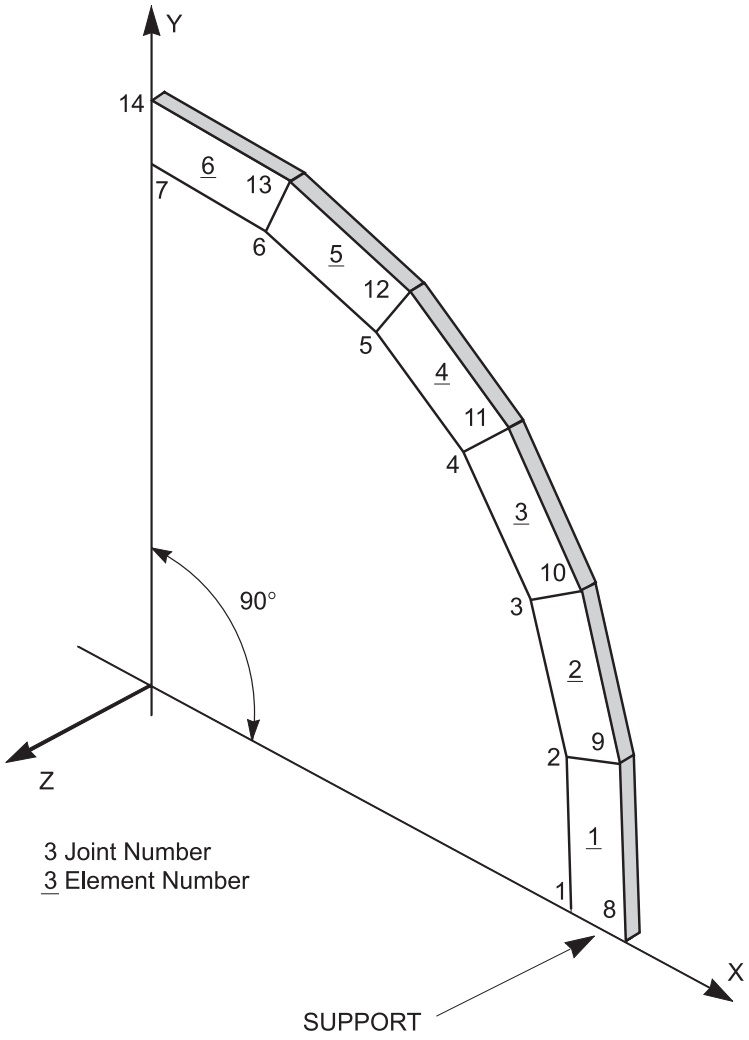
The displacement at the free end in the direction of the load is compared with theoretical results (Reference [1]) for both element types in Figure 11-2.

For the Plane element, more elements along the length of the beam would give better results. The aspect ratio of the element is quite large in this model.

For the Shell element, the in-plane (membrane) results are good; the out-of-plane (bending) results are not as good. This is because of the stiffer twisting behavior when the thickness is large compared with the width.

Reference

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



INNER RADIUS	= 4.12
OUTER RADIUS	= 4.32
ARC	= 90°
THICKNESS	= 0.1
MODULUS OF ELASTICITY	= $1.0X10^7$
POISSON'S RATIO	= 0.25

LOADING: UNIT FORCES AT FREE END

Figure 11-1
Curved Beam Example — Shell Element Model

Load Direction	SAP2000		Theoretical
	9-Node Plane Stress Element	Shell Element	
In-plane Shear	0.0775	0.0851	0.0873
Out-of-plane Shear		0.4518	0.5022

Figure 11-2
Comparison of Tip Displacements

Example 12

Twisted Beam — Static Loads

Description

This is a twisted cantilever beam of proportions shown in Figure 12-1. Unit loads are applied at the free end. The problem is the same as suggested in Reference [1]. The problem is solved using Shell elements.

Significant Options of SAP2000 Activated

- SHELL element analysis

Input Data

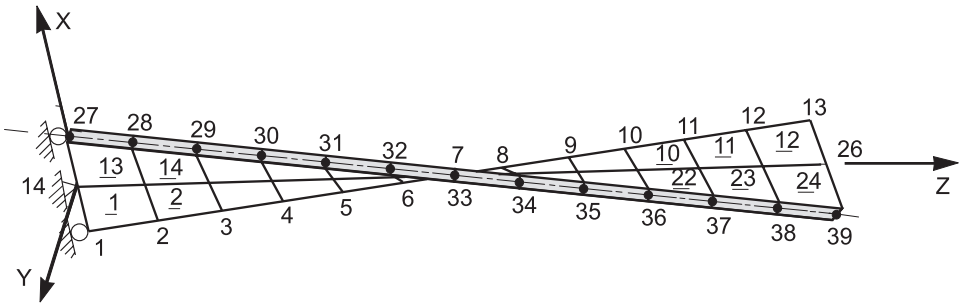
The computer model used is shown in Figure 12-1. The input data file for this example is TWSBMSHL.

Comparison of Results

The displacements at the tip in the direction of the loads are compared with theoretical results (Reference [1]) in Figure 12-2. The comparison is excellent.

Reference

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



14 Joint Number
14 Element Number

LENGTH	= 12.0
WIDTH	= 1.1
THICKNESS	= .32
TWIST (root to tip)	= 90°
MODULUS OF ELASTICITY	= $29.0X10^6$
POISSON'S RATIO	= 0.22

LOADING: UNIT FORCES AT FREE END

Figure 12-1
Twisted Beam Example

Load Direction	SAP2000 Shell Element 12x12 Mesh	Theoretical
In-plane Shear	0.005413	0.005424
Out-of-plane Shear	0.001770	0.001754

Figure 12-2
Comparison of Tip Deflections

Example 13

Beam On Elastic Foundation — Static Loads

Description

This is a simply supported beam on an elastic foundation. Half of the beam is modeled as shown in Figure 13-1. The geometry and the loads are the same as used in Reference [1].

Significant Options of SAP2000 Activated

- Plane element analysis using the plane stress option
- Spring supports representing elastic foundations

Input Data

The computer model used is shown in Figure 13-1. Pound-inch units are used. Nine-node Plane elements with the plane stress option are used. Half of the beam is modeled using 10 elements and symmetry is utilized to obtain the boundary conditions. Springs are used to model the elastic foundation.

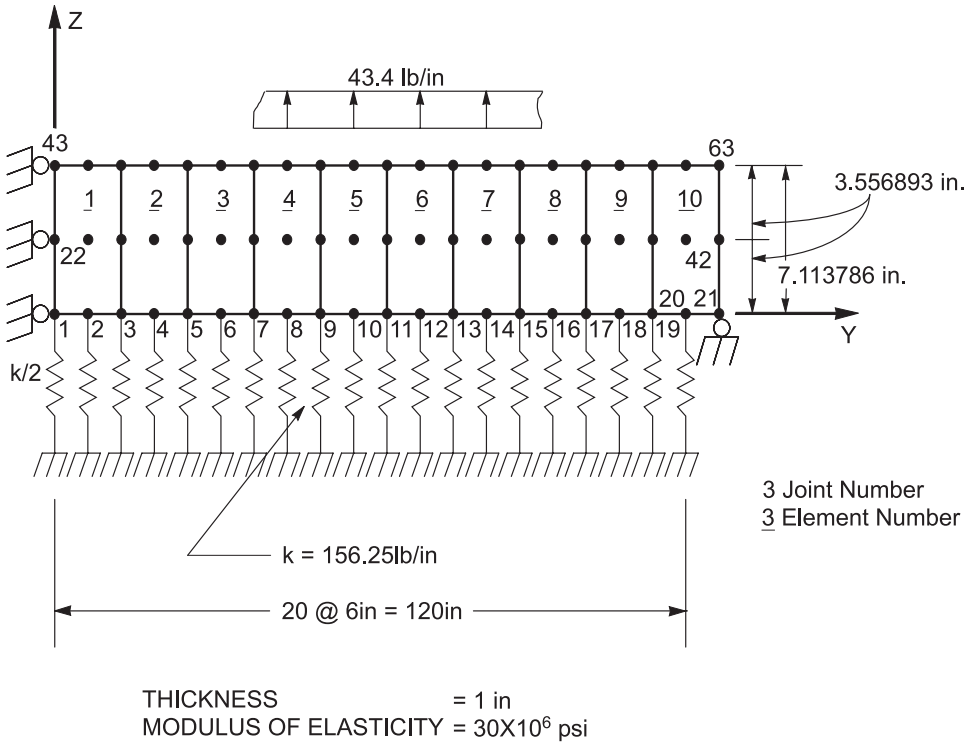
The input data file for this example is BEAMONFN.

Comparison of Results

The transverse displacements along the center of the beam and the maximum bending stress (using the average of the values at the top and bottom fiber) are compared with theoretical results (Reference [1]) in Figure 13-2. The comparison is good.

Reference

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



THICKNESS = 1 in
MODULUS OF ELASTICITY = $30X10^6$ psi

Figure 13-1
Beam on Elastic Foundation Example

Axial Station	Transverse Displacements		Maximum Bending Stress	
	SAP2000	Theoretical	SAP2000	Theoretical
0	1.0458	1.0453		
6			18029	18052
12	1.0336	1.0331		
18			17751	17773
24	0.9973	0.9967		
30			17183	17206
36	0.9373	0.9367		
42			16304	16327
48	0.8546	0.8541		
54			15082	15106
60	0.7507	0.7502		
66			13476	13501
72	0.6275	0.6270		
78			11436	11462
84	0.4874	0.4870		
90			8902	8930
96	0.3335	0.3331		
102			5810	5839
108	0.1695	0.1693		
114			2089	2119
120	0.0002	0.0000		

Figure 13-2
Comparison of Results

Example 14

Rectangular Plate — Static Loads

Description

This is a rectangular plate as shown in Figure 14-1. This problem is solved using two different aspect ratios: one square plate of 2 x 2, and another rectangular plate of 2 x 10. Two Load cases are used: Load case 1 for a concentrated load at the center and Load case 2 for uniform load. Also, the problems are solved once with the edges clamped, and once with the edges simply supported. The problem is modeled using both Shell and Solid elements. The geometry, properties and loads used are those suggested in Reference [1].

Significant Options of SAP2000 Activated

- Plate bending analysis using Shell elements
- Plate bending analysis using Solid elements
- Static load analysis

Input Data

The computer model using Shell elements is shown in Figure 14-1. For both types of elements, a 6 x 6 mesh is used on a quarter of the plate, with symmetry conditions applied to represent the remainder of the plate. Incompatible bending modes are included for the Solid element solutions.

The input data file for the model using Shell elements with clamped edges is RCPLTSHL. The input data file for the model using Solid elements with simple supports is RCPLTSOL. Both are for a rectangular (2 x 10) plate.

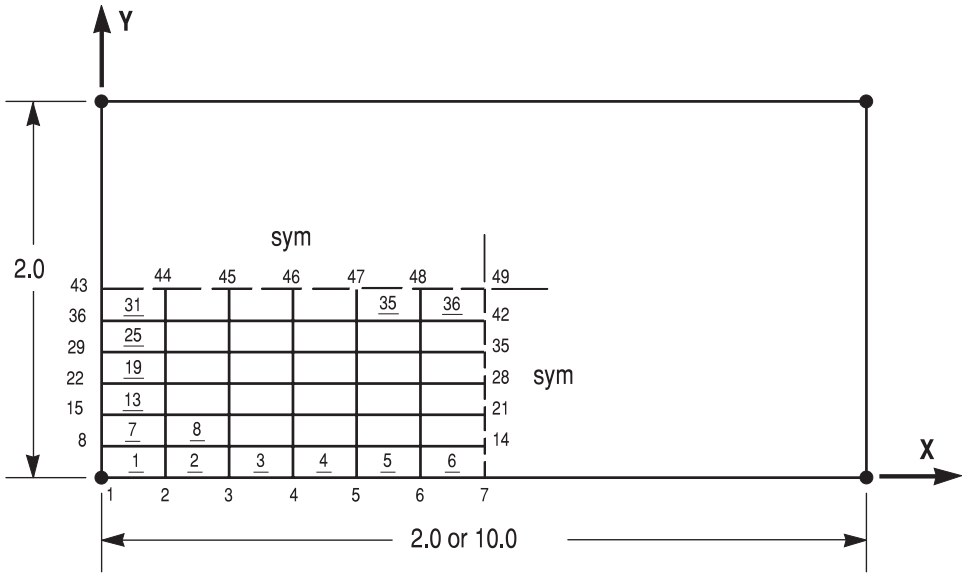
Two different thicknesses are used in models using Solid elements. One is 100 times thicker than the model using Shell elements, the other is 1000 times thicker. The loads for each model are inversely proportional to the cube of the thickness so that all displacements are of the same order of magnitude.

Comparison of Results

The central deflection results for the various boundary conditions, loading conditions, and element types are compared in Figure 14-2 with the theoretical results. The comparison is good. The thicker model using SOLID elements gives better results because of the improved aspect ratio.

Reference

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



6 Joint Number
6 Element Number

THICKNESS (SHELL ELEMENT) = 0.0001
THICKNESS (SOLID ELEMENT) = 0.01 or 0.1
MODULUS OF ELASTICITY = 1.7472×10^{7}
POISSON'S RATIO = 0.3

LOADING: UNIFORM LOAD = 1.0×10^{-4}
CENTRAL LOAD = 4.0×10^{-4}

BOUNDARIES: SIMPLY SUPPORTED OR CLAMPED

Figure 14-1
Rectangular Plate Example

Loading	Boundary Condition	Aspect Ratio (b/a)	SAP2000			Theoretical
			0.0001 Thick Shell Element	0.01 Thick Solid Element	0.1 Thick Solid Element	
Uniform	Simply Supported	1.0	4.061	3.175	4.084	4.062
		5.0	12.92	9.37	12.91	12.97
	Clamped	1.0	1.29	0.83	1.30	1.26
		5.0	2.60	1.83	2.61	2.56
Concentrated	Simply Supported	1.0	11.77	8.76	11.89	11.60
		5.0	17.74	6.40	15.97	16.96
	Clamped	1.0	5.76	3.44	5.87	5.60
		5.0	7.80	1.78	6.39	7.23

Figure 14-2
Comparison of Deflection at Center

Example 15

Cantilever Plate — Eigenvalue Problem

Description

This is a square cantilever plate and the first five eigenvalues are computed. The plate is shown in Figure 15-1.

Significant Options of SAP2000 Activated

- Shell elements for plate bending analysis
- Eigenvalue analysis
- Large capacity analysis — over 4900 dynamic degrees of freedom

Input Data

Three different mesh sizes are used: a 10 x 10 mesh, a 19 x 19 mesh, and a 40 x 40 mesh. The computer model used for the 19 x 19 mesh is shown in Figure 15-1. Kip-inch units are used. Unit mass for the plate is specified to generate the mass matrix.

The input data file is PLATE for the 19 x 19 mesh and LARGEPLT for the 40 x 40 mesh.

Comparison of Output

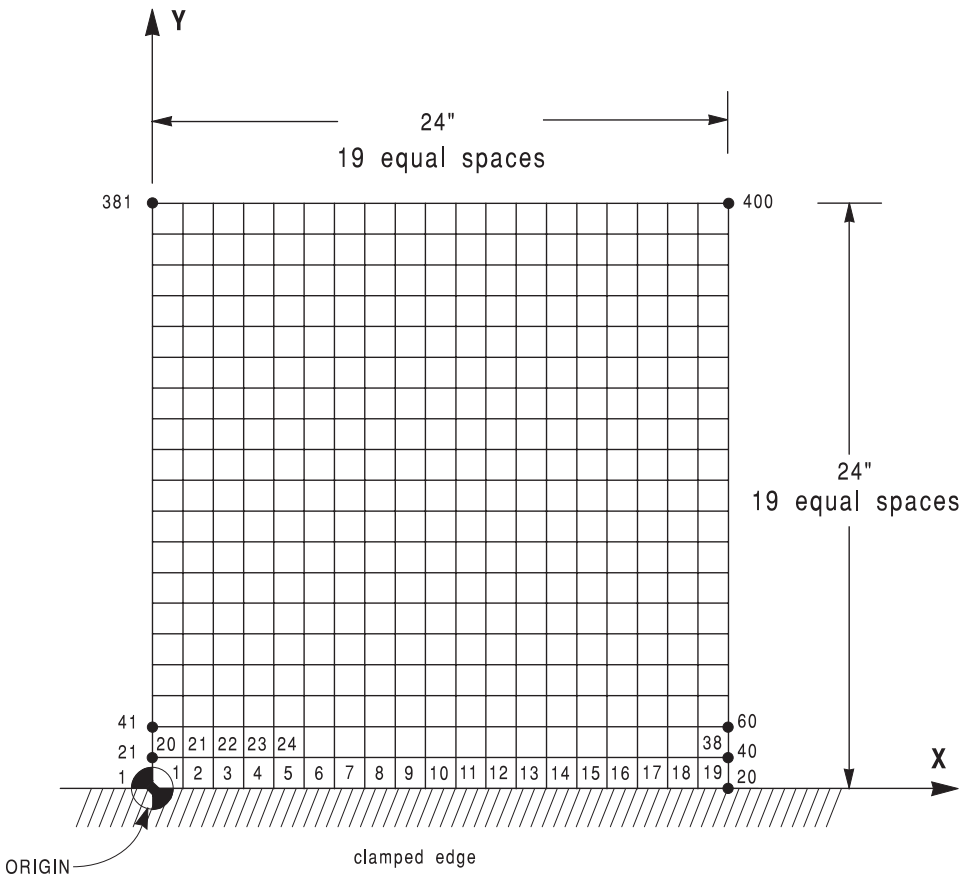
Reference [1] gives results for the first five natural frequencies of a square cantilever plate obtained by using the Ritz method with beam mode shapes. A comparison of the SAP2000 results with the Reference [1] results is given in Figure 15-2.

The comparisons of the periods of vibration for the first, third and fourth modes are excellent; however, the comparisons of the periods of vibration for the second and fifth modes are not as good because the results of Reference [1] involve approximating plate mode shapes with beam mode shapes.

The SAP2000 results using three different mesh sizes are very close.

Reference

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



PLAN

PLATE 24"x 24"x 1"
MODULUS OF ELASTICITY = 29500 ksi
MASS DENSITY = 0.49/1728/386.4 kip-sec^2 /in/in^3

Figure 15-1
Cantilever Plate Example

Mode	SAP2000			Reference [1]*
	40 x 40 MESH	19 x 19 MESH	10 x 10 MESH	
1	0.01779	0.01781	0.01787	0.01790
2	0.00647	0.00648	0.00654	0.00732
3	0.00284	0.00285	0.00288	0.00292
4	0.00221	0.00223	0.00228	0.00228
5	0.00186	0.00187	0.00190	0.00201

** Using Ritz's method with beam mode shapes*

Figure 15-2
Comparison of Natural Periods of Vibration

Example 16

Scordelis-Lo Roof — Static Loads

Description

This is a reinforced concrete single span cylindrical roof as shown in Figure 16-1. The roof is to be analyzed for gravity loads. The problem solved is the same as suggested in Reference [1].

Significant Options of SAP2000 Activated

- Three-dimensional shell analysis
- Gravity load analysis
- Large capacity analysis — 7991 static degrees of freedom

Input Data

Two different mesh sizes are used for this problem: one, a 6 x 6 mesh used on a quarter of the roof as shown in Figure 16-1; another, an 18 x 72 mesh used on half the roof. Pound-foot units are used. Symmetry boundary conditions is used to represent the omitted parts of the structure. Unit weight is specified to apply the uniformly distributed loading.

The input data file for the 6 x 6 mesh is ROOF. The input data file for the 18 x 72 mesh is LARGEROF.

Comparison of Results

The theoretical vertical deflection at the center of the free edge is 0.3086, and Reference [1] suggests a value of 0.3024 for comparison of finite element behavior. SAP2000 gives a value of 0.3068 for the 6 x 6 mesh and 0.3012 for the 18 x 72 mesh. The comparison is excellent.

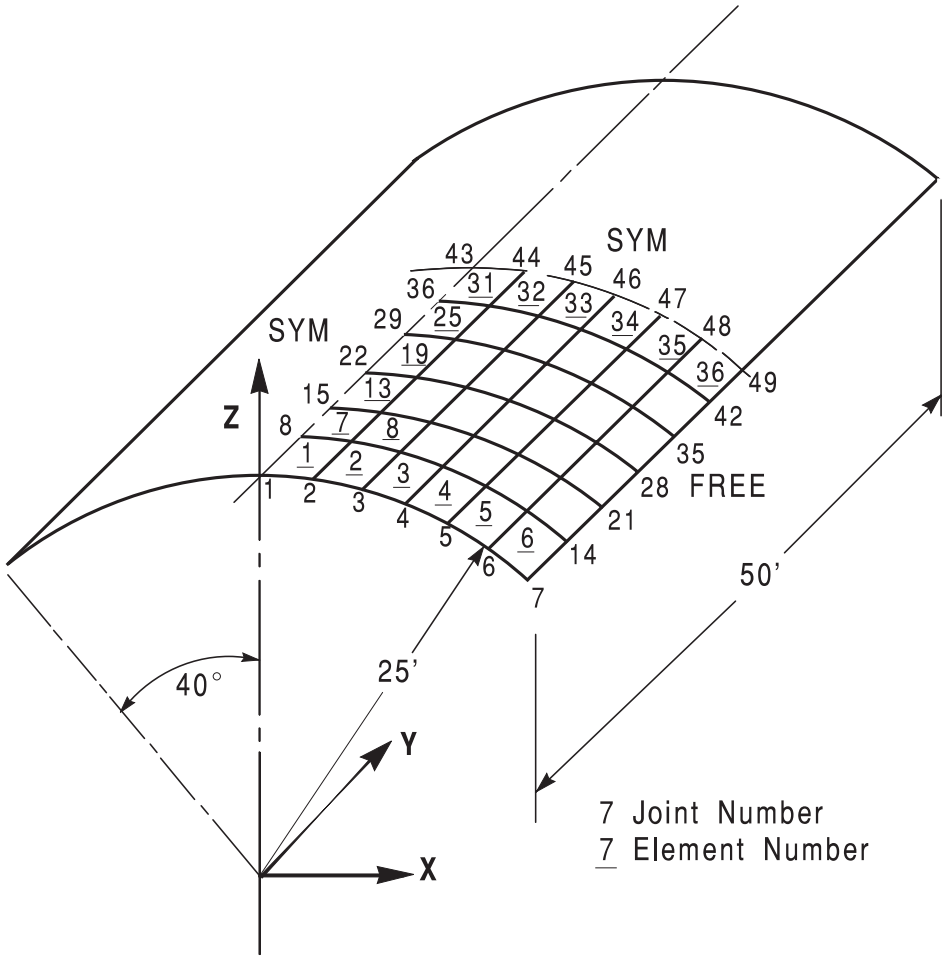
A comparison of SAP2000 displacement and bending moment results using the 6 x 6 mesh, and theoretical results provided in Reference [2] using the theory presented in Reference [3], is given in Figures 16-2 and 16-3. The theoretical results are measured from the figures given in Reference [2]. Nodal averages are used for the SAP2000 bending moments shown. The comparison is good.

References

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

2. **Zienkiewicz, O. C.**
   The Finite Element Method, McGraw-Hill, 1977.

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



THICKNESS = 3 in
MODULUS OF ELASTICITY = 4.32×10^8 psf
POISSON'S RATIO = 0.0
GRAVITY LOAD = 90 psf
(UNIFORM ON SURFACE AREA)

BOUNDARY CONDITION: SIMPLY SUPPORTED
ON CURVED EDGES

Figure 16-1
Scordelis-Lo Roof Example

Location	Axial Deformations at Support		Vertical Displacements at Central Section	
	SAP2000 6x6 Mesh	Theoretical	SAP2000 6x6 Mesh	Theoretical
0°	0.0000	0.0004	0.046	0.045
6.67°	0.0005	0.0009	0.031	0.027
13.33°	0.0018	0.0020	-0.013	-0.018
20.00°	0.0029	0.0030	-0.078	-0.082
26.67°	0.0024	0.0021	-0.155	-0.155
33.33°	-0.0017	-0.0016	-0.234	-0.241
40.00°	-0.0118	-0.0120	-0.307	-0.309

Location	Transverse Moments at Central Section		Twisting Moments at Support	
	SAP2000 6x6 Mesh	Theoretical	SAP2000 6x6 Mesh	Theoretical
0°	-2099	-2090	-91	0
6.67°	-1978	-2000	-359	-380
13.33°	-1614	-1620	-698	-670
20.00°	-1045	-1000	-982	-1000
26.67°	-408	-430	-1183	-1240
33.33°	37	100	-1248	-1290

Figure 16-2
Comparison of Results

Example 17

Hemispherical Shell — Static Loads

Description

This is a hemispherical shell loaded by point loads at the edge as shown in Figure 17-1. The problem solved is identical to the one suggested in Reference [1].

Significant Options of SAP2000 Activated

- Three-dimensional shell analysis

Input Data

The computer model used is shown in Figure 17-1. An 8 x 8 mesh of Shell elements is used on a quarter of the hemispherical shell. Symmetry is used to define the boundary conditions. A single restraint in the vertical direction is applied at the center of the free edge to provide stability.

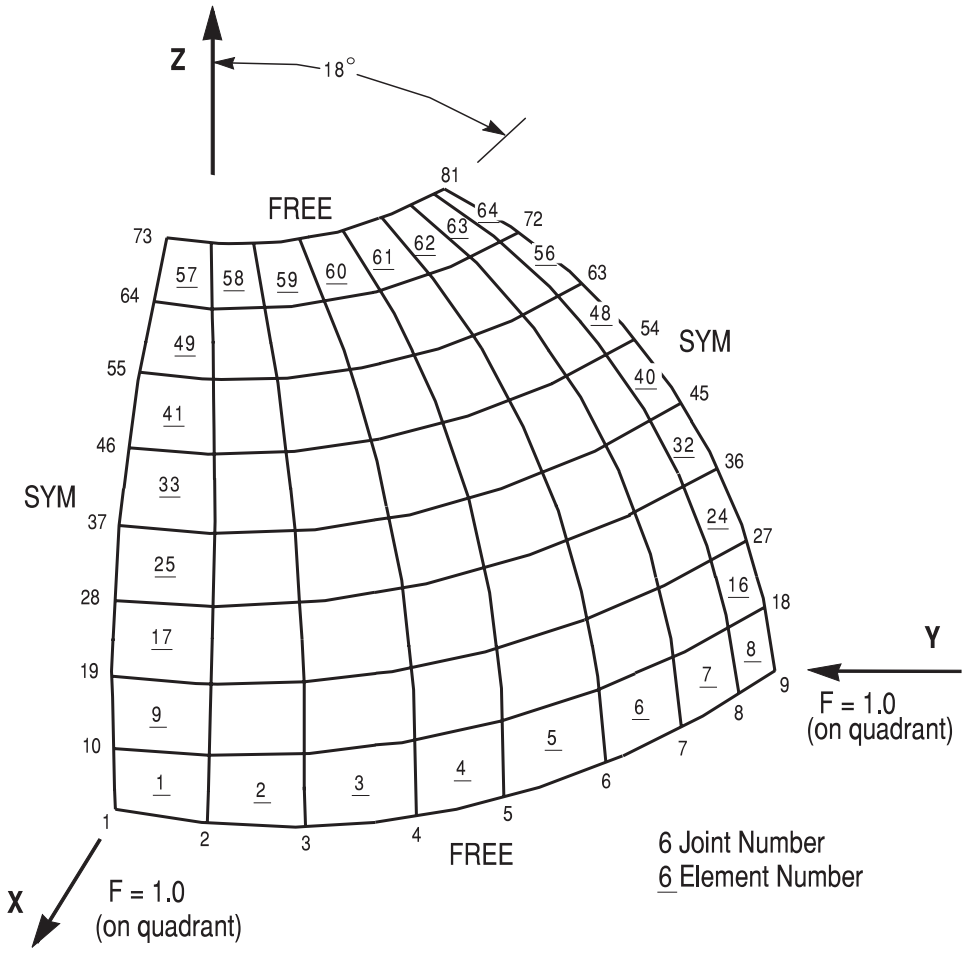
The input data file for this example is SHELL.

Comparison of Results

The theoretical lower bound for the displacement under the load in the direction of the load is 0.0924, where the hole at the center is not present. Reference [1] suggests a value of 0.094 for comparison of results. The SAP2000 solution gives a value of the displacement under the load of 0.0937. The comparison is excellent.

Reference

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



RADIUS = 10.0
THICKNESS = 0.4
MODULUS OF ELASTICITY = 6.825×10^7
POISSON'S RATIO = 0.3

LOADING: CONCENTRATED FORCES AS SHOWN

Figure 17-1
Hemispherical Shell Example

Example 18

Portal with P-delta

Description

These are two two-dimensional, one story, one bay, portal frames fixed at the base as shown in Figure 18-1. One frame carries a central load of 1000 pounds and the other carries an eccentric load of the same magnitude. The critical buckling load for the frame in the symmetrical, non-sway mode is 6082 pounds ($P_{cr} = \frac{2.55\pi^2 EI}{h^2}$) and in the asymmetrical, sway mode is 1784 pounds ($P_{cr} = \frac{.748\pi^2 EI}{h^2}$). The purpose of this example is to verify the adequacy of the P-Delta algorithm in SAP2000 for these type of problems. Theoretical results for these problems can be derived using the stability function approach. Tables for stability functions can be obtained from References [1].

Significant Options of SAP2000 Activated

- P-Delta analysis of frameworks

Input Data

The two frames are modeled together in SAP2000 using Frame elements. The loading is applied as span loads to the beams of the structures. Pound-inch units are used.

The theoretical results are computed assuming no axial deformation and no shear deformation in the members. To duplicate this behavior a large axial area is specified for the elements and the shear areas are defaulted as zero (program interprets this to mean that the shear deformation is to be ignored.) It is important to note that the axial deformations of the members were not forced to be zero by restraining or constraining the joints. Setting the axial deformation to be zero in this manner also sets the axial load in the member to be zero. This then completely eliminates the P-Delta effect from that member.

For the P-Delta analysis the default value of the relative displacement tolerance (.001) is used and the maximum number of iterations is set at 10 even though the solution converges at a much lower number of iterations. All loads are in a single Load case and this is used as the P-Delta load combination.

The input data for this example is PORTALPD.

Comparison of Results

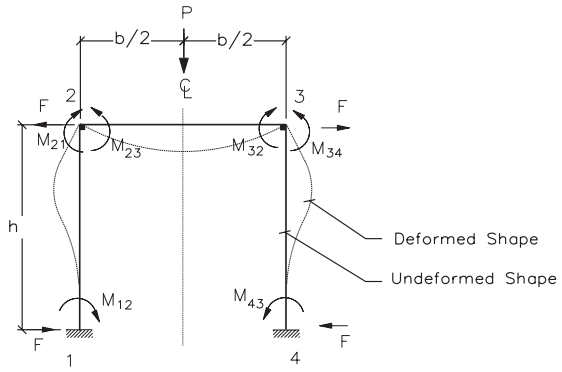
The lateral displacement, joint rotation and member end-moment results for the two load configurations are compared in Figure 18-2 with theoretical results and with results for a model in which the P-Delta effect was ignored. The agreement between the SAP2000 and theoretical results is excellent.

Theoretical values are obtained using the slope-deflection method of analysis by including the effect of axial loads in members. In this case the stiffness and carry-over factor of each member, instead of being constant, are functions of the axial load in the member. These functions are tabulated in Reference [1].

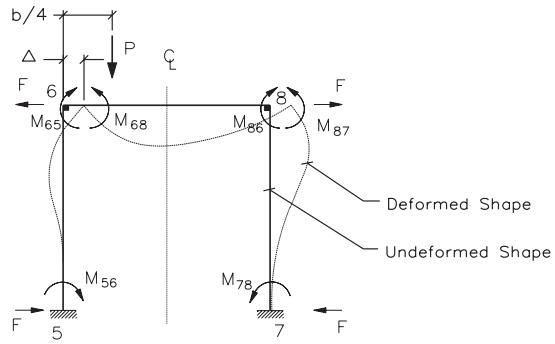
It is interesting to study the effect of P-Delta for these frames. In the symmetrical frame (central load), the decrease in the stiffness of the columns due to compressive axial loads causes a decrease in the beam end moments (top column moments) but increases the bottom column moments. For the asymmetrical frame (eccentric load), the same effect is seen in the column that carries the larger axial load. However, the column with the lesser axial load is now comparatively stiffer and attracts a larger moment at each end when P-Delta effects are considered.

Reference

1. **Livesley, R. K., and Chandler, D. B.**
Stability Functions for Structural Frameworks, Manchester University Press, UK, 1956.



(a) Deformation Under Central Load



(b) Deformation Under Eccentric Load

TYPICAL PROPERTIES :

SPAN, b	= 100 in
HEIGHT, h	= 100 in
AREA, A	= 1 in^2
MOMENT of INERTIA, I_{22},I_{33}	= 1/12 in^4
MODULUS of ELASTICITY, E	= 29x10^6 psi

Figure 18-1
Portal Frame Example

QUANTITY	With P-Delta		Without P-Delta
	SAP2000	Theoretical	SAP2000 & Theoretical
Rotation Joint 2 (θ_2)	0.09178	0.09192	0.08620
Moment Joint 1 (M_{12})	4589.1	4606.6	4166.7
Moment Joint 2 (M_{21})	8260.4	8254.0	8333.3
Shear Force (F)	128.5	128.6	125.0

Frame Under Central Load

QUANTITY	With P-Delta		Without P-Delta
	SAP2000	Theoretical	SAP2000 & Theoretical
Lateral Displacement (Δ)	1.894	1.893	1.385
Rotation Joint 6 (θ_6)	0.1014	0.1013	0.0924
Rotation Joint 8 (θ_8)	0.0367	0.0367	0.0369
Moment Joint 5 (M_{56})	2550.9	2544.9	2455.4
Moment Joint 6 (M_{65})	6183.6	6088.6	6919.6
Moment Joint 7 (M_{78})	4503.5	4456.9	3794.6
Moment Joint 8 (M_{87})	6124.9	6153.0	5580.4
Shear Force (F)	101.6	101.4	93.75

Frame Under Eccentric Load

Figure 18-2
Comparison of Results for a Planar Rigid Portal Frame

Example 19

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 Nllink Gap elements. The structure experiences pounding due to ground motion. El Centro 1940 (N-S) record is used in the nonlinear time history analysis.

The geometry of the structure is shown in Figure 19-1.

Significant Options of SAP2000 Activated

- Two-dimensional frame analysis
- Use of uniaxial Nllink gap elements
- Nonlinear time-history analysis

Input Data

A Diaphragm constraint is applied to each floor of the two frames. Kip-inch units are used.

The joints at column line 3 are connected to the corresponding joints at column line 4 by Nllink Gap elements. The local 1 axes of these elements are in the global X direction, and gap properties are specified for the local 1 direction. The opening dis-

placement for the gaps is 0.25 inches, corresponding to the distance between the outer faces of the buildings.

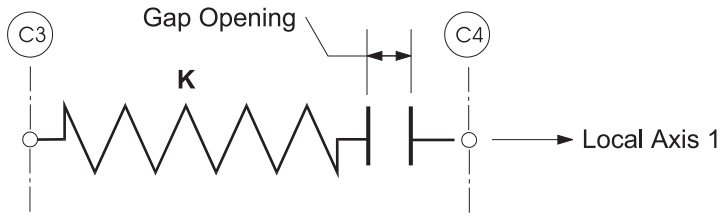
The nonlinear gap stiffness was set to 1000 k/in, an estimate of the stiffness of a tributary region around the point of contact. The linear effective stiffness, used for calculating the modes, was set to zero, since the gap elements are normally open. Care should be taken not to use values of nonlinear stiffness or linear effective stiffness that are too large relative to the stiffness of connecting elements. Overly large values of stiffness can cause numerical sensitivity, which can reduce the accuracy and the efficiency of the solution.

The modes were calculated using Ritz vectors, with the ground acceleration and the nonlinear deformation loads used as starting vectors. All eleven modes of the structure were requested.

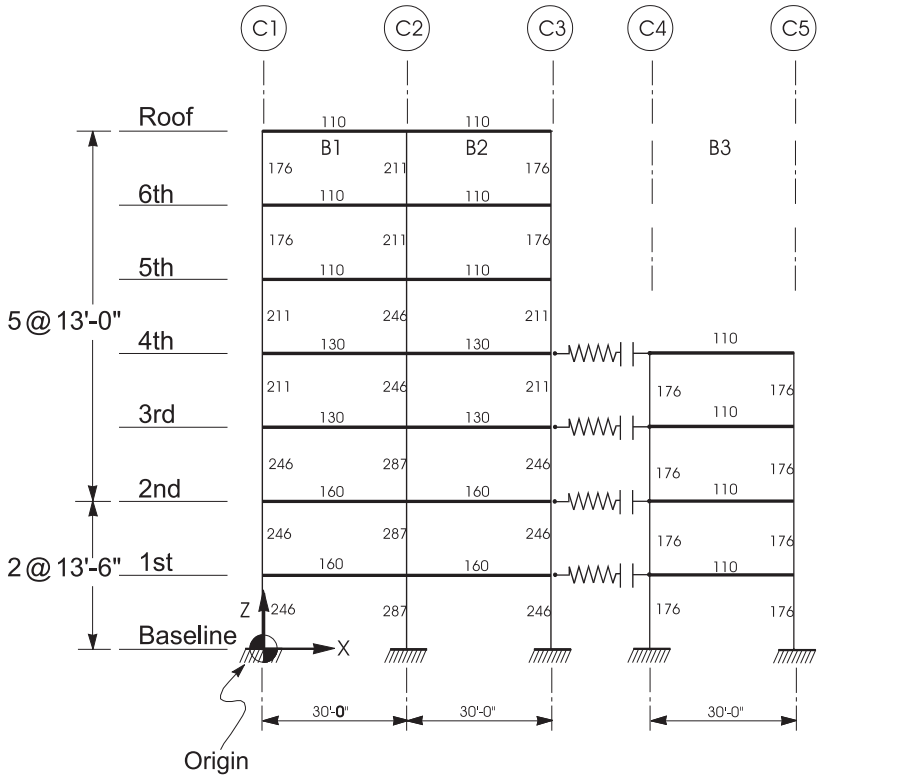
The input data for this example is POUND. The time history of the ground acceleration is given in file ELCN-THU

Comparison of Results

This example is included as a sample only. A typical results plot is shown in Figure 19-2. It shows the variations of displacement of Column lines 3 and 4 and the link force at Story level 4. It can be clearly seen 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 most part the pounding has the effect of keeping the buildings in phase. For display purposes the displacements are scaled up by a factor of 100.



NIlink Gap Element



Elevation

Figure 19-11
Geometry of Two Pounding Frames

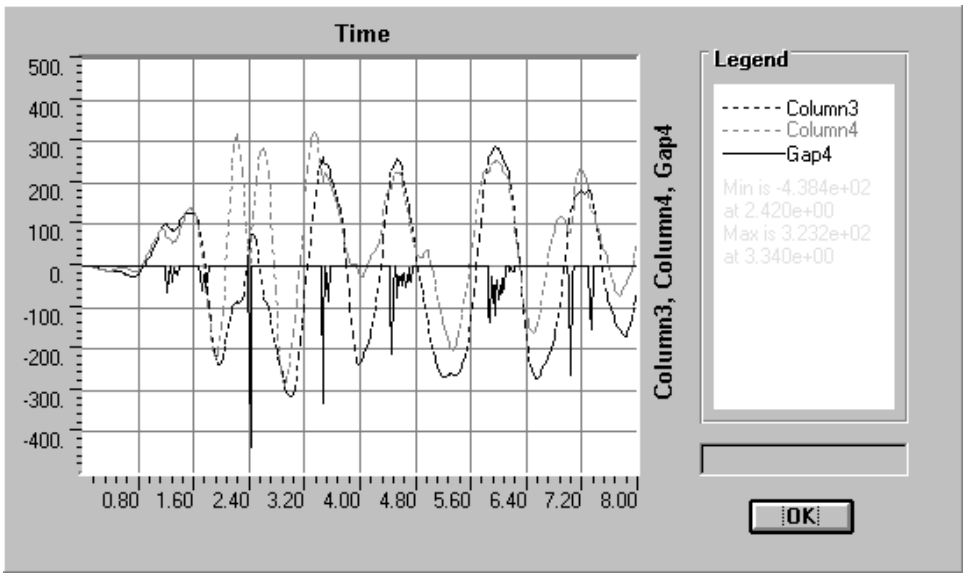


Figure 19-2
Displacements of Pounding Columns and Pounding Force at Story 4

Example 20

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 20-1.

Significant Options of SAP2000 Activated

- Three-dimensional frame analysis
- Use of biaxial Nllink friction-pendulum elements
- Nonlinear quasistatic time-history analysis
- Nonlinear time-history analysis

Computer Model

The structure is modeled as a reinforced concrete frame with 9 column lines and 12 bays. Kip-inch units are used. The modulus of elasticity is taken to be 3000 ksi. The self-weight of the concrete is taken as 150 pcf.

The floor slab is taken to be 8” thick covering all the specified floor bays at the base and the 1st story levels. At the second 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 20-1. A diaphragm constraint is applied to each story. The floor slab is modeled with Shell Membrane elements. These are used for the purpose of generating mass only; their stiffness is ignored due to the diaphragm Constraints.

Friction-pendulum type base isolators of the type described in Reference [1] are modeled using the SAP2000 Nllink Isolator2 elements.

The isolator properties are defined as follows:

Stiffness in direction 1 (vertical)	1E3
Stiffness in directions 2 and 3 (horizontal)	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 single Load case is defined that models the self-weight of the structure. This will be applied before the seismic analysis in order to generate the proper frictional response in the isolators. This self-weight must be applied as part of the time-history analysis, as described below.

Seventeen Ritz vectors are requested for the modal analysis, using the two horizontal ground accelerations, the gravity load, and all the nonlinear deformation loads as the starting load vectors. There are 27 nonlinear deformation loads, one for each of the three translational degrees of freedom for the nine isolators. However, only three of the 18 horizontal degrees of freedom are independent due to the diaphragm constraint. The program will automatically discard 15 dependent starting load vectors. Thus the total number of independent starting load vectors is 15. By listing the two ground accelerations as first, two Ritz vectors are generated for these two starting load vectors, and one for each of the remaining 13, for a total of 17 modes.

It is very important that the nonlinear deformation loads be used as starting load vectors in order to capture nonlinear behavior. In this case, the axial load in each isolator must be adequately modeled since the shear force depends strongly upon it.

Two time-history analyses are performed. The first history applies the self-weight quasistatically. This history consists of 10 time steps of one second, for a total time of 10 seconds. The load is increased linearly to full value for the first five seconds,

and allowed to come to equilibrium for the remaining five seconds. A damping value of 99% is used for all modes to prevent vibration.

The second history starts from the first history, and applies the seismic acceleration. This history consists of 2000 time steps of 0.02 seconds, for a total time of 40 seconds. Zero damping is used for all modes since most energy dissipation is expected to be due to the friction properties of the isolators, with little damage occuring in the superstructure.

The input data file for this example is ISOLAT2 and the input time history files are LP-TH0 and LP-TH90.

Comparison of Results

This example is included as a sample only. No results are available for comparison. Typical plots are shown for the seismic time history. Figure 20-2 shows the time variation of input, kinetic, potential (strain), damping, and frictional energies; these energies do not start at zero since they continue from the self-weight time history. Figure 20-3 shows the time variation of the displacements of the second floor at Column line 1. Figure 20-4 shows the interaction diagram for the two shear forces in the isolator beneath Column line 1. Figure 20-5 shows the force-displacement plot for motion in the X direction of the isolator beneath Column line 1.

References

1. **Zayas, V. and Low, S.**
"A Simple Pendulum Technique for Achieving Seismic Isolation," *Earthquake Spectra*, Vol. 6, No. 2, Earthquake Engineering Research Institute, Oakland, California,1990.

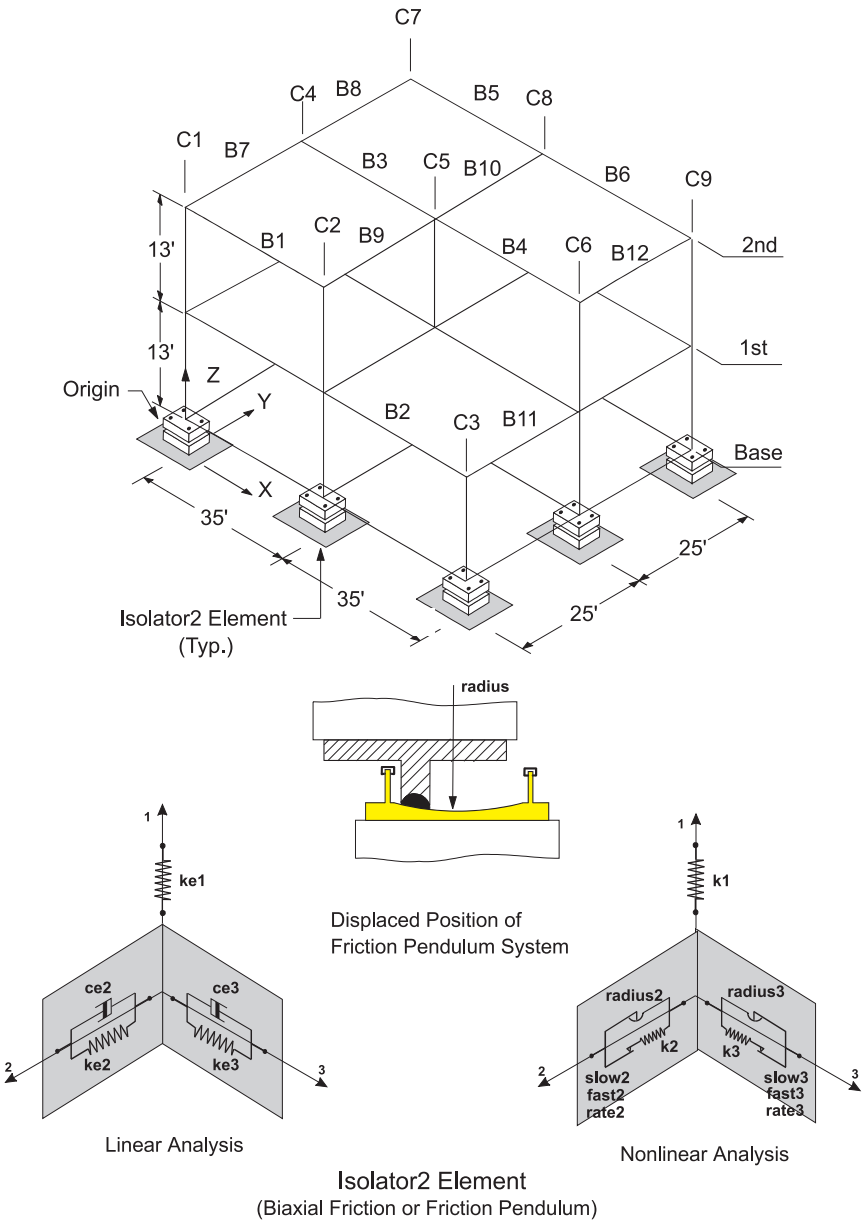


Figure 20-1
Model of Friction-Pendulum Base-Isolated 3-D Frame

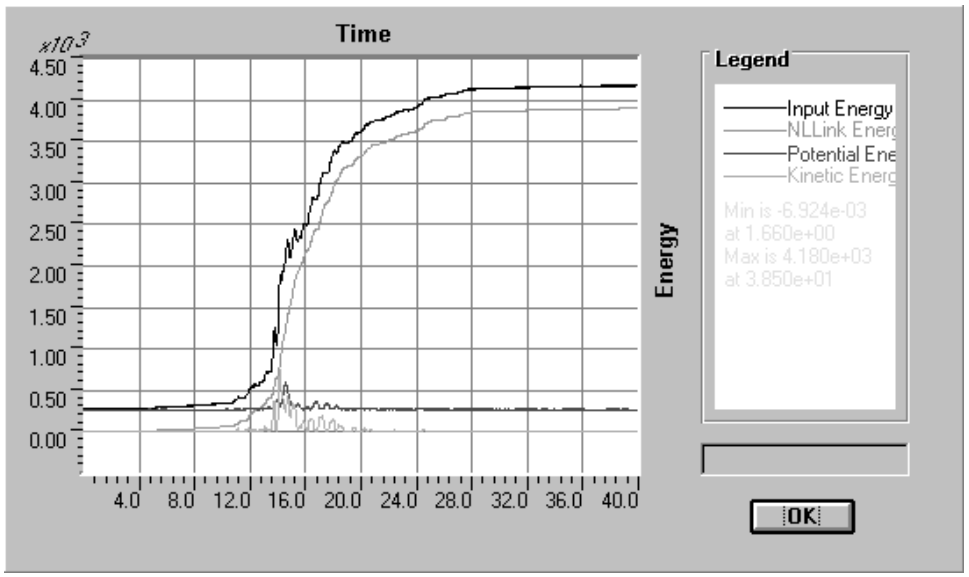


Figure 20-2
Variation of Energy during Seismic History

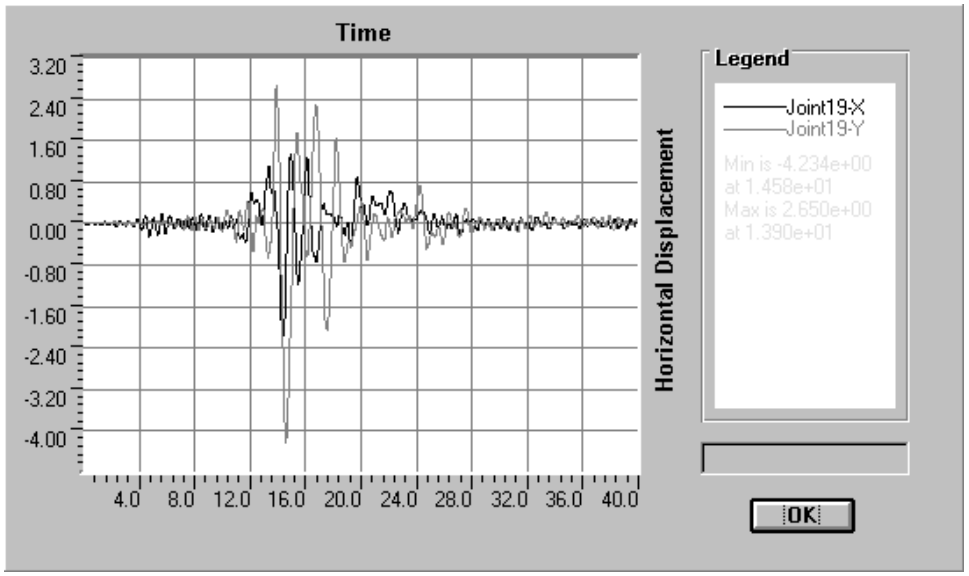


Figure 20-3
Horizontal Displacements at Top of Column 1 during Seismic History

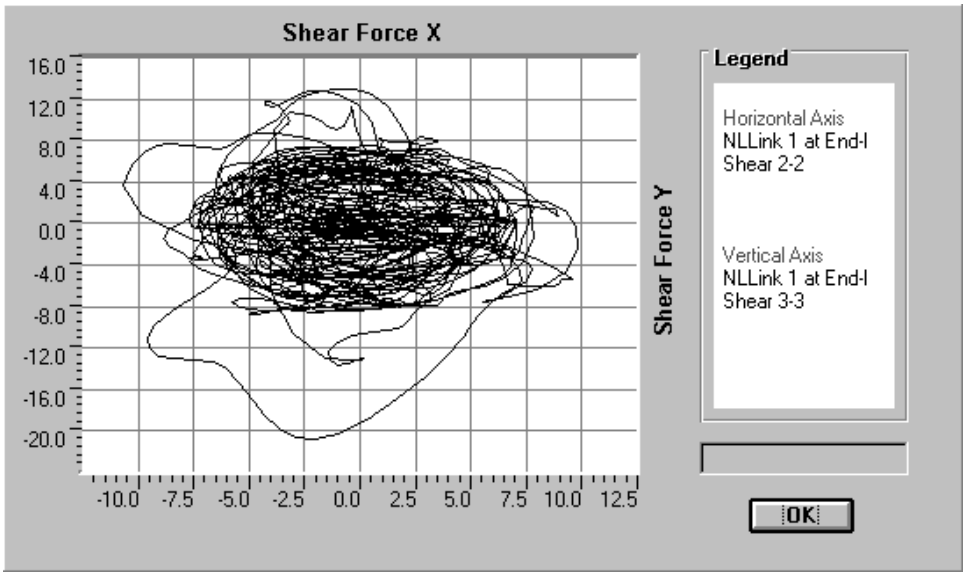


Figure 20-4
Shear Force Interaction in Isolator under Column 1 during Seismic History

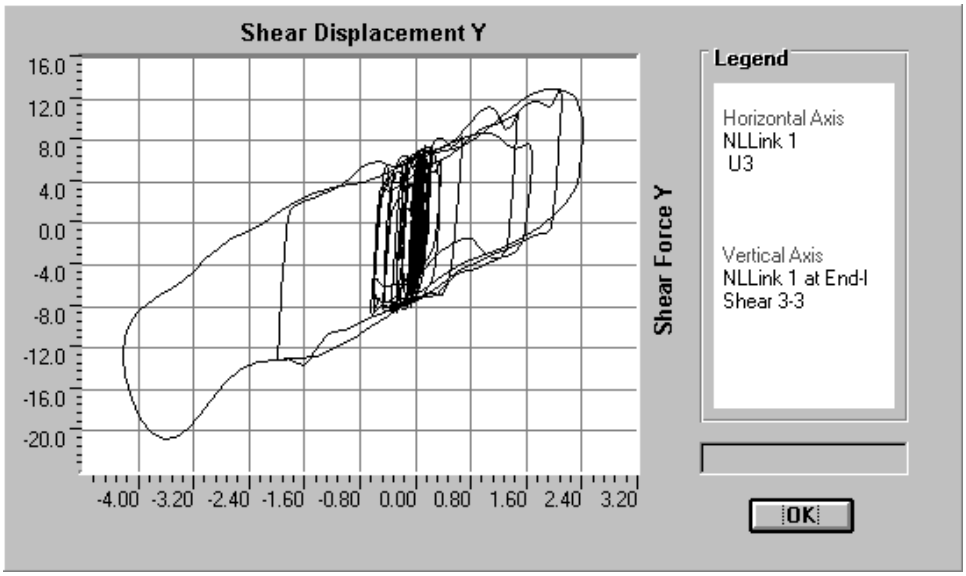


Figure 20-5
Shear Force vs. Deformation in Isolator under Column 1 during Seismic History