SOFTWARE VALIDATION TEST PLAN AND TEST RESULTS FOR SAP2000, VERSION 8.3.1

Prepared by

Luis Ibarra
Thomas Wilt

Center for Nuclear Waste Regulatory Analyses
Southwest Research Institute
San Antonio, Texas

Approved by:

12-09-2005

Asadul Chowdhury **Date**
Element Manager
Mining, Geotechnical, & Facility Engineering

CONTENTS

CONTENTS (continued)

Section Page

FIGURES

TABLES

1.0 SCOPE OF THE VALIDATION

SAP2000 (2003) is a structural analysis computer code program that is based on over 30 years of development. SAP2000 performs both static and dynamic analysis. This analysis may be either linear or include nonlinear effects such as geometric nonlinearities (e.g., P-delta and large displacement effects) as well as material nonlinearities (i.e., plastic hinge). Also within SAP2000 are a variety of load options and structural elements. Since SAP2000 contains many features, validating all of the features is beyond the scope of this report. As a result, the planned SAP2000 validation includes the features that most likely will be used during the course of the independent analyses to be carried out as part of Center for Nuclear Waste Regulatory Analyses (CNWRA) regulatory review process. The specific features to be tested are listed for each of the examples to be presented in Section 6.

Validation for SAP2000 is accomplished by comparing the results of the created structural models with hand calculation results or with results presented in the literature.

2.0 REFERENCES

Cook, R.D. and W.C. Young. "Advanced Mechanics of Materials." Macmillan Publishing Company. 1985.

Harris, C.M. and C.E. Crede. "Shock and Vibration Handbook." McGraw-Hill Book Company. 1976.

Roark, R.J. and W.C. Young. "Formulas for Stress and Strain." 5th Edition. McGraw-Hill Book Company. 1975.

SAP2000. *Software Verification Examples, Version 8.1.2, Integrated Software for Structural Analysis and Design.* Berkeley, CA: Computers & Structures, Inc. January 2003.

SAP2000 Analysis Reference Manual, Version 8; Inc, Berkeley, California, USA, July 2002.

Timoshenko, S. and Wionowsky-Krieger. "Theory of Plates and Shells." 2nd Edition. McGraw-Hill Book Company, Inc., New York City, New York. 1959.

3.0 ENVIRONMENT

3.1 Software

The SAP2000 program Version 8.3.1, is distributed as an executable, and it is designed to run in several platforms. SAP2000 is an acquired software which is used to perform structural analysis, and the present installed version of the program is currently under configuration control (Scientific Notebook 656).

3.2 Hardware

The SAP2000 program is installed in a personal computer (PC) with a single Intel Pentium IV CPU running at 2.8 Ghz with 512 Mb of memory and local 60 Gb hard disk. The PC operates using Microsoft Windows 2000 and is identified on the CNWRA domain as Aspen.

4.0 PREREQUISITES

None.

5.0 ASSUMPTIONS AND CONSTRAINTS

None.

6.0 TEST CASES

As documented in the Scientific Notebook 656, the SAP2000 program is a software that has been controlled and released. For configuration control, staff tested several selected structural models provided by the developer: 1-022, 1-024, 2-008, 2-011, and 6-009. The developer also provides a comparison of key results with independent calculations. This information is also provided by the developer in a SAP2000 Software Verification Examples (2003). The tested structural analyses covered most of the applications that the CNWRA staff will utilize when modeling structural systems in SAP2000. The obtained results coincided with those reported in the SAP2000 documentation. Therefore, the CNWRA staff is confident that the program has been properly installed.

For SAP2000 validation, some of the structural models proposed by the developer were reproduced by CNWRA staff and the results were compared with independent hand calculations included in the SAP2000 Software Verification Examples (2003). The number of cases tested was increased to cover the basic functions of SAP2000 that are expected to be used by CNWRA staff. It is the objective of this report to show that identical results were obtained, for the selected test cases, as compared to the SAP2000 results given in the SAP2000 Software Verification Examples (2003).

6.1 Example 1-001 of SAP2000 (2003): Frame—General Loading

The geometry, as well as the boundary conditions, of the three-element frame is as shown in Figure 6-1.

Each member has the following material properties: an elastic modulus of 3,600 k/in^2, unit weight of 0.15 k/ft^3. The section properties are: cross-sectional area of 144 in^2, moment of inertia of 1,728 in^4.

This frame is subjected to a static analysis for six separate load cases consisting of different types of distributed and concentrated loads. The requirements being tested in this case are for uniformly distributed loads and joint moments and forces.

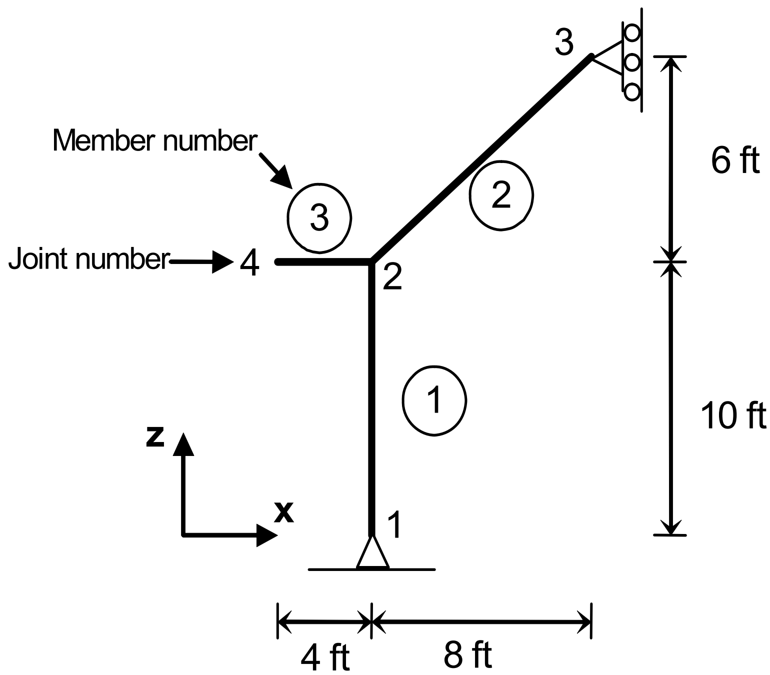


Figure 6-1. Three Element Frame

The uniformly distributed loads are of the type

- Self load (self weight = 0.15 k/ft^3)
- Uniformly distributed load in global coordinates, projected load (Figure 6-2a)
- Uniformly distributed load in frame object local coordinates (Figure 6-2b)

The SAP2000 features tested in this example are:

- Calculation and Application of
  - self weight
  - projected, uniform load

- Application of
  - uniform load in global coordinates
  - uniform load in local coordinates
  - trapezoidal and triangular distributed loads
  - applied joint forces moments

The joint moment and force cases are as shown below in Figure 6-3.

6.1.1 Test Input

The test input is located in the following subdirectory \SAP2000\Validation\Example 1-001\1-001a.SDB on the accompanying CD.

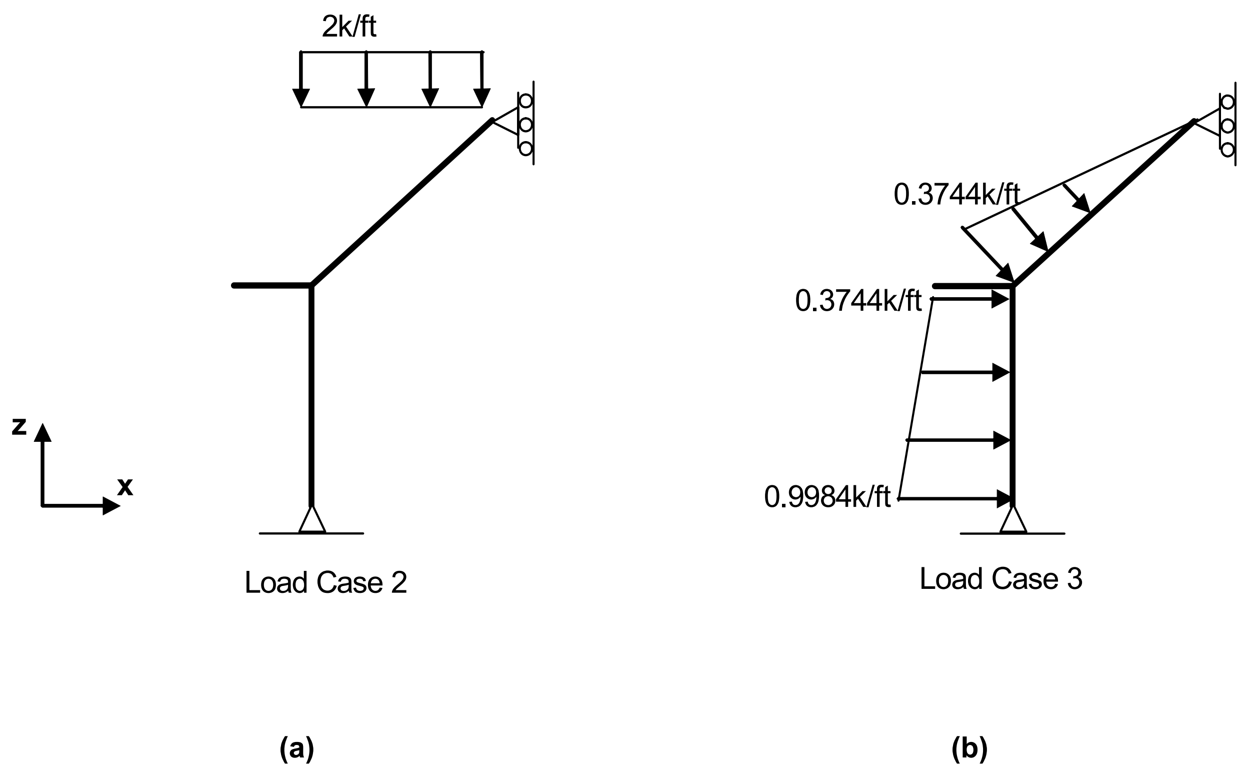


Figure 6-2. Loads in Global and Local Coordinate Systems

6.1.2 Test Procedure

The test procedure used here are the following.

- The geometry and loading cases 1, 2, 3, 4, 6, and 7 as shown in Figures 6-1 to 6-3, (based on the SAP2000 software verification manual) as described above were implemented in SAP2000.

- The SAP2000 analysis was executed and the joint displacements were obtained from the output file.

- The obtained results were compared with the hand calculations provided by the developer.

6.1.3 Test Results

The appropriate nodal displacement results (U_x, U_z) were tabulated as shown in Table 6-1. These results were compared with the results shown in the SAP2000 Software Verification Examples (2003). Note that the results match those given in the SAP2000 Software Verification Examples (2003).

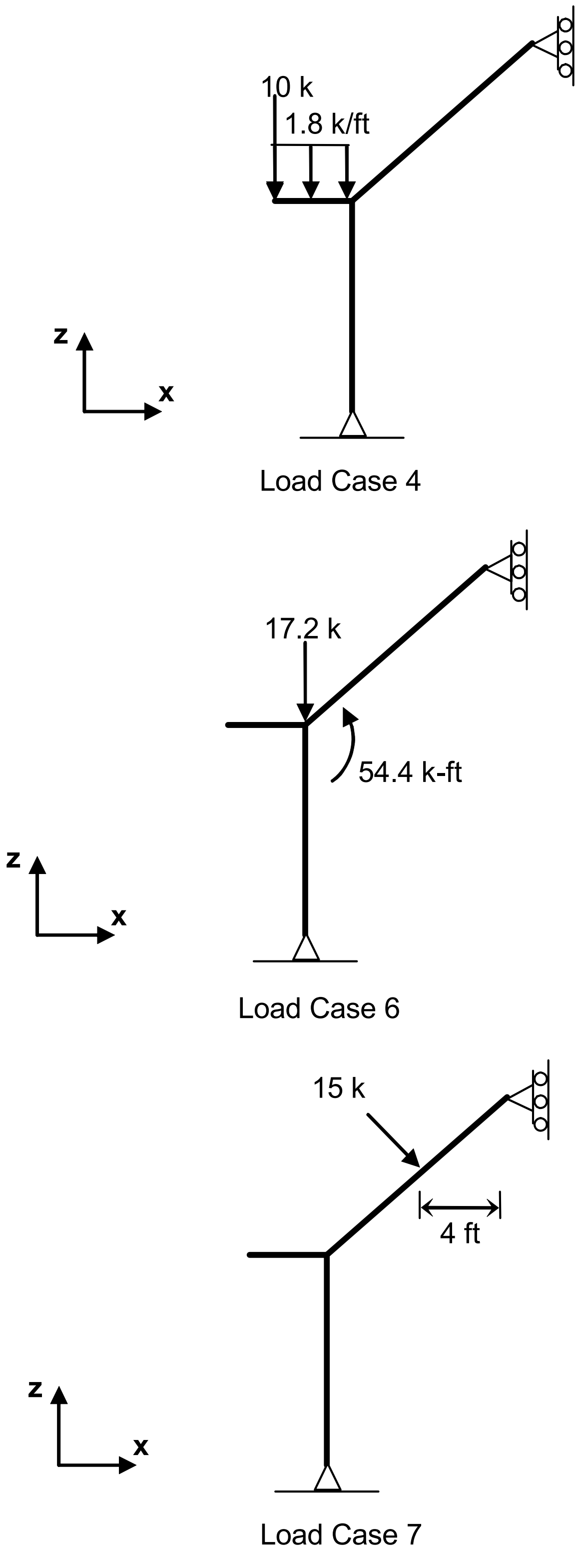


Figure 6-3. Joint Moment and Force Loads

Table 6-1. Nodal Displacement Results				
Load Case	**Output Parameter (in.)**	**SAP2000**	**Independent**	**Percent Difference**
1	U_z (joint 3)	−0.02639	−0.02639	0
2	U_z (joint 3)	0.06296	0.06296	0
3	U_z (joint 3)	0.06296	0.06296	0
4	U_z (joint 3)	−0.2963	−0.2963	0
6	U_x (joint 2)	0.11554	0.11556	−0.01
7	U_x (joint 2)	0.00651	0.00651	0

6.2 Example 1-002 of SAP2000 (2003): Frame—Temperature Loading

In this example, the various types of frame temperature loads that can be modeled in SAP2000 were tested in a cantilever beam. In SAP2000 three types of temperature loads can be applied on a frame elements: (i) an overall change in temperature with respect to a reference temperature, (ii) a temperature variation along the element length in the longitudinal direction, and (iii) a temperature gradient perpendicular to the element length.

The specific requirements tested in this case were

- The specification of reference temperature and joint patterns
- The application of:
  - temperature increase
  - transverse temperature gradient

The results of the analysis are given in terms of

- Displacements for the case of free expansion, Model A (1-002a)
- Reaction forces caused by temperature loads in the restrained cases, Model B (1-002b)

The geometry, as well as the boundary conditions, of the cantilever beam for both “Model A” (1-002b) and “Model B” (1-002b) is as shown in Figure 6-4. Each member has the following material properties: an elastic modulus of 29,000 k/in^2, coefficient of thermal expansion of 0.0000065 / °F . The section properties are: cross-sectional area of 6 in^2, moment of inertia of 4.5 in^4.

6.2.1 Test Input

The test inputs are located at \SAP2000\Validation\Example 1-002\1-002a.SDB, and 1-002b.SDB on the accompanying CD.

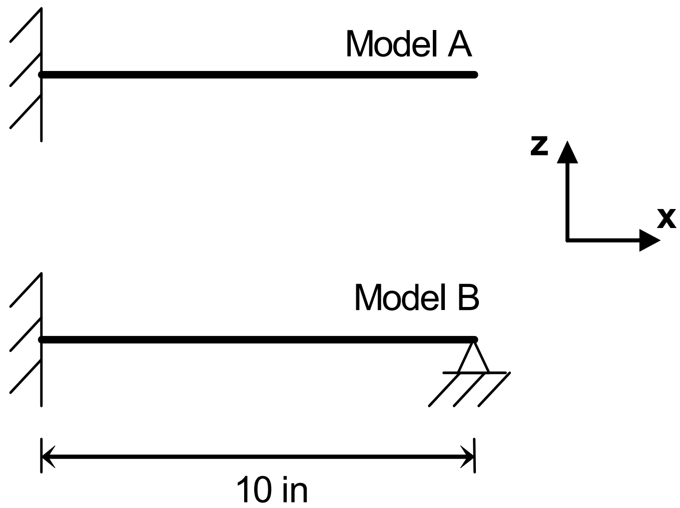


Figure 6-4. Cantilever Beam: Model A (1–002a) and Model B (1–002b)

6.2.2 Test Procedure

The specific test procedure for this example is as follows. The geometry and loading cases of Example 1-002 were implemented using SAP2000. The specific temperature load cases under consideration are

- Load Case 1: A reference temperature of 27 °C [80 °F] and a temperature load of 38 °C [100 °F] {an increase of 11 °C [20 °F]}

- Load Case 2: A variation along the x-axis of 1.1 °C [2 °F] per inch of element length {a total of 11 °C [20 °F]}

- Load Case 3: A gradient along the y-axis of 11 °C [20 °F] per inch {33 °C [60 °F] for 3 in. of section height}

The analysis was executed and joint displacements and reactions forces were obtained for Models A (1-002a) and B (1-002b), respectively. Finally, the obtained results were compared with the hand calculations provided by the developer using the appropriate equation obtained from Roark and Young (1975).

6.2.3 Test Results

The nodal displacement (U_x) and reaction force (F_x) are tabulated, as shown in Tables 6-2a and 6-2b, and compared with the independent results. Similar accuracy was obtained as given in the SAP2000 Software Verification Examples.

Table 6-2a. Model A

Load Case	Output Parameter (in.)	SAP2000	Independent	Percent Difference
1	U_x (free end)	0.0013	0.0013	0
2	U_x (free end)	0.00070	0.00065	7.6
3	U_x (free end)	−0.0065	−0.0065	0

Table 6-2b. Model B

Load Case	Output Parameter (kips)	SAP2000	Independent	Percent Difference
1	F_x (at prop.)	−22.62	−22.62	0
2	F_x (at prop.)	−11.31	−11.31	0
3	F_x (at prop.)	2.545	2.545	0

6.3 Example 1-005 of SAP2000 (2003): Frame—Displacement Loading

This example verifies the conditions of support settlement and rotation of normal supports, and spring supports as applied to a single portal frame. The first three cases presented in the SAP2000 Software Certification Examples (2003) were reproduced in this example, and compared with hand calculations provided by the developer of SAP2000.

The geometry, as well as the initial boundary conditions, of the portal frame is as shown in Figure 6-5.

Each member has the Young's modulus of 29,000 k/in^2 and the following section properties: a cross-sectional area of 144 in^2 and a moment of inertia of 1,728 in^4.

In this example, the requirements being tested are

- The settlement of the supports
- Settlement of a support that also has a linear translational spring
- Rotation of a support

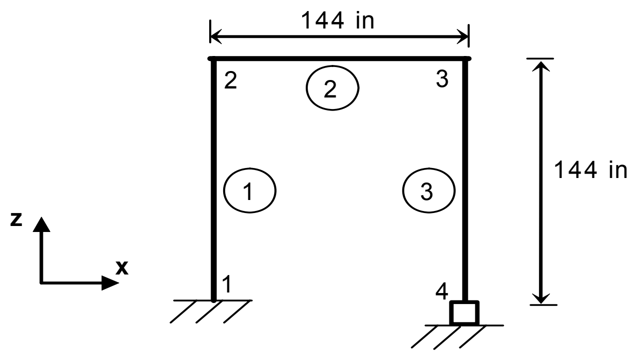


Figure 6-5. Portal Frame

6.3.1 Test Input

The test inputs are located at \SAP2000\Validation\Example 1-005\1-005a.SDB (Load Case A), 1-005b.SBD (Load Case B), and 1-005c.SDB (Load Case C) on the accompanying CD.

6.3.2 Test Procedure

The first three load cases of Example 1-005 are shown in Figure 6-6, and as given in the SAP2000 Verification manual, are implemented in SAP2000.

The analysis was executed and reaction forces were obtained for the three models. The SAP2000 results were compared with the hand calculations provided by the developer.

6.3.3 Test Results

The reaction force (F_z) and moment (M_y) obtained were tabulated, Table 6-3, and compared with the independent results as found in the SAP2000 verification manual. Similar results was obtained as those given in the SAP2000 Software Verification Examples.

6.4 Example 1-006 of SAP2000 (2003): Frame—Non-Prismatic Sections

This example tests the SAP2000 nonprismatic frame section property in a cantilever nonprismatic beam made up of four segments. In SAP2000 the axial, torsion, weight, and mass properties can vary linearly and the bending property variation may be linear, parabolic or cubic. For this validation test, three loading cases were included: (i) self-weight, (ii) an axial force applied at the end of the beam, and (iii) a moment force applied at the end of the beam. The geometry, as well as the initial boundary conditions, of the portal frame is as shown in

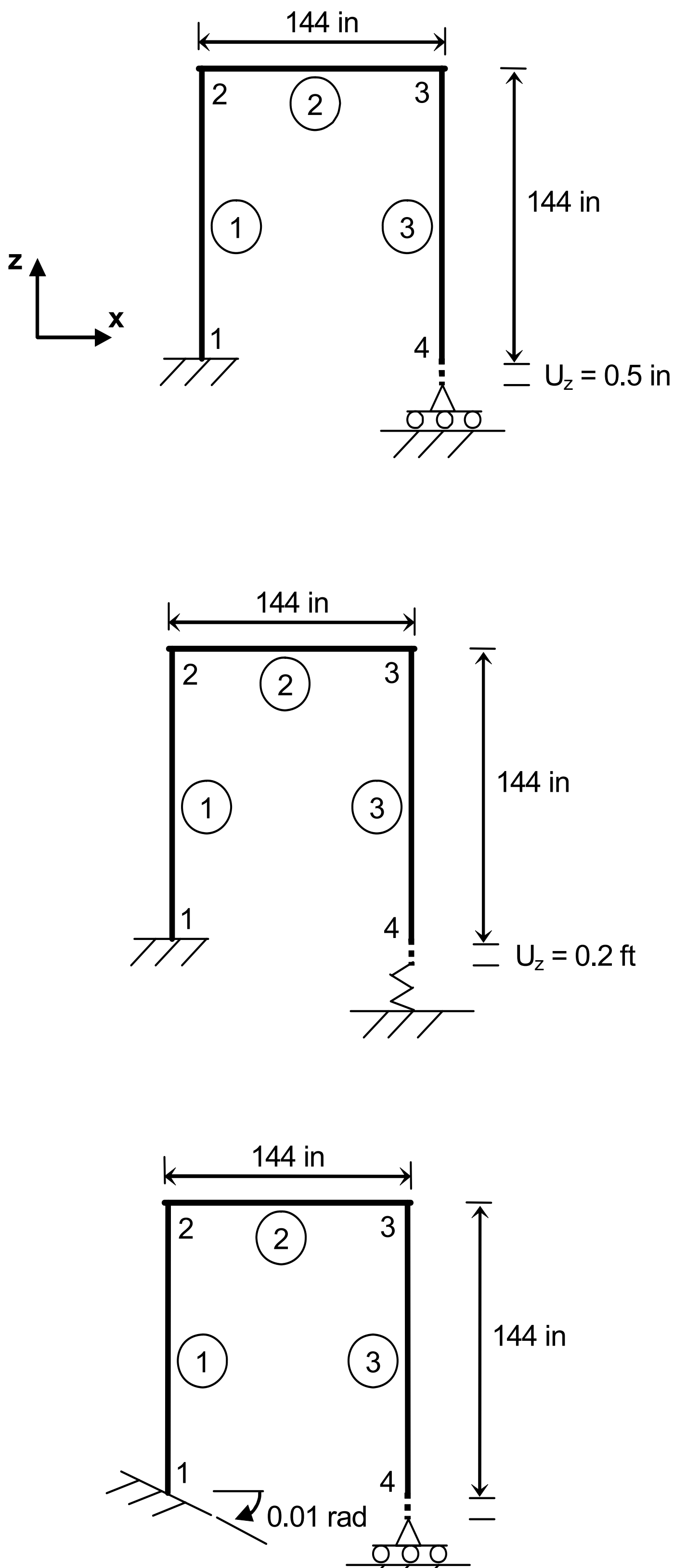


Figure 6-6. Load Cases A, B, and C

Table 6-3. Results Comparison				
Model	**Output Parameter**	**SAP2000**	**Independent**	**Percent Difference**
A. Support Settlement	F_z (joint 1) kip	6.290	6.293	−0.05
	M_y (joint 1) kip-in	−906.25	−906.25	0
B. Spring Support Settlement	F_z (joint 1) kip	1.11	1.115	0
	M_y (joint 1) kip-in	−160.490	−160.492	−0.001
C. Support Rotation	F_z (joint 1) kip	−18.120	−18.125	−0.03
	M_y (joint 1) kip-in	2610	2610	0

Figure 6-7. Each member has an elastic modulus of 3,600 k/in^2, a Poissons ratio of 0.2, a shear modulus of 1,500 k/in^2 and unit weight of 0.15 k/ft^3. The section properties for the beam (Figure 6-7) are as shown in Table 6-4.

As shown in Figure 6-7 for Segments 1, 3, and 4 the section area varies linearly and is constant for Segment 2. For Case 3. The required moment of inertia is considered to vary linearly in Segment 1, takes a constant value in Segment 2, has a parabolic variation in Segment 3 and has a cubic variation in Segment 4.

Considering the above descriptions, the requirements being tested in this case were:

- Structural behavior of a nonprismatic frame section for
  - — self-weight calculations
  - — linear variation of section area
  - — linear, parabolic, and cubic variation of moment of inertia

6.4.1 Test Input

The test input is located at \SAP2000\Validation\Example 1-006\1-006.SDB on the accompanying CD.

6.4.2 Test Procedure

The geometry, as described in Figure 6-7 was implemented in SAP2000. Three different cases were analyzed (1) self-weight, (2) an axial force, F_x at the free end with a magnitude of 500 kips, and (3) a moment M_y at the free end with a magnitude of 5,000 k-in [this last load case is case number 6 in the SAP2000 Verification Examples (2003)]. The analysis was executed to obtain displacements, rotations and reaction forces at the end of the beam. The SAP2000 results were compared with the hand calculations provided by the developer.

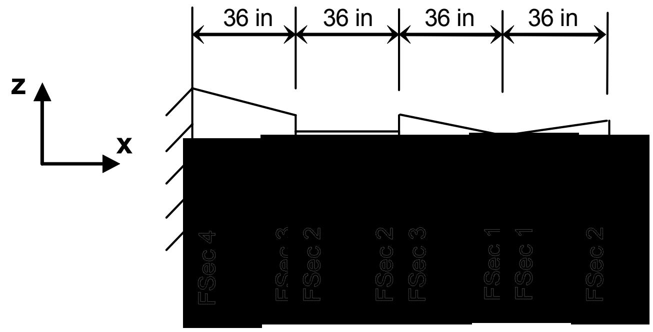


Figure 6-7. Cantilever Beam with Variable Cross-Section

Table 6-4. Section Properties of Example 1-006				
	Fsec1	**Fsec2**	**Fsec3**	**Fsec4**
Width b, in	12	12	12	12
Depth d, in	12	18	24	30
Area A, in^2	144	216	288	360
Bending I_{33}, in^4	1,728	5,832	13,824	27,000
Bending I_{22}, in^4	1,728	2,592	3,456	4,320

6.4.3 Test Results

The results for the above described case is tabulated in Table 6-5 and compared with the independent results. The results obtained match the results as given in the SAP2000 Software Verification Examples.

6.5 Example 1-008 of SAP2000 (2003): Frame—Partial Fixed End Releases

This example tests the SAP2000 frame with object partially fixed end releases using a cantilever beam subjected to uniform load representing twice the self weight of the beam. At the fixed end

Table 6-5. Results Comparison				
Load Case	Output Parameter	SAP2000	Independent	Percent Difference
1	M_y (fixed end) k-in	−184.95	−184.95	0
2	U_x (free end) in	0.09090	0.09087	0.03
3 (6)	R_y (free end) rad	0.03740	0.03742	0.05

of the cantilever the frame object is assigned a partial fixed moment spring having a stiffness of 3,888,000 k-in/rad and a partial fixed shear spring having a stiffness of 540 k/in.

The geometry, as well as the boundary conditions, of the cantilevered beam is as shown in Figure 6-8. The beam has an elastic modulus of 4,320 k/in^2, a Poisson's ratio of 0.2, a shear modulus of 1,800 k/in^2 and unit weight of 0.15 k/ft^3. The beam has a cross-sectional area of 540 in^2, a moment of inertia of 40,500 in^4 and a shear area of 450 in^2. As outlined above, the requirements being tested in this case are the shear and bending partial fixity end releases as applied to the single frame element as shown in Figure 6-8.

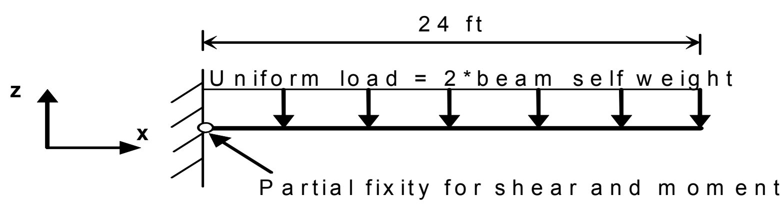


Figure 6-8. Cantilevered Beam, Example 1–008

6.5.1 Test Input

The test input is located at \SAP2000\Validation\Example 1-008\1-008.SDB on the accompanying CD.

6.5.2 Test Procedure

The above described beam geometry and corresponding load case of Example 1-008 was implemented in SAP2000. The analysis was executed to obtain the displacement U_z at the tip of the cantilever and the SAP2000 results were compared with the hand calculations provided by the developer.

6.5.3 Test Results

The result obtained from SAP2000, for U_z, is -0.803641 in. and the provided hand calculation has a deflection of 0.8036 in. for a percent difference of 0%.

6.6 Example 1-022 of SAP2000 (2003). Frame—Two-Dimensional Moment Frame with Static and Dynamic Loads

In this example, a seven-story, two-dimensional, fixed base frame subjected to lateral earthquake loads was modeled. The lateral earthquake load was modeled in four different ways: (i) a static lateral load, (ii) a response spectrum, (iii) a modal time history and as (iv) direct integration time history. For the response spectrum analysis, the square root sum of the squares and complete quadratic modal combination techniques are applied. The earthquake excitation is the N-S component of the 1940 El Centro earthquake, which is read by the model when the analysis is being performed. It is important to note that only the U_x, U_z and R_y degrees of freedom are active in this model.

The geometry, boundary conditions, loading and section properties of the frame are as shown in Figure 6-9. All of the members shown have an elastic Modulus of 29,500 k/in^2. The lateral displacements of the columns at each story level were constrained together using a separate diaphragm constraint at each story level. The mass at each story level was specified as 0.49 kip-sec^2/in.

The requirements being tested in this case were (i) diaphragm constraint, (ii) joint force assignments, (iii) joint mass assignments, (iv) modal analysis for eigenvalues, (v) response spectrum analysis, (vi) modal time history analysis for base excitation, and (vii) direct integration time history analysis for base excitation.

6.6.1 Test Input

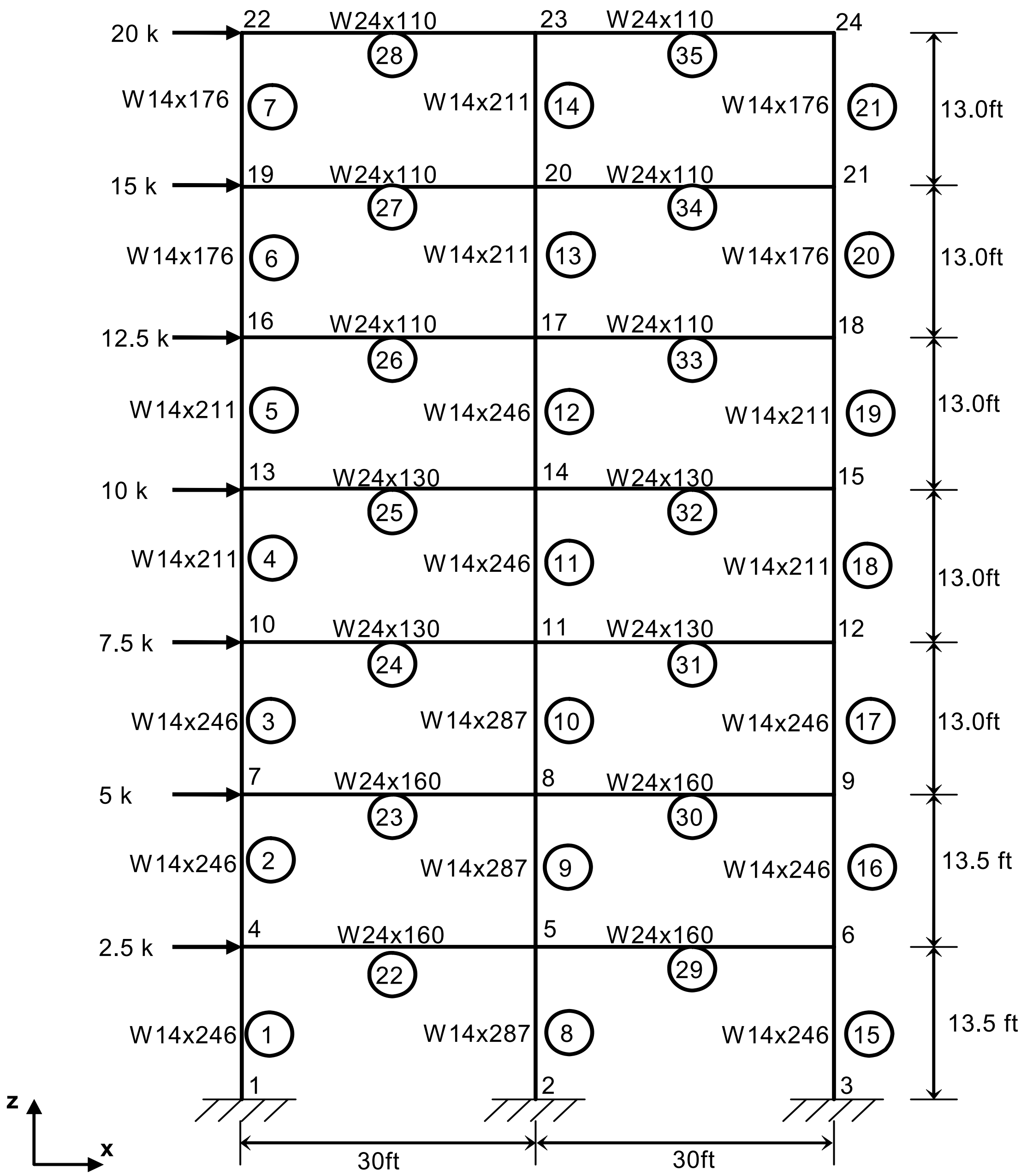
The test input is located at \SAP2000\Validation\Example 1-022\1-022.SDB on the accompanying CD.

6.6.2 Test Procedure

The geometry and six loading cases of Example 1-022 were implemented in SAP2000. The six loading cases considered were: (i) modal analysis, (ii) static lateral analysis, (iii) response spectrum analysis using square root sum of the squares, (iv) response spectrum using complete quadratic combination, (v) modal time history analysis and (iv) direct integration time history analysis. The analysis was executed to obtain displacements and forces at selected nodes and elements of the frame.

6.6.3 Test Results

The results obtained from SAP2000 are shown below in Tables 6-6 thru 6-11. The results compared well with the independent results given in the SAP2000 Verification Examples (2003).



Joint mass applied to joints 5, 8 ,11, 14, 17, 20, 23
mass = 0.49 k-sec^2/in in x direction only

Figure 6-9. Seven-Story, Two-Dimensional Frame

Table 6-6. Time Periods for Modal Analysis			
Output Parameter	**SAP2000**	**Independent**	**Percent Difference**
Mode 1 period (sec)	1.2732	1.2732	0
Mode 2 period (sec)	0.431	0.4313	0
Mode 3 period (sec)	0.242	0.242	0
Mode 4 period (sec)	0.1602	0.1602	0
Mode 5 period (sec)	0.119	0.119	0
Mode 6 period (sec)	0.0951	0.0951	0
Mode 7 period (sec)	0.0795	0.0795	0

Table 6-7. Static Lateral Analysis Results			
Output Parameter	**SAP2000**	**Independent**	**Percent Difference**
U_x at joint 22 (in)	1.45076	1.45076	0
Axial force in frame 1 (kip)	69.99	69.99	0
Moment in frame 1 at joint 1 (k-in)	2324.68	2324.68	0

Table 6-8. Response Spectrum Analysis Square Root Sum of the Squares			
Output Parameter	**SAP2000**	**Independent**	**Percent Difference**
U_x at joint 22 (in)	5.436	5.438	−0.04
Axial force in frame 1 (kip)	261.7	261.8	−0.04
Moment if frame 1 at joint 1 (k-in)	9860	9868	−0.08

Table 6-9. Response Spectrum Analysis Complete Quadratic Combination			
Output Parameter	**SAP2000**	**Independent**	**Percent Difference**
U_x at joint 22 (in)	5.431	5.438	−0.13
Axial force in frame 1 (kip)	261.5	261.8	−0.12
Moment if frame 1 at joint 1 (k-in)	9910	9868	0.43

Table 6-10. Modal Time History Results			
Output Parameter	**SAP2000**	**Independent**	**Percent Difference**
U_x at joint 22 (in)	5.49	5.46	0.55
Axial force in frame 1 (kip)	263	258	1.94
Moment if frame 1 at joint 1 (k-in)	9104	8740	4.16

Table 6-11. Direct Time Integration Time History Results			
Output Parameter	**SAP2000**	**Independent**	**Percent Difference**
U_x at joint 22 (in)	5.48	5.46	0.37
Axial force in frame 1 (kip)	263	258	1.94
Moment if frame 1 at joint 1 (k-in)	9183	8740	5.07

6.7 Example 1-026 of SAP2000 (2003): Frame—Moment and Shear Hinges

In this example a static nonlinear analysis was performed on a cantilever beam utilizing moment and shear hinges, both of which are located at its fixed end. A vertical load, P, is applied to the cantilever and increased until the vertical tip deflection is equal to 2 in.

The geometry, boundary conditions and loading are shown in Figure 6-10. The cantilever has an elastic modulus of 3,600 k/in^2, Poisson's ratio of 0.2 and a shear modulus of 1,500 k/in^2. The section properties consist of a cross-sectional area of 216 in^2 (b = 12 in, d = 18 in Figure 6-10), a moment of inertia of 5,832 in^4 and a shear area of 180 in^2. Also shown in Figure 6-10 are the moment-rotation and shear-deformation input curves for the moment and shear hinges, respectively.

It is important to note that in this example both bending and shear deformations are included. The requirements tested in this case involved the static nonlinear analysis of a frame structure with moment and shear hinges. The load involves increasing P until the tip deflection in the Z direction is 2 in.

6.7.1 Test Input

The test input is located at \SAP2000\Validation\Example 1-026\1-026.SDB on the accompanying CD.

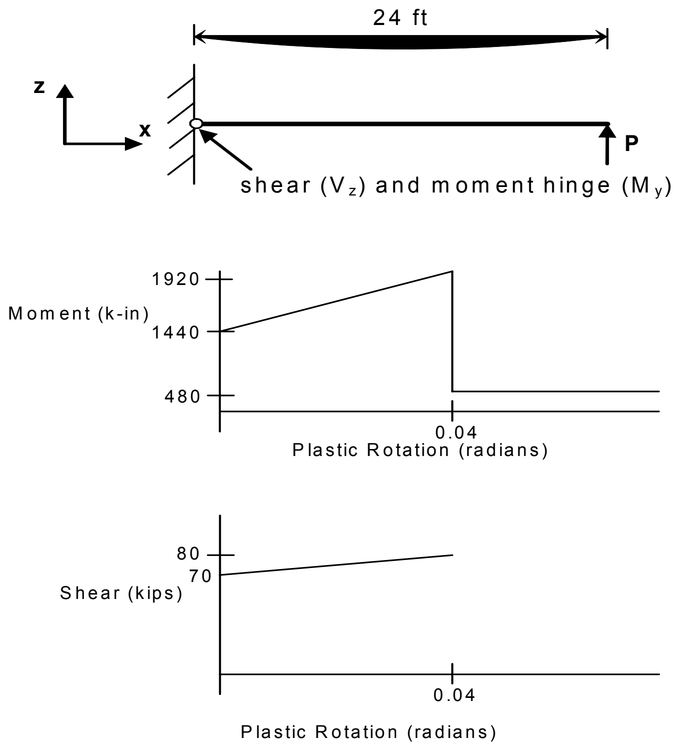


Figure 6-10. Cantilever Beam with Moment and Shear Hinges

6.7.2 Test Procedure

The geometry and loading case of Example 1-026 were implemented in SAP2000 and the analysis was executed to obtain displacements and reactions of the cantilever beam. The SAP2000 results were compared with hand calculated results using the unit load method (virtual work) described in Cook and Young (1985), which are provided by the developer.

6.7.3 Test Results

The displacements are tabulated and compared with the independent results (Table 6-12). The results compare favorably with the independent results in the SAP2000 Verification Examples (2003)

Table 6-12. Displacements and Reactions				
Point	**Output Parameter**	**SAP2000**	**Independent**	**Percent Difference**
1	Force P (free end) kips	60	60	0
	U_z (free end) in	0.0185	0.0185	0
	R_y (free end) rad	−0.0010	−0.0008	25
2	Force P (free end) kips	80	80	0
	U_z (free end) in	1.3847	1.3847	0
	R_y (free end) rad	−0.0411	−0.0411	0

6.8 Example 2-008 of SAP2000 (2003): Shell—Cantilever Plate Eigenvalue Problem

A cantilever plate was analyzed to find the periods of the first five plate-bending modes of vibration. As shown in Figure 6-11, the plate is 24 in. square and one inch thick which the lower bottom edge to be totally fixed (i.e., joints 1 and 2 have U_y and R_x restrained).

The plate has an elastic modulus of 29,000 k/in^2, a Poisson's ratio of 0.3, and a mass per unit area of 7.35 × 10^{-7} k-sec^2/in^3 is assigned to the plate. The entire plate was modeled as a single "area object" and a 40 × 40 mesh was created using the automatic area meshing feature of SAP2000.

The SAP2000 features being tested in this case are (i) eigenvalue analysis using shell elements, (ii) "area object" mass assignment, (iii) "area object" automatic mesh generation.

6.8.1 Test Input

The test inputs are located at \SAP2000\Validation\Example 2-008\2-008a.SDB and 2.008b.SDB on the accompanying CD.

6.8.2 Test Procedure

The geometry of Example 2-008 was implemented in SAP2000 and an eigenvalue analysis was executed to obtain the first five periods of vibration of the cantilever plate. This plate was anlayzed using both the "thin plate" and "thick plate" options in SAP2000. The SAP2000 results are compared with hand calculated results presented in Harris and Crede (1976).

6.8.3 Test Results

The first five periods of vibration are given in Table 6-13 (thin plate) and Table 6-14 (thick plate) and compared with the independent results. Note that all of the results compare favorably

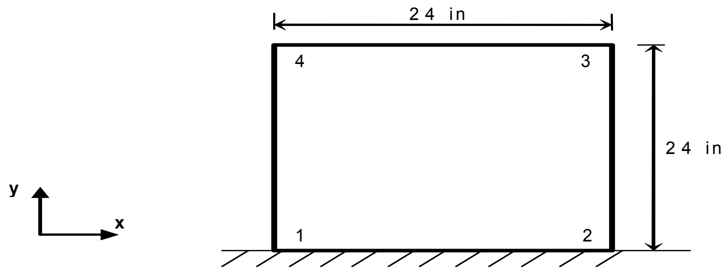


Figure 6-11. Cantilever Plate

Table 6-13. Time Periods for Thin Plate				
Output Parameter	**Mesh**	**SAP2000**	**Independent**	**Percent Difference**
Period Mode 1 sec	40 × 40	0.01735	0.01723	0.70
Period Mode 2 sec	40 × 40	0.00708	0.00704	0.57
Period Mode 3 sec	40 × 40	0.00283	0.00281	0.71
Period Mode 4 sec	40 × 40	0.00222	0.00219	1.37
Period Mode 5 sec	40 × 40	0.00200	0.00193	3.62

Table 6-14. Time Periods for Thick Plate				
Output Parameter	**Mesh**	**SAP2000**	**Independent**	**Percent Difference**
Period Mode 1 sec	40 × 40	0.01739	0.01723	0.93
Period Mode 2 sec	40 × 40	0.00712	0.00704	1.14
Period Mode 3 sec	40 × 40	0.00285	0.00281	1.42
Period Mode 4 sec	40 × 40	0.00222	0.00219	1.37
Period Mode 5 sec	40 × 40	0.00200	0.00193	3.62

with the independent results and with those results given in the SAP2000 Verification Examples (2003).

6.9 Example 2-009 of SAP2000 (2003): Shell—Plate on Elastic Foundation

In this example, a single 50 kip point load was applied at the center of an "infinitely large" plate is supported on an elastic foundation. The plate is 300 in. long by 300 in. wide and one inch thick, Figure 6-12. The entire plate was treated as a single "area object." A 50 × 50 mesh of shell elements with the thin plate option was used. The elastic modulus is 29,000 k/in^2 with a Poisson's ratio of 0.3 and the soil subgrade modulus (elastic foundation) is 800 k/ft^3. The SAP2000 features being tested in this case are (i) plate bending analysis using shell elements, (ii) area object spring assignment, (iii) joint force loads.

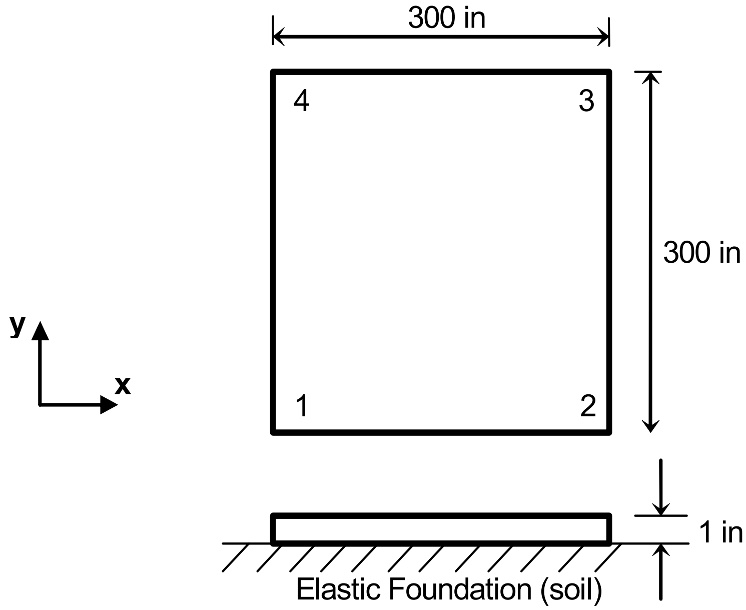


Figure 6-12. Plate on an Elastic Foundation

6.9.1 Test Input

The test inputs are located at \SAP2000\Validation\Example 2-009\2-009a.SDB and 2.009b.SDB on the accompanying CD.

6.9.2 Test Procedure

The geometry and load case of Example 2-009 were implemented in SAP2000. The analysis was executed to obtain the displacements at the center of the plate and the SAP2000 results were compared with hand calculated results presented in Timoshenko and Wionowsky-Krieger (1959).

6.9.3 Test Results

The displacement at the center of the plate, U_z, is shown in Table 6-15 for comparison with the independent results provided. Note that the obtained result compares favorably with that reported by Timoshenko and Wionowsky-Krieger (1959).

Table 6-15. Center Displacement for Thin Plate Elements

Model and Modulus	Output Parameter	SAP2000	Independent	Percent Difference
50 × 50 mesh k = 800 k/ft3	U_z at center of plate (in)	−0.1827	−0.1782	2.53

6.10 Example 2-010 of SAP2000 (2003): Cylinder with Internal Pressure

In this example, a cylinder was analyzed subjected to an internal pressure load. As shown in Figure 6-13, the cylinder is 200 in. tall, has a 60 in. radius and a wall thickness of one in. The material properties for the cylinder are: an elastic modulus of 29,000 k/in^2, Poisson's ratio of 0.3 and a shear modulus of 11,154 k/in^2. The applied load is a uniform radial pressure of 1 k/in^2 on the inside face of the entire cylinder. The model uses an 8 × 16 mesh (height x circumference) of shell elements using the thin plate option.

The features being tested in this case were (i) a three dimensional shell analysis, (ii) surface pressure loads, and (iii) joint "local axes." (Figure 6-13).

6.10.1 Test Input

The test input is located at \SAP2000\Validation\Example 2-010\2-010.SDB on the accompanying CD.

6.10.2 Test Procedure

The geometry and the load case of Example 2-010 was implemented in SAP2000 and the analysis was executed to obtain the stresses and displacements at the top of the cylinder. The

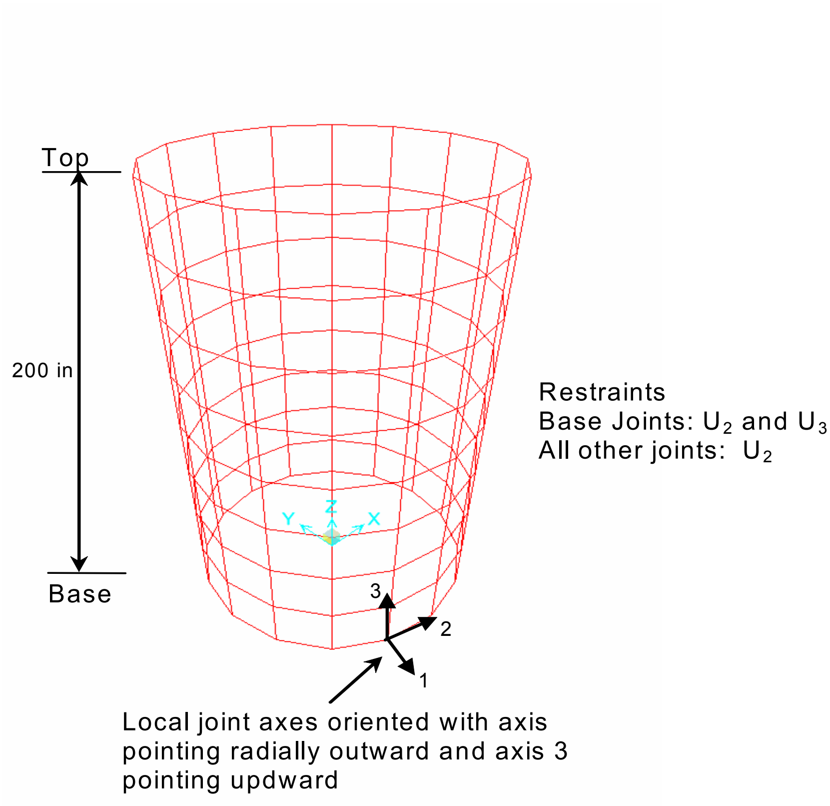


Figure 6-13. Cylinder Subjected to Uniform Internal Pressure

SAP2000 results were compared with hand calculated results based on equations of Roark and Young (1975), as presented in SAP2000.

6.10.3 Test Results

The stresses (σ_{11}) and displacements (U_z ane U_3) given in local axes at the top of the cylinder are shown in Table 16-16 and compared with the independent results. Note that the results compare well with the independent solution given in the SAP2000 Verification Examples (2003).

Table 6-16. Cylinder Under Internal Pressure Displacements and Stresses

Output Parameter	SAP2000	Independent	Percent Difference
U_1 (any joint) (in)	0.12175	0.12414	−1.92
U_3 (at top joint) (in)	−0.12175	−0.12414	−1.92
σ_{11} (anywhere) (k/in^2)	59	60	−1.67

6.11 Example 2-013 of SAP2000 (2003): Constant Temperature Load Through Shell Thickness

A flat rectangular shell with irregularly shape elements is subjected to a temperature load that is constant through the thickness of the plate. The original temperature for the plate is 16 °C [60 °F], the temperature load consists of raising the temperature to 71 °C [160 °F].

The model under consideration is shown in Figure 6-14. The plate has a thickness of 0.001 in. The material properties for the plate are: an elastic modulus of 1,000,000 lb/in^2, a Poisson's ratio of 0.25, and coefficient of thermal expansion of 5.5×10^{-6}.

Two identical models were created except for the joint restraints. The first model (A) has a fixed restraint at one of the corners allowing free expansion of the plate. The second model (B) has pinned restraints at the four corner points, thus restraining the temperature expansion. The SAP2000 features tested in this case were (i) temperature loading and (ii) using a reference temperature with shell elements.

6.11.1 Test Input

The test inputs are located at \SAP2000\Validation\Example 2-013\2-013a.SDB and 2-013b.SDB on the accompanying CD.

6.11.2 Test Procedure

The geometry and the thermal load of Example 2-013 were implemented in SAP2000. The analysis was executed to obtain the joint displacements and thermal stresses in the plate. The SAP2000 results were compared with hand calculated results based on equations presented in Cook and Young (1985) and given in the SAP2000 Verification Examples (2003).

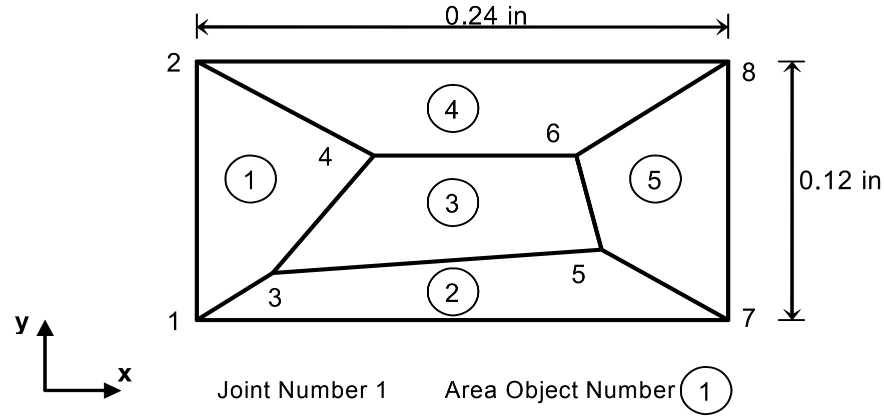
6.11.3 Test Results

The obtained joint displacements (U_x and U_y) and thermal stresses (σ_{xx} and σ_{yy}) in the plate are shown in Table 6-17 and compared with the independent results. Excellent agreement is observed between the present SAP2000 results and those given in the SAP2000 Verification Examples (2003).

6.12 Example 6-009 of SAP2000 (2003): Link—Plastic Kinematic Link

The behavior of a plastic kinematic link element was tested using a single-degree-of-freedom system. Multi-linear elastic force-deformation characteristics were defined for tension and compression behavior.

The SAP2000 model consists of a single joint and one link element (Figure 6-15). A nonlinear static analysis is used to "pull" the link element to a positive 12 in. displacement. A second



Joint Coordinates (inches)			
Joint	x	y	z
1	0	0	0
2	0	0.12	0
3	0.04	0.02	0
4	0.08	0.08	0
5	0.18	0.03	0
6	0.16	0.08	0
7	0.24	0	0
8	0.24	0.12	0

Figure 6-14. Rectangular Plate Under Thermal Loading

Table 6-17. Displacements and Thermal Stresses				
Model	**Output Parameter**	**SAP2000**	**Independent**	**Percent Difference**
A Free Expansion	U_x (joint 8) (in)	0.000132	0.000132	0
	U_y (joint 8) (in)	0.00007	0.00007	0
	σ_{xx} (joint 8) (lb/in^2)	0	0	0
	σ_{yy} (joint 8) (lb/in^2)	0	0	0
B Restrained	U_x (joint 8) (in)	0	0	0
	U_y (joint 8) (in)	0	0	0
	σ_{xx} (joint 8) (lb/in^2)	−733.33	−733.33	0
	σ_{yy} (joint 8) (lb/in^2)	−733.33	−733.33	0

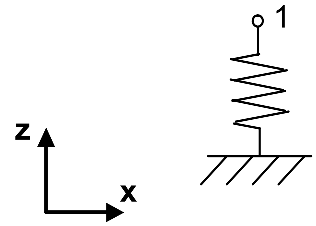


Figure 6-15. Kinematic Link

nonlinear analysis started from the final end conditions of the first analysis case and “pushed” the link element to a negative 12 in. displacement.

The SAP2000 features being tested in this case were (i) plastic kinematic links, (ii) displacement-controlled nonlinear static analysis, and (iii) link gravity load.

6.12.1 Test Input

The test inputs are located at D:\SAP2000\Validation\Example 6-009\6-009.SDB on the accompanying CD.

6.12.2 Test Procedure

The simple single element geometry and the corresponding force-deformation behavior (Figure 6-16) for the single link of Example 6-009 were implemented in SAP2000. The analysis was executed to obtain the force-deformation characteristics of the single-degree-of-freedom system. The SAP2000 results were compared with hand calculation results provided by the developer.

6.12.3 Test Results

The SAP2000 generated force-deformation curve is shown in Figure 6-16 and is similar to that shown in the SAP2000 Software Verification Examples (2003). Selected numerical results are also shown in Table 6-18.

The present results obtained from SAP2000 shows perfect agreement with the results given in the SAP2000 Verification Examples (2003).

6.1.3 Summary of Results

For all of the cases presented in this report, the expected agreement with the SAP2000 Software Verification Examples (2003) was obtained. Specifically, the perfect reproduction of the SAP2000 results serves to show that the SAP2000 software is performing the calculations with comparable accuracy.

7.0 Industry Experience (for Acquired Software Only)

The SAP2000 program, as well as the predecessor versions (SAPIV, SAP90, etc.) has been extensively used in industry and academia. SAP2000 has thousands of users in more than 100 countries. (Computers and Structures, Inc. www.csi.berkeley.com) a representative example of the global presence of SAP2000 is the firm Computers and Engineering based in Germany which provides consulting engineering services and software for engineering companies. Some of the projects in which SAP2000 has been used to analyze structural systems can be found on their web page (http://www.comp-engineering.com).

Point	Deformation (inch)	Force (kip)
A	−16	−55
B	−6	−50
C	−1	−40
D	0	0
E	2	50
F	6	70
G	16	80

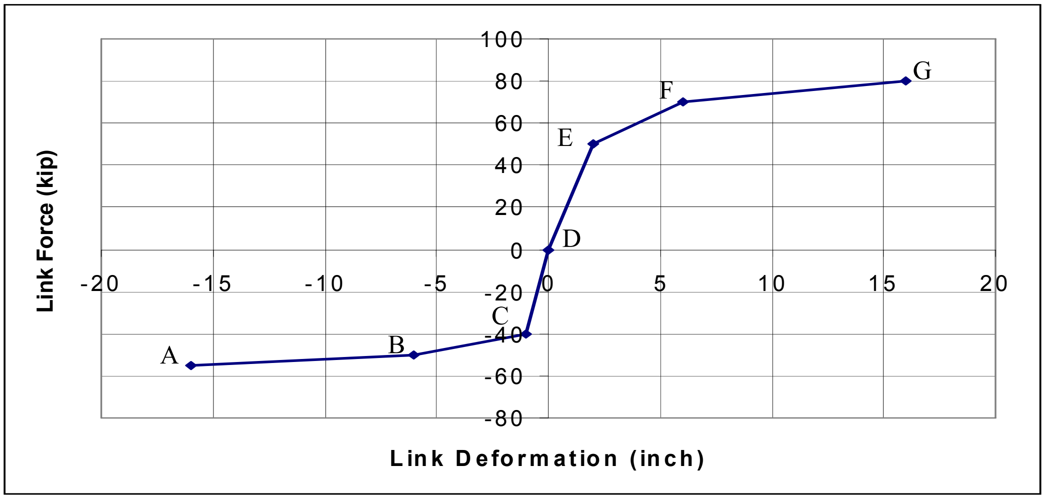


Figure 6-16. Force-Deformation Behavior for Link

Table 6-18. Force Comparison					
Output Parameter	**Link Deformation**	**Analysis Case**	**SAP2000**	**Independent**	**Percent Difference**
Link force at specified deformation for specified analysis case kips	1.2 in	NLSTAT1	30	30	0
	4.2 in	NLSTAT1	61	61	0
	12 in	NLSTAT1	76	76	0
	11.52 in	NLSTAT2	64	64	0
	9.6 in	NLSTAT2	10	10	0
	2.4 in	NLSTAT2	−36	−36	0
	−12 in	NLSTAT2	−52.25	−52.25	0

A cursory search of the open literature shows many examples in which SAP2000 is used as the structural analysis tool. The following are some representative examples.

1. Aristizabal-Ochoa, J.D., *Elastic Stability and Second-Order Analysis of Three-Dimensional Frames: Effects of Column Orientation,* J. Engrg. Mech., Volume 129, Issue 11, pp. 1254-1267 (November 2003)

2. Salonikios, T., Karakostas, C., Lekidis, V., Anthoine, A., *Comparative Inelastic Pushover Analysis of Masonry Frames*, Engineering Structures, Volume 25, Issue 12, pp. 1515-1523, October 2003.

3. Romero, M.L., and Museros, P., *Structural Analysis Education through Model Experiments and Computer Simulation*, J. Profl. Issues in Engrg. Educ. and Pract., Volume 128, Issue 4, pp. 170-175 (October 2002)

4. Mabsout, M.E., Baddaf, I.Y., Tarhini, K.M., and Frederick, G.R., *Load Reduction in Steel Girder Bridges*, Pract. Periodical on Struct. Des. and Constr., Volume 7, Issue 1, pp. 37-43 (February 2002)

5. Mylonakis, G., Papastamatiou, D., Psycharis, J, and Mahmoud, K., *Simplified Modeling of Bridge Response on Soft Soil to Nonuniform Seismic Excitation*, J. Bridge Engrg., Volume 6, Issue 6, pp. 587-597 (November/December 2001)

In addition, there are numerous conference proceedings in which SAP2000 is cited. For example,

1. Goodson, M. W. and John E. Anderson, J. E. Ph.D., *Soil-Structure Interaction – A Case Study,* 2005 Structures Congress, ASCE, April 20-24, 2005.

2 Grierson, D. E., Safi M., Xu, L. and Liu, Y., *Simplified Methods for Progressive-Collapse Analysis of Buildings*, 2005 Structures Congress, ASCE, April 20-24, 2005.

3. Haight, R., Chang, S., and Kushmock, R., *Orthotropic Deck Rehabilitation at the Throgs Neck Bridge*, 2005 Structures Congress, ASCE, April 20-24, 2005.

4. Spoth, T., Serzan, K., and Condell, S., *The New Tacoma Narrows Suspension Bridge Design of the Suspended Superstructure*, 2005 Structures Congress, ASCE, April 20-24, 2005.

5. Hart, G. C., Ekwueme, C. G. and Tekie, P. B., *Analysis of the Collapse of a Bridge Falsework*, 2005 Structures Congress, ASCE, April 20-24, 2005.

6. Buscemi, N. and Marjanishvili, S., *SDOF Model for Progressive Collapse Analysis*, 2005 Structures Congress, ASCE, April 20-24, 2005.

7. Manos, G. C., Soulis, V. J., and Diagouma A., *Preliminary Numerical Investigation of the Dynamic Characteristics of Historic Monuments*, 13th World Conference on Earthquake Engineering, Vancouver, B.C., Canada, August 1-6, 2004

8. Charney, F. A. and Ibrahim, Y. E. *A New "Viscoplastic" Passive Energy Device*, 13th World Conference on Earthquake Engineering, Vancouver, B.C., Canada, August 1-6, 2004

9. Talebi, S. and Kianoush, M. R., *Behavior of Reinforced Concrete Frames Designed for Different Levels of Ductility*, 13th World Conference on Earthquake Engineering, Vancouver, B.C., Canada, August 1-6, 2004

10. Hayes, Jr. J. R. , Woodson, Pekelnicky, R. G, Poland, C. D., Corley, W. G. Sozen, Mahoney, M. M. and Hanson, R. D., *Earthquake Resistance and Blast Resistance: A Structural Comparison,* 13th World Conference on Earthquake Engineering, Vancouver, B.C., Canada, August 1-6, 2004

11. Hilmy, S. I., Elhassan, R. M. Mosalam, K. M. and Porush, A., *Seismic Performance and Rehabilitation of Perforated Waffle-Slab/Column Lateral System in Moderate Seismic Areas*, Seventh U.S. National Conference on Earthquake Engineering (7NCEE), Boston MA., July 21-25, 2002.

12. Miyamoto, K., and Glasgow, R., *Vertical Addition to an Existing Non-Ductile Concrete Structure Enhanced by Passive Energy Dissipators*, Seventh U.S. National Conference on Earthquake Engineering (7NCEE), Boston MA., July 21-25, 2002.

13. Islam, S., Wu, H., *Seismic Retrofit of a 14-Story Suspended Floor Building Using Advanced Analysis and Energy Dissipation Devices*, Seventh U.S. National Conference on Earthquake Engineering (7NCEE), Boston MA., July 21-25, 2002.

14. Carden, L. P., Itani, A. M. and Buckle, I. G., *Influence of Shear Connectors in Slab on Girder Bridge Superstructures Subjected To Transverse Earthquake Loading*, Seventh U.S. National Conference on Earthquake Engineering (7NCEE), Boston MA., July 21-25, 2002.

15. Marinilli, A. and Castilla, E., *"Cage-Like" Steel Reinforcement in Seismic Performance of Reinforced Concrete Elements of Low Aspect Ratio*, Seventh U.S. National Conference on Earthquake Engineering (7NCEE), Boston MA., July 21-25, 2002.

16. Timchenko, I., *Seismic Vulnerability Assessment of Buildings on the Basis of Numerical Analyses*, 12th European Conference on Earthquake Engineering, London, Sept. 9-13, 2002.

17. Beamish, M. J. and Billings, I. J., *Auckland Harbour Bridge Seismic Retrofit,* 12th World Conference on Earthquake Engineering, Auckland, New Zealand, 2000.

18. Davis, H. A. and Robertson, D. R., *Hearst Memorial Mining Building Seismic Improvements, University of California, Berkeley*, 12th World Conference on Earthquake Engineering, Auckland, New Zealand, 2000.

19. Bell, D. K. and Davidson, B. J., *Simulation of the Non-Linear Seismic Response of an Arch Dam*, 12th World Conference on Earthquake Engineering, Auckland, New Zealand, 2000.

20. Kehoe, B. E. and Attalla, M. R., *Considerations of Vertical Acceleration on Structural Response,* 12th World Conference on Earthquake Engineering, Auckland, New Zealand, 2000.

21. Black C. J. and Aiken, I. D., *Joint Time-Frequency Analysis of Base Isolated Buildings*,12th World Conference on Earthquake Engineering, Auckland, New Zealand, 2000.

Lately, SAP2000 is being used in academia as a tool for teaching. Educational versions of SAP2000 is being included in some of the latest engineering textbooks, i.e.,

1. James K. Nelson and Jack C. McCormac, *Structural Analysis: Using Classical and Matrix Methods*, Wiley, John and Sons, Inc, December 2002.

2. Mario Paz and William Leigh, *Structural Dynamics: Theory and Computation*, Fifth Edition, Kluwer Academic Publishers, December 2003.

The above examples of the widespread use of SAP2000 should serve to demonstrate the confidence and acceptance of the validity of the software.