

“Optimized Modeling and Design of Structures using SAP2000”

presented by



SEMINAR TOPICS

- Object Based Modeling Using Templates
- Application of Seismic/Wind Loads- Including IBC 2003 code
- Extrusion Options Points/Lines/Areas to Solid Elements
- Shell and Solid Internal Meshing
- Edge Line Constraints Application
- Interactive Database Editing using Microsoft Excel/Access
- Importing/ Exporting Options
- Open Structure Wind Loads Analysis
- Influence Lines and Surfaces
- Use of Graphical Section Cuts
- Eigenvector and Ritz Analysis
- Curved Frame, Cable and Tendon Elements
- Tension/Compression Limits
- Truss Optimization Modeling
- Aluminum Frame Design
- Cold Formed Steel Frame Design
- Section Designer Module
- Transmission Towers
- Dynamic Analysis- Time History and Response Spectrum Analysis
- Steel/ Concrete Frame Design
- Steel and Aluminum Frame Drift Optimization
- Post Tensioning Options

SPECIAL ITEMS

- Staged Construction Analysis
- Creep and Shrinkage
- Large Deformation Analysis
- Multiple P-Delta Analysis
- Buckling Analysis
- Blast Analysis
- Nonlinear Link Behavior- Base Isolator and Damper Elements
- Bridge Analysis

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:

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

Phone: (510) 845-2177

FAX: (510) 845-4096

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

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

web: <http://www.csiedu.com>

Table of Contents

Introduction	4
Example I General Modeling Techniques – 2-D Shear Wall Analysis	5
Example II Modeling Techniques using Templates – Transmission Tower	11
Example III Truss Modeling – Curved Steel Truss Example	17
Example IV Bridge Analysis – Concrete Box Girder Bridge	25
Example V Advanced Modeling Techniques – Eight-story Parking Garage	41
Example VI 2-D Frame Analysis	45
Example VII Nonlinear Time History Analysis	51
About the Speakers	57
“Mesh Transitioning and Compatibility – Automated Line Constraint in ETABS and SAP2000”	59

Introduction

This lecture is generally geared towards the intermediate user level of SAP2000. However, if you have never used SAP2000 before, do not be set back. We have designed this course in such a way that even the inexperienced SAP2000 user will have no problem following along. All of the examples that we present (except for the few real life models in the beginning) will be drawn from scratch to exhibit the most general and common modeling techniques mentioned above.

We have chosen seven examples that we will describe in the presentation. In these seminar notes, you will find descriptions, computer model definitions, and results for each example. As we present each example, please feel free to follow along.

EXAMPLE I **General Modeling Techniques – 2-D Shear Wall Analysis**

Description

This is a two-story frame analysis, subjected to lateral static loading. There are 4 frames in the X-Z plane that consist of 1 inch concrete shell elements. Each frame is meshed differently as shown in Figure 1-1. Only the 2nd frame has edge constraints assigned to it.

Significant Options of SAP2000 Exemplified

- ❑ Shell Internal and External Meshing
- ❑ Edge Line Constraints Application
- ❑ Interactive Database Editing using Microsoft Excel

Computer Model Definition

The 2D frames are two stories high with a story height of 24 ft. Each bay is spaced 24 ft. apart. Kip-inch-second units are used. The modulus of elasticity used for concrete is 3600 ksi.

Other parameters associated with the structure are as follows:

Coefficient of Thermal Expansion	= 6.500E-06
Poisson's ratio	= 0.3
Concrete Compression Strength, f'_c	= 4 ksi
Bending Reinforcement Yield Stress, f_y	= 60 ksi

Slab Properties:

Slab Thickness (Bending)	= 1 in
Slab Thickness (Membrane)	= 1 in

Building Loads:

Lateral Load on Roof level (Load applied in x, y, and z directions)	= 1 kip
--	---------

Interactive Database Editing

Use the Interactive Database Editing form to make any additions, deletions, or modifications to the tables selected using the Choose Tables for Interactive Editing form. The changes can then be applied to the model and shown in the display area of the SAP2000 window. The Interactive Database Editing form has a display area and various menu options and buttons that can be used to add records to or delete records from the table shown in the display area, copy records, move records, and find or find and replace user-specified data.

- **{Table Name}** drop-down list. Use the drop-down list near the top of the form to select the table to be edited. If the alternative method of displaying the form is used (see above), only one table name will be listed in this drop-down list.
- **Spreadsheet** area. Modify records in the table by typing directly in the cells, including using numerical formulas (e.g., 2+3/4). Right-click within any field (column) and a menu of pertinent operations will appear. The menu includes the Field Description option, which when clicked displays a form that shows a description for the item. If the field expects numeric input, the menu includes the Calculator option, which can be used to speed data entry when formulae are involved. If the field expects text input, use the View menu > Drop-Down List command to display a drop-down list of all acceptable values for the selected cell.

Important: Objects and other defined entities are identified by their names in the tables. If you change any names, those items will be considered as new when the tables are brought back into the model. Do not use interactive table editing to rename objects or other defined entities! Instead, use the Edit menu > Change Labels command.

The units used for a given field can be changed by clicking on the cell just below the column heading. Select the new units from the drop-down list that appears. Changing the units for one field will change the units for other related fields. The units chosen while interactively editing tables are temporary. All values will be converted back to the current model units when the editing is applied to the model.

We are interested in changing the joint lateral load for frames 2, 3 and 4 from 1 kip to 1.25 kips. Go to Edit> Interactive Database Editing>expand and select the joint load assignments option and click OK. A table will appear with all of the joint loads assigned to the structure. Click the 'To Excel button'. Change all loads to 1.25 kips wherever you see a 1 kip load. The excel spreadsheet should look like Figure 1-1:

Joint	LoadCase	CoordSys	F1	F2	F3	M1	M2	M3
Text	Text	Text	Kip	Kip	Kip	Kip-in	Kip-in	Kip-in
5	x	GLOBAL	1	0	0	0	0	0
5	z	GLOBAL	0	0	1	0	0	0
6	y	GLOBAL	0	1	0	0	0	0
6	x	GLOBAL	1	0	0	0	0	0
6	z	GLOBAL	0	0	1	0	0	0
6	y	GLOBAL	0	1	0	0	0	0
9	x	GLOBAL	1.25	0	0	0	0	0
9	y	GLOBAL	0	1.25	0	0	0	0
9	z	GLOBAL	0	0	1.25	0	0	0
10	x	GLOBAL	1.25	0	0	0	0	0
10	y	GLOBAL	0	1.25	0	0	0	0
10	z	GLOBAL	0	0	1.25	0	0	0
15	x	GLOBAL	1.25	0	0	0	0	0
15	y	GLOBAL	0	1.25	0	0	0	0
15	z	GLOBAL	0	0	1.25	0	0	0
16	x	GLOBAL	1.25	0	0	0	0	0
16	y	GLOBAL	0	1.25	0	0	0	0
16	z	GLOBAL	0	0	1.25	0	0	0
16	x	GLOBAL	1.25	0	0	0	0	0
16	y	GLOBAL	0	1.25	0	0	0	0
16	z	GLOBAL	0	0	1.25	0	0	0
21	x	GLOBAL	1.25	0	0	0	0	0
21	y	GLOBAL	0	1.25	0	0	0	0
21	z	GLOBAL	0	0	1.25	0	0	0
21	x	GLOBAL	1.25	0	0	0	0	0

Figure 1-1 Excel Spreadsheet

Next, save the Excel file and click ‘From Excel’ in the Interactive Database dialogue box. All of the changed values will be imported into SAP2000. Click the ‘Apply to Model’ button and then press Done. As shown in Figure 1-2, the loads have been successfully added to the model.

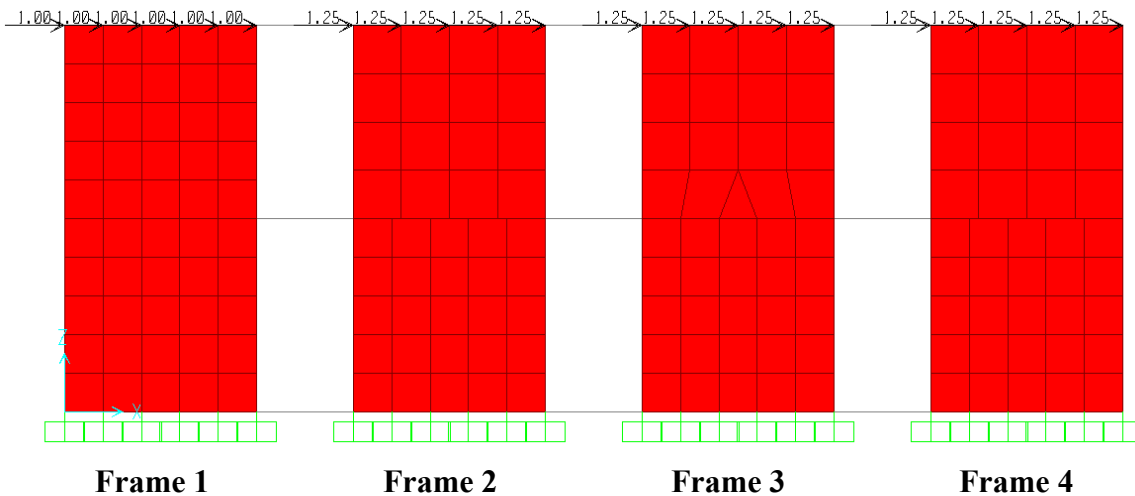


Figure 1-2 2-D view of meshed shell elements

As you can see in Figure 1-2, Frame 1 is uniformly meshed in a 5x5 pattern. Frame 2 is meshed 5x5 on top and 4x4 on the bottom. **The auto edge constraint option has been**

enabled for Frame 2 only. Frame 3 has a triangular shaped shell object drawn in the middle for joint connectivity. Frame 4 is identical to Frame 2 except for the fact that there is no edge constraint assigned to Frame 4. Equivalent joint loads have been assigned to all 4 structures depending on the number of mesh points along the tops of the frames.

Auto Edge Constraint

You can assign a specific mesh to area elements under Assign>Area>Generate Edge Constraints. In typical finite element analysis, shell elements are connected to other elements at corner points only. When an element does not frame into the corner point of a shell element, but instead frames into the edge of the shell element, no connection exists between the element and the shell element. SAP2000 is able to perform the analysis in this fashion. However, SAP2000 also has the capability of using an auto edge constraint. The SAP2000 auto edge constraint feature allows you to specify that elements framing into the edge of a shell element be connected to the shell element. SAP2000 internally takes care of connection between the elements by constraining points lying along an edge of the shell element to move with that edge of the element. Additional information regarding the auto edge constraint functionality in SAP2000 can be found in “Mesh Transitioning and Compatibility – The Automated Line Constraint in ETABS and SAP2000”. This paper can be found on page 59 of these notes.

Results

Shown in Figure 1-3 below are the displacements of the top right corner joint for each frame. You can see that Frame 4 has uneven meshing between levels and no auto edge constraints. This is why the deflection for Frame 4 is 33% larger than any of the other frames. Frame 4 also has some separation between floors due to the lack of edge constraints.

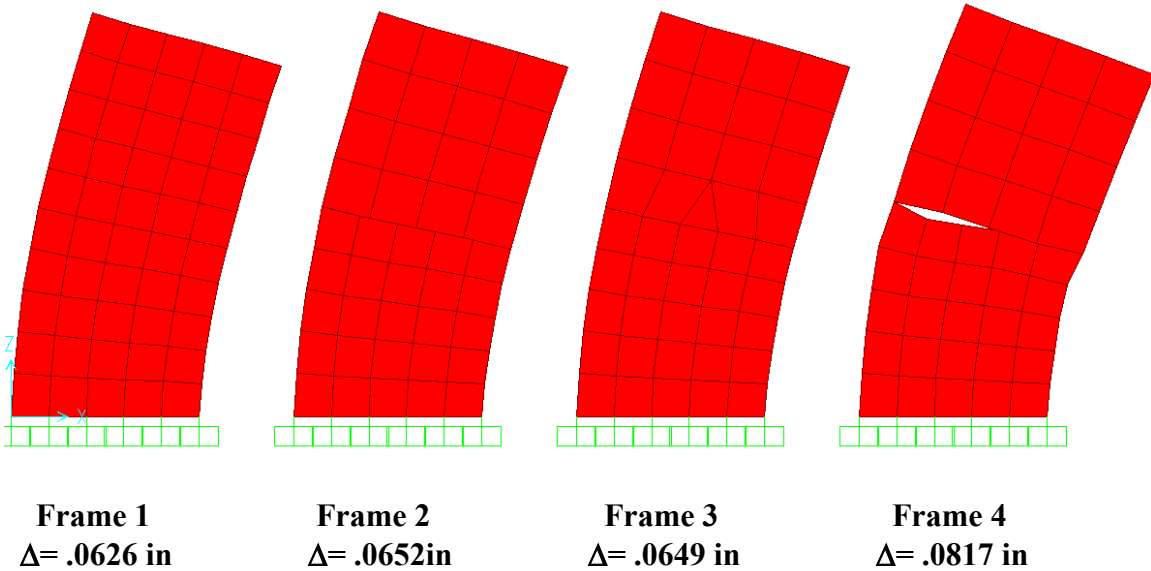


Figure 1-3 *Deflected Shape in X-Direction*

Figure 1-4 shows the stress diagram (S22) for all four shell structures. To view stress diagrams, go to Display>Show Forces/Stresses> Shells and select the component you wish to see. You can see that the stress contours shown in Frame 4 are not continuous because the edge constraint option has not been activated. A direct load path has not been established in Frame 4.

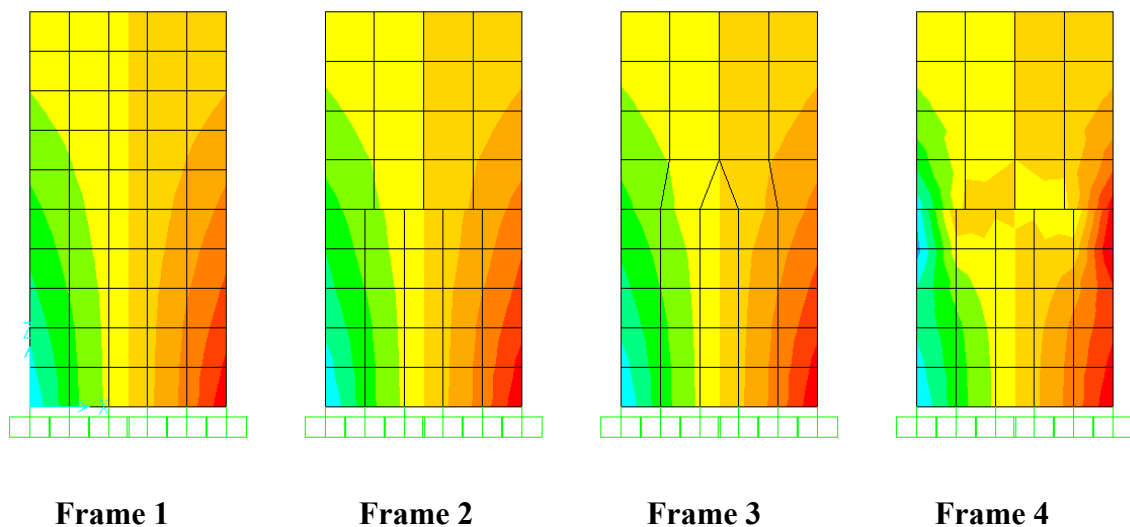


Figure 1-4 *Stress Contours (S22 component)*

EXAMPLE II Modeling Techniques using Templates – Transmission Tower

Description

This is a four-story transmission tower structure with brace elements located at each level. Each floor has a 20 ft. story height. Each bay is 16 ft. long in the X-direction and 16 ft. long in the Y-direction. The structure is laterally supported by a numerous brace elements located throughout the tower. Please refer to Figure 2-1 to see the system geometry setup.

Significant Options of SAP2000 Exemplified

- ❑ Object Based Modeling Using Templates
- ❑ Open Structure Wind Load Analysis
- ❑ Transmission Towers
- ❑ Dynamic Analysis- Response Spectrum Analysis
- ❑ Eigenvector and Ritz Analysis

Computer Model Definition

To create this model, go to the File Menu> New Model and select the 3D trusses option. Click on the 3D Truss Type pull down menu in the top left hand corner. Select the Transmission Tower 3 option. Leave all of the default settings and select the parametric definition button to view the model parameters.

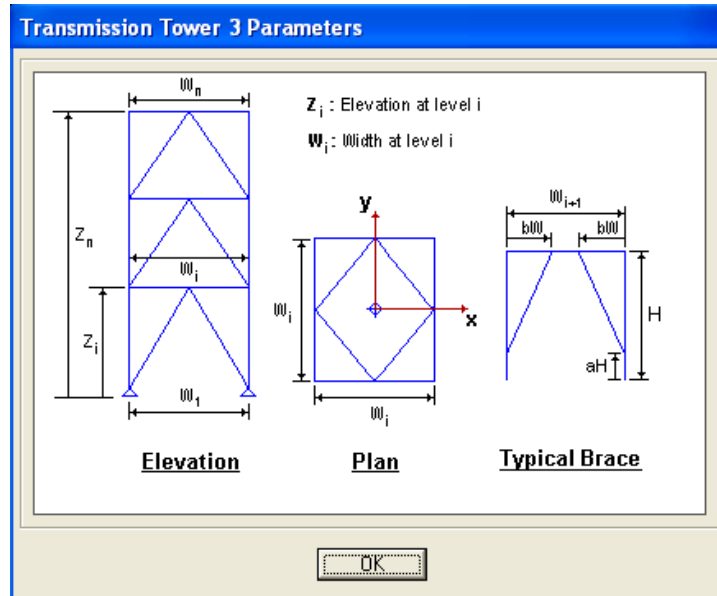


Figure 2-1 *Transmission Tower 3 Parameters*

Wind load Assignment

In the model, we will assign wind load using the ASCE 7-02 code. Under the Define Menu, click on the Load Cases option. Type in a load case name, select a WIND load type and click the Add New Load button. Highlight the WIND load case and click on the Modify Lateral Load button.

If ASCE 7-02 code was selected in the Auto lateral load drop-down box on the Define Loads form (the form that precedes use of this form) and the Exposure from Area Objects option is selected, two check boxes will be added in this area of the form: 1) Include Area Objects check box and 2) Include Frame Objects (Open Structure) check box. Since the transmission tower contains no area elements, choose the exposure from frame objects option.

Please refer to Figure 2-2 as a guide. The Wind Exposure Parameters will be disabled in this form. The Exposure Height is 80 ft. (maximum Z). For the Wind Coefficients, leave all of the default values for this example (Wind speed: 70 mph, exposure type: B). Click the OK button.

Figure 2-2 ASCE 7-02 Wind Loading Form

To view the ASCE 7-02 wind loads assigned to the structure, go to the Display menu and select Show Load Assigns>Frame/Cable. From the pull down menu, select the Wind load case and click on the Open Structure Wind Loads option in the bottom right hand corner of the screen. The loads will be displayed on all of the frame elements including the columns and braces. Please see Figure 2-3:

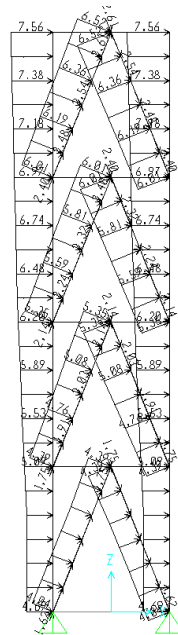


Figure 2-3 Open Structure Wind Load Display (Elevation)

Response Spectrum Analysis

A response spectrum function is simply a list of period versus spectral acceleration values. In ETABS the acceleration values in the function are assumed to be normalized; that is, the functions themselves are not assumed to have units. Instead the units are associated with a scale factor that multiplies the function and is specified when you define the response spectrum case.

Define response spectrum functions as follows:

Click the Define menu > Functions> Response Spectrum Functions command to access the Define Response Spectrum Functions form. The Response Spectra area of this form lists the names of all the currently defined response spectrum functions. Select the UBC97 spectrum and click OK.

After a response spectrum function has been defined, define a response spectrum case as follows:

Click the Define menu- > Analysis Cases >Add New Case> and from the pull down menu, select Response Spectrum. Click on the Modal Damping button on the bottom of the dialogue box. Enter the structural damping as .05. It is important that you understand the structural and function damping item. This item specifies modal damping that is present for all modes in the response spectrum analysis. Also, SAP2000 assumes that the response spectrum functions specified for the response spectrum case are all specified for this particular damping ratio.

Select the Complete Quadratic Combination (CQC) option. This is the CQC method described by Wilson, Kiureghian and Bayo (1981). This modal combination technique takes into account the statistical coupling between closely spaced modes caused by modal damping. Increasing the modal damping increases the coupling between closely spaced modes. If the modal damping is 0 for all modes, then the CQC method degenerates to the SRSS method.

Select the SRSS directional combination option. For each displacement, force or stress quantity in the structure, modal combination produces a single, positive result for each direction of acceleration. These directional values for a given response quantity are combined to produce a single positive result. SRSS combines the directional results by taking the square root of the sum of their squares. All other input items remain unchanged. The results obtained using this method do not vary regardless of the excitation angle that you specify. This is the recommended method for directional combination and is the default.

Under the Loads Applied area, specify any defined response spectrum function for each of the three local coordinate system directions of the response spectrum case. In this case,

the default spectrum name is *UNIFRS* unless changed in the previous response spectra definition dialogue box. Click Add and the OK to conclude the response spectrum input for this model.

Eigenvector and Ritz Analysis

Research has indicated that the natural free-vibration mode shapes are not the best basis for a mode-superposition analysis of structures subjected to dynamic loads. It has been demonstrated (Wilson, Yuan, and Dickens 1982) that dynamic analyses based on load-dependent Ritz vectors yield more accurate results than the use of the same number of natural mode shapes.

Ritz vectors yield excellent results because they are generated by taking into account the spatial distribution of the dynamic loading, whereas the direct use of the natural mode shapes neglects this very important information.

In addition, the Ritz-vector algorithm automatically includes the advantages of the proven numerical techniques of static condensation, Guyan reduction, and static correction because of higher-mode truncation.

The Analysis Case Data - Modal form is used to view and change the definition of a modal analysis case. A modal analysis is always linear. Choose between Eigenvector or Ritz vector modal analysis types. Eigenvector analysis determines the undamped free-vibration mode shapes and frequencies of the system. These natural modes provide an excellent insight into the behavior of the structure. Ritz vector analysis seeks to find modes that are excited by a particular loading. Ritz vectors can provide a better basis than do eigenvectors when used for response-spectrum or time-history analyses that are based on modal superposition.

Tip: The program automatically creates an eigenvector modal analysis case when a new model is started. Calculating and reviewing the first few natural modes (eigenvectors) of a structure is an excellent way to check the model for errors and to gain insight into its behavior.

Note: An infinite number of modal analysis cases can be created. However, for most purposes, one modal analysis case is enough.

EXAMPLE III **Truss Modeling – Curved Steel Truss Example**

Description

This is a 166 ft steel truss with curved upper and lower chord elements. The lower chord of the truss has a top elevation of 12 ft and the upper chord has a top elevation of 24 ft.

Significant Options of SAP2000 Exemplified

- Curved Frame Elements
- Edit Grid Options
- Truss Optimization Modeling
- Steel Frame Design
- Steel Frame Drift Optimization

Computer Model Definition

To create this model, go to the File menu>New Model and select the Grid Only option. Under the Cartesian Tab, fill in the number of Grid lines in the X-direction (2), Y-direction (1) and Z-direction (2). The grid spacing is 2000 inches in the X direction, 288 inches in the Y direction, and 72 inches in the Z direction. Click OK and the grid will be shown.

Curved Frame Element Generation

Go the Draw menu>Draw frame element and select the curved frame option in the line object type menu. Click on the grid intersection that defines the beginning point for the curved frame. Select the X-Z button on the top of the screen and click the point that is the origin of the X-Z plane. Then, click on the grid intersection to define the end of the curved frame at point X=2000 and Z= 0. When the mouse button is released, the Parameters for Curved Frame and Cables for Line Object form are displayed. Use the form to review/change the section property, number of linear segments, computed coordinates of the linear segments and so on. Fill in the parameters as shown in Figure

3-1 and click OK. Create a second curved but with the end points starting at X=0, z=72 and ending at X=2000 and Z=72. The 3rd

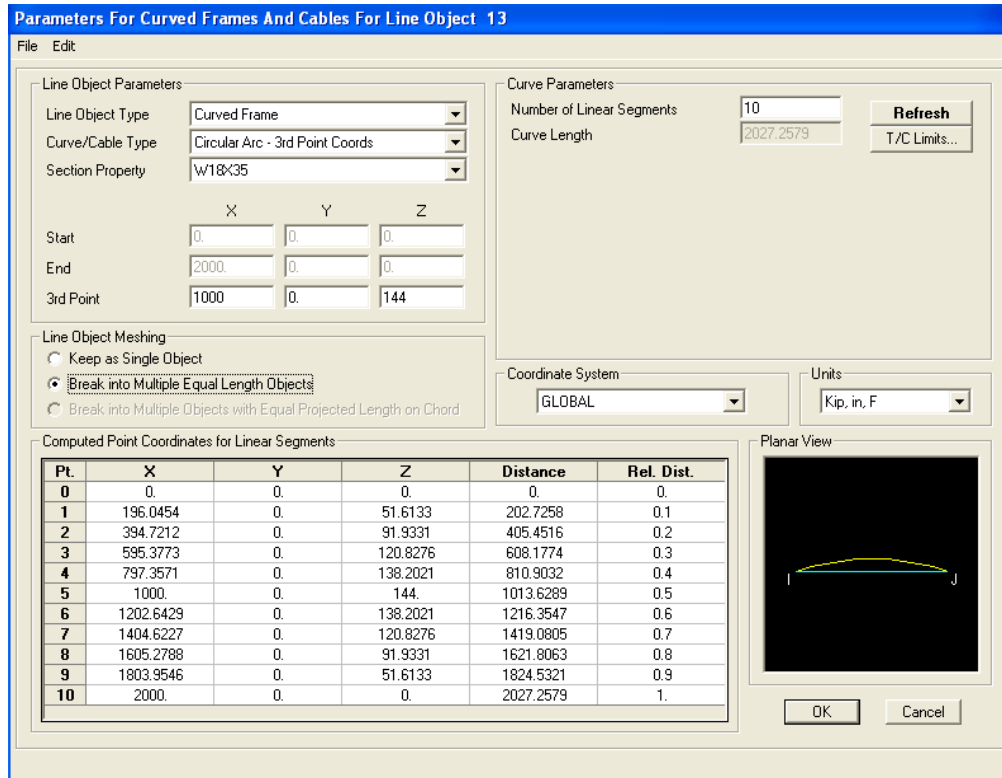


Figure 3-1 Curved Frame Parameters Dialogue Box

Create a second curve but this one with the end points starting at X=0, z=72 and ending at X=2000 and Z=72. The 3rd point for the second curve has coordinates (1000, 0, 288). You can see the 3rd point for the first curve in Figure 3-1. (1000, 0, 144)

Now, the 2 curves are shown in the X-Z plane. Under the View menu>Set Building Options, deselect the invisible button under the Joints box. You will now be able to see that the curved beams have been broken up into 10 segments each. Using the Draw>Draw frame element option, draw in a truss configuration as shown in Figure 3-2: Next, assign joint restraints to the two end joints. The left joint is pinned and the right joint is a roller restraint.

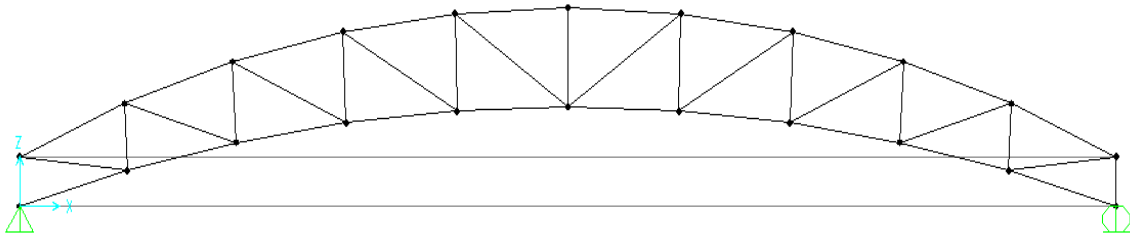


Figure 3-2 Curved Truss

Auto Select Lists (Truss Optimization)

Auto select section lists are lists of previously defined steel sections (including cold-formed steel). When an auto select section list is assigned to a frame member, the program can automatically select the most economical, adequate section from the auto select section list when designing the member. Auto Select Section Lists are used in Steel Frame, Aluminum Frame, and Cold-Formed Steel Frame design. At least two steel frame sections must be defined before you are allowed to define an auto select section list.

Define an Auto Select Section List as follows:

1. Click the Define menu > Frame Sections command to display the Frame Properties form.
2. In the "Click to" area of that form, scroll down the drop-down list that initially reads "Add I/Wide Flange" to locate and highlight "Add Auto Select." This will bring up the Auto Selection Sections form.
3. Use the Auto Selection Sections form to name your auto select section list and choose the sections to be added or removed from that list.
 - a. **Auto Section Name:** Type a name into the edit box.
 - b. **Choose Sections:** Locate the sections you wish to include in your auto select list in the *List of Sections* display box. Use standard Windows methods to select the desired sections (i.e., click on a section and hold down the left mouse button as you drag the mouse down the list to select multiple sections; highlight a section and click the left mouse button to select sections one at a time; hold down the Ctrl key and click the left mouse button to make random multiple selections). Click the Add button to add the selected beams to the Auto Selections list on the right side of the form. If you want to remove a section(s), highlight it in the *Auto Selections* list and click the **Remove** button. Removing a section from the Auto Selections list does not removed it from the List of Sections.

Next, select all of the members of the truss, go to the Assign>frame element>frame section box and select AUTO1 section list. Now, a specified list of sections is assigned to the truss elements.

Steel Displacement Targets

For displacement optimization, SAP2000 predicts which members should be increased in size to control the displacements based on the energy per unit volume in the members. The members with more energy per unit volume are increased in size a larger percentage than those with smaller energies per unit volume. Some members with small energy per unit volume may be decreased in size if they are still acceptable for strength considerations.

Auto Select Lists must be assigned to the frame elements for the displacement optimization to work. When SAP2000 increases or decreases a section size, it uses the available sizes in the auto select list. If the appropriate sections are not included in the auto select lists, you may never reach your displacement target no matter how many times you iterate by rerunning the analysis and design.

Specify a displacement optimization specification as follows:

1. Click the Design menu > Steel Frame Design > Set Lateral Displacement Targets command to open the Lateral Displacement Targets form where you can specify displacement targets for various load cases. Our intent is that you pick a point, typically at the roof level of your building, and specify a maximum displacement target (in any direction) for one or more load cases.
2. In the Loads area of the Lateral Displacement Targets form, select the Load Case for which you want to optimize the displacement by selecting it from the drop-down list.
3. Specify the location for the displacement by specifying a Point.
4. Specify the target displacement.
5. Click the Add button.
6. Repeat Steps 1 through 5 as needed.
7. Rerun your analysis with the new section sizes to review the displacements. Displacement optimization is an iterative process that typically requires you to rerun the analysis and design multiple times. There is no guarantee that you will reach your displacement target using this option.

Now, before the model is ready to be analyzed, we must set the analysis options. Go to Analyze> Set Analysis options. Select the Plane Frame option, which restricts the available degrees of freedom for a 2-D frame. Next, go to Analyze>Run Analysis to bring up the Analysis cases to run form. The Analyze menu > Set Analysis Cases to Run and Analyze menu > Run Analysis commands both open the Set Analysis Cases to Run form. Use this form to:

- Check the analysis status of the cases
- Delete results for cases that have already been run
- Set which cases are to be run
- Run the analysis, or just save the settings

The table on the form lists all defined analysis cases, showing:

Case Name: The name defined for an analysis case.

Type: The type of analysis case.

Status: Shows the status of the case following its most recent analysis. Values are:

- **Not Run:** The case has never been run or its results have been deleted
- **Could Not Start:** An attempt was made to run the case, but a prerequisite case was did not finish
- **Not Finished:** The analysis was unable to reach its requested final state
- **Finished:** The analysis completed successfully

Action: A value set by the user indicating what should happen the next time an analysis is run:

Run: Indicates that SAP2000 should run the analysis case.

Do Not Run: Indicates that SAP2000 should not run the analysis case.

Running Cases

Use the Set Analysis Cases to Run form to change the action for individual cases or for all cases simultaneously. If a case is set to Run, all prerequisite cases will also be run, even if they have been set to Do Not Run.

Click the Run Now button and the analysis will begin and all cases set to run and their prerequisite cases will be run. Click the OK button and the action settings will be saved but the analysis will not run. Click the Cancel button and the action settings will be left unchanged.

Run the Analysis to see the deflected shape of the truss. See Figure 3-3:

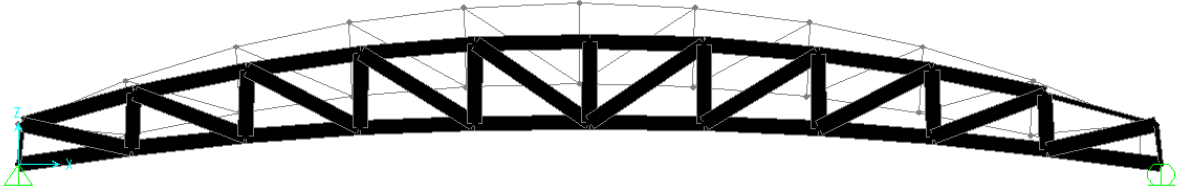


Figure 3-3 Deflected Shape of Curved Truss (extruded view)

Steel Frame Design

The Design menu > Steel Frame Design > Start Design/Check of Structure command starts the steel frame design process.

Run a steel frame design as follows:

1. Click the Analyze Menu > Run Analysis command to run an analysis of the model. You must analyze the model before you can design it. Thus, the Design menu > Steel Frame Design > Start Design/Check of Structure command is not available if you do not first run an analysis.
2. Click Design menu > Steel Frame Design > Start Design/Check of Structure command.

Note:

This command is unavailable if no frame elements with the Steel Frame design procedure have been included in the model.

If you have selected frame elements when you click this command, only the selected frame elements are designed. If no frame elements are selected when you click this command, all steel frame elements with the Steel Frame design procedure are designed.

Review some of the results of the steel frame design directly on the SAP2000 model (i.e., on screen) using the Design menu > Steel Frame Design > Display Design Info command. This command displays Design Input and Output parameters.

Display design information on screen as follows:

1. Run an analysis.
2. Perform at least one design using the Design menu > Steel Frame Design > Start Design/Check of Structure command.

3. Select the Design menu > Steel Frame Design > Display Design Info command to access the Display Design Results form.
4. Check the *Input* or *Output* check box and select from the drop-down list the type of information to be displayed on screen.
 - Right click on a frame element to display the Steel Stress Check Information Form, which also can be displayed using the Interactive Steel Frame Design command. See Figure 3-4:

Steel Stress Check Information (AISC-LRFD93)

Frame ID: 34 Analysis Section: W27X94
 Design Code: AISC-LRFD93 Design Section: W8X24

COMBO ID	STATION LOC	---MOMENT RATIO	=	AXL + B-MAJ + B-MIN	INTERACTION CHECK-----	//-MAJ-SHR---MIN-SHR-	RATIO	RATIO
DSTL1	0.00	0.560 (C)	=	0.560 + 0.000 + 0.000		0.000	0.000	
DSTL1	72.00	0.554 (C)	=	0.554 + 0.000 + 0.000		0.000	0.000	
DSTL1	144.00	0.549 (C)	=	0.549 + 0.000 + 0.000		0.000	0.000	
DSTL2	0.00	0.480 (C)	=	0.480 + 0.000 + 0.000		0.000	0.000	

Modify/Show Overwrites: Overwrites
 Display Details for Selected Item: Details
 Display Complete Details: Tabular Data

Current Stylesheet: Program Default

OK Cancel Table Format File

Figure 3-4 Steel Stress Check Information

EXAMPLE IV **Bridge Analysis – Concrete Box Girder Bridge Example**

Description

This example demonstrates the powerful bridge module in SAP2000. The model is a concrete box girder bridge with a 220 ft span and 2 lanes. The bridge has 3 columns with different heights supporting the deck at midspan. There are parametric variations along the length of the bridge as well as prestressed tendons assigned to the deck. The bridge abutments are skewed 15 degrees at the 2 ends of the bridge deck.

Computer Model Definition

The overall deck depth has a depth 5 ft and a width of 36 ft. Kip-feet-second units are used. To see the deck cross-section geometry, please refer to Figure 4-1.

Other parameters associated with the structure are as follows:

Clear span of bridge	= 220 ft
Overall depth of deck	= 5 ft
Width of deck, b	= 36 ft
Concrete strength, f'_c	= 4000 psi
Yield strength of steel, f_y	= 60000 psi
Concrete unit weight, W_c	= 150 pcf
Modulus of elasticity, E_c	= 3600 ksi
Modulus of elasticity, E_s	= 29000 ksi
Poisson's ratio, ν	= 0.2

Significant Options of SAP2000 Exemplified

- Bridge Analysis
- Influence Lines and Surfaces
- Use of Prestress Tendons

To build the bridge model, a 12-step process is required. Follow the steps listed below to build a bridge model based on user-specified definitions. The user must define the following parameters:

1. Layout Lines
2. Deck Sections
3. Abutments
4. Bents
5. Diaphragms
6. Hinges
7. Parametric Variations
8. Bridge Object definitions
9. Update of linked model
10. Lanes
11. Vehicles/ Vehicle Classes
12. Analysis Cases

The user can quickly define a basic model that applies program defaults using the following abbreviated approach:

- a. Define a layout line using Step 1.
- b. Define a deck section using Step 2.
- c. Skip to Step 8 to create a bridge object.
- d. Create a linked model using Step 9.

For the abbreviated approach, SAP2000 will apply default abutment, bent, hinge, and diaphragm properties. If necessary, Steps 3, 4, 5 and 6 of this Wizard can be used to change those default definitions. In addition, prestressed tendons can be added as part of the bridge object definition (see Step 8).

Each one of the 12-steps is described in great detail.

Step 1: Layout Lines

The first step in creating a bridge object is to define the layout line. Layout lines are used as reference lines for defining the vertical and horizontal layout of bridge objects and lanes.

Layout lines are defined in terms of stations, bearings and grades. The lines may be straight, bent or curved both in the horizontal and the vertical plane. Horizontal curves are circular (with spirals if necessary) and vertical curves are parabolic. In this example, the End Station is defined as 220 ft.

Use the Quick Start options to quickly define a layout line. You will see the many choices available for both Horizontal and Vertical curves. Select the Straight curve in both cases.

Step 2: Deck Section Definition

Various parametric bridge deck sections are available for use in defining a bridge. They include concrete box girders, concrete beam and steel beam sections. Select the External Girders Vertical option. Enter the total width and depth shown in Figure 4-1 below. After a deck section has been defined it can be assigned to a span as part of the bridge object definition (see Step 8).

Define Bridge Section Data - Concrete Box Girder - Vertical

Section Data

Item	Value
General Data	
Bridge Section Name	BSEC1
Material Property	CONC
Number of Interior Girders	2
Total Width	36.
Total Depth	5.
Slab and Girder Thickness	
Top Slab Thickness (t1)	1.
Bottom Slab Thickness (t2)	0.6667
Exterior Girder Thickness (t3)	1.
Interior Girder Thickness (t4)	1.
Fillet Horizontal Dimension Data	
f1 Horizontal Dimension	1.5
f2 Horizontal Dimension	1.5
f3 Horizontal Dimension	0.5
f4 Horizontal Dimension	1.5
f5 Horizontal Dimension	1.5
f6 Horizontal Dimension	0.5

lb, ft, F

OK

Cancel

Figure 4-1 Define Bridge Section Data Form

Step 3: Abutment Definition

Abutment definitions specify the support conditions at the ends of the bridge.

An abutment definition can be a specified Link/Support property or it can be a user defined support condition. The user support condition allows each of the six degrees of freedom at the abutment to be specified as fixed, free or partially restrained with a specified spring constant.

An abutment definition also allows the horizontal location of the abutment supports to be specified. A single abutment support can be located at the reference line location or multiple abutment supports can be located either at each girder or equally spaced over the bridge width. When multiple locations are indicated the specified abutment support properties are provided at each support location.

It is also possible to specify that a closure (vertical diaphragm) of some thickness is to be provided at the abutment. This closure is only applicable to area object and solid object models.

For this example, select the U2 direction to have a 'Free' release type. The other directions should have a 'Fixed' release type. Under the Horizontal Location of Abutment Supports, select the every girder location option.

Step 4: Bent Definition

Bent definitions specify the geometry and section properties of the bent cap beam and the bent column(s). They also specify the base support condition of the bent columns.

The specified base support condition for a bent column can be Fixed, Pinned or a user defined column support. A user defined column support can be a specified Link/Support property or it can be a user defined support condition. The user support condition allows each of the six degrees of freedom at the column base to be specified as fixed, free or partially restrained with a specified spring constant. The user defined column support is defined separately from the bent.

It is also possible to specify that a vertical diaphragm is to be provided at the bent location. The diaphragm is only applicable to area object and solid object models. It does not apply to spine models.

After a bent is defined it can be assigned to the bridge as part of the bridge object definition (see Step 8).

In this example, click on the Bridge menu > Bents and select the Add New Bridge Bent option. In the Bent Data box, type in the number of columns: 3. Next, click on the Modify/Show Column Data box in the lower left hand corner. Fill out the form as shown in Figure 4-2 below and click OK. Make sure you are in Kip-ft units.

Bridge Bent Column Data

Bridge Bent Name: BENT1 Units: Kip, ft, F

Column	Section	Distance	Height	Angle	Base Support
1	FSEC1	2.5	22.	0.	Fixed
2	FSEC1	15.	25.	0.	Fixed
3	FSEC1	27.5	28.	0.	Fixed

Notes:

1. The distance is measured from the left end of the cap beam to the center of the column.
2. The column height is measured from the top of the cap beam to the bottom of the column.
3. The column angle is measured in degrees counterclockwise from a line parallel to the bent to the column local 2 axis.

Moment Releases at Top of Column

Column	R1 Release	R2 Release	R3 Release	R1 Stiffness	R2 Stiffness	R3 Stiffness
1	Fixed	Fixed	Fixed			
2	Fixed	Fixed	Fixed			
3	Fixed	Fixed	Fixed			

OK Cancel

Figure 4-2 Bridge Bent Column Data Form

Step 5: Diaphragm Definition

Diaphragm definitions specify properties of vertical diaphragms that span across the bridge.

A diaphragm property can be solid concrete; steel X, V or K bracing; or a single steel beam. Solid concrete diaphragm properties are only applicable to concrete bridge sections. Steel diaphragm properties are only applicable to steel bridge sections.

Diaphragms in general are only applicable to area object and solid object models. They do not apply to spine models.

After a diaphragm definition has been created it can be assigned to one or more spans in the bridge object (see Step 8).

It is not necessary to define a diaphragm property before defining a bridge object. If no diaphragms are defined when a diaphragm is first added to a bridge object, the program automatically creates a default diaphragm property. For this example, we will not assign specific diaphragm properties.

Step 6: Hinge Definition

Hinge definitions specify properties of hinges (expansion joints) and restrainers.

A hinge property can be a specified Link/Support property or it can be a user-defined spring. The user spring allows each of the six degrees of freedom at the hinge to be specified as fixed, free or partially restrained with a specified spring constant.

A restrainer property can be a specified Link/Support property or it can be a user-defined restrainer. The user restrainer is specified by a length, area and modulus of elasticity.

A hinge definition also allows the horizontal location of the hinge springs and restrainers to be specified. A single hinge spring (and restrainer) can be located at the reference line location or multiple hinge springs (and restrainers) can be located at each girder or equally spaced over the bridge width. When multiple locations are indicated the specified spring and restrainer properties are provided at each support location.

It is also possible to specify that a vertical diaphragm is to be provided at the hinge location. The specified diaphragm is provided on each side of the hinge. This diaphragm is only applicable to area object and solid object models. It does not apply to spine models.

After a hinge definition has been created it can be assigned to one or more spans in the bridge object (see Step 8). It is not necessary to define a hinge property before defining a bridge object. If no hinges are defined when a hinge is first added to a bridge object, the program automatically creates a default hinge. That is the case for this example.

Step 7: Parametric Variation Definition

Parametric variations define variations in the deck section along the length of the bridge. Any parameter used in the parametric definition of the deck section can be specified to vary. One or more of the parameters can vary at the same time. Each varying parameter can have its own unique variation.

Example uses of parametric variations include varying the bridge depth and the thickness of girders and slabs along the length of the bridge. The variations may be linear, parabolic or circular.

After a variation has been defined it can be assigned to spans in the bridge object (see Step 8). When a variation is defined it should be defined with the same length as the bridge span to which it is assigned.

For this example, we will define 2 variations (one for each side of the bridge.) Under the Bridge Menu>Parametric Definitions>Add New Variation, select the quick start button and select the Parabolic Linear variation and click OK. Fill out the form as shown in Figure 4-3 below:

Variation Definition

Variation Name: PVAR1 Units: Kip, ft, F

Point ID	Segment Type and Point Type Segment Is From Point(n - 1) to Point(n)	Distance ft	Dim. Change ft	Slope ft / ft
3		100.	5.	
1	Start of Span	0.	0.	
2	Parabola End Point and Initial Slope	90.	5.	0.
3	Linear to End of Span	100.	5.	

Variation Sketch: Use Equal Horizontal And Vertical Scales In Sketch

Dimension Change Sign:

Distance: Dim. Change: Slope:

Figure 4-3 *Parametric Variation Definition Form*

Next, follow the same steps from above to create a 2nd variation. Click on the Add Copy of Variation button in the Define Variations dialogue box. Go to Quick Start and click on the Linear Parabolic option (the exact opposite of the previous variation). Name this variation *PVAR2* and fill in the Distance and Dimension Change boxes as shown in Figure 4-4:

Variation Definition				
Point ID	Segment Type and Point Type Segment Is From Point(n - 1) to Point(n)	Distance ft	Dim. Change ft	Slope ft / ft
1		0.	5.	
1	Start of Span	0.	5.	
2	Linear	10.	5.	
3	Parabola End Point and Final Slope	100.	0.	0.
4	Linear to End of Span	100.	0.	

Figure 4-4 Parametric Variation Definition Form

Step 8: Bridge Object Definition

The bridge object is the heart of the bridge modeler. The following is included in the bridge object definition:

- a. The bridge spans are defined.
- b. Deck section properties are assigned to each span.
- c. Parametric deck section variations may be assigned to each span.
- d. Abutment properties and skews are assigned.
- e. Bent properties and skews are assigned.
- f. Hinge locations, properties and skews are assigned.
- g. Super elevations are assigned.
- h. Prestressed tendons are defined.

Any time a bridge object definition is modified the linked model must be updated (see step 9) for the changes to appear in the SAP2000 model.

The prestress tendon quick start options allow quick and easy layout of prestressed tendons. The prestress tendon parabolic calculator makes quick work of the layout of parabolic prestress tendons.

Click on the modify/show bridge object in the Define Bridge Object menu. In the Define Bridge Object Reference Line box, fill in the parameter as shown in Figure 4-5:

Define Bridge Object Reference Line

Span Label	Bridge Object Span Type	Layout Line Name	Station ft	Item Label
SpanStart		BLL1	10.	StartAbt
SpanStart	Start Abutment	BLL1	10.	StartAbt
Span1	Span to Bent	BLL1	110.	Bent2
SpanToEnd	Span to Abutment	BLL1	210.	EndAbt

Notes: 1. Bridge object location is based on bridge section insertion point following specified layout line.

Figure 4-5 Bridge Object Reference Line Form

Next, click the on Modify/Show spans button. In this dialogue box, for Span 1, double click on the span varies box. A Bridge section variation box will open. Double-click on the variation for Total Depth box and select PVAR1 and click OK. Do the same for the next span except select PVAR2 for the variation.

Click on the Modify/Show Bridge Prestress button and select Add new Tendon. Fill in a tendon area of 10 in² and load force of 1500 kips. Select a Prestress load case. (To create a Prestress load case, go to Define>Static Load Cases dialogue box.) Click on the Quick Start button for vertical layout and select parabolic tendon 1 and click Ok twice. Click on the Copy to all girders options, which will create 4 total tendons.

Step 9: Update Linked Model

The update linked model command creates the SAP2000 object-based model from the bridge object definition. If an object-based model of the bridge object already exists, it will be deleted when the new object-based model is created using all of the latest changes to the bridge object definition.

Spine models, area object models and solid object models of the bridge can be created when the linked model is updated. The type of object-based model created from the bridge object definition can be switched at any time.

Under the Bridge menu, select the Update Linked Bridge Model option. Then click on the Update as Area Object option. Your model should look like Figures 4-6 and 4-7:

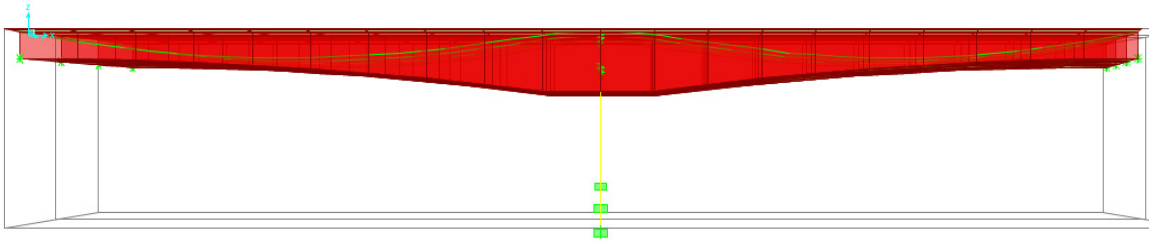


Figure 4-6 *Bridge Elevation*

In Figure 4-6, you can see the parametric variation along the length of the deck. You can also see the parametric tendons located inside the bridge deck.

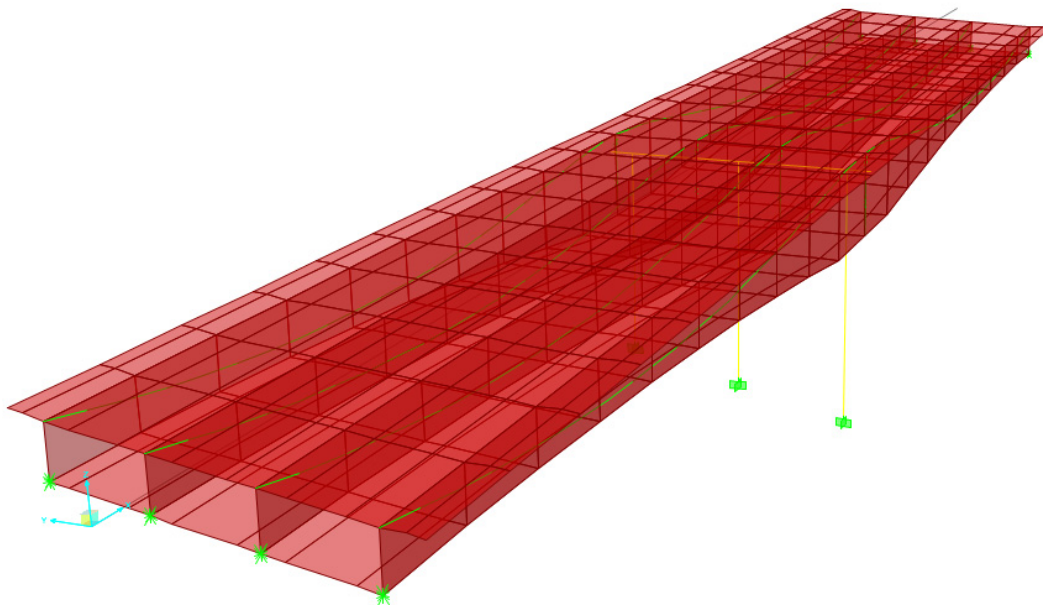


Figure 4-7 3D View of *Bridge*

Step 10: Lane Definition

Lanes must be defined if you want to analyze your bridge for vehicle live loads.

Lanes can be defined with reference to either layout lines or existing frame objects. A single lane is referenced to one or more layout lines or one or more frame objects but not a combination of layout lines and frame objects. Lanes can be defined with width if desired. Lanes are used in the definition of Moving Load type analysis cases and in Moving Load type load cases which can be used in static and dynamic multi-step analysis cases.

For this example, click on the Add New Lane Defined from Layout Line button. Add a lane at two stations. (0 ft and 220 ft) Each of these stations has the same centerline offset (7ft) and lane width (14ft). Click OK. Next, add a copy of a lane and change offset by specified amount (-14ft).

Step 11: Vehicle Definition

Vehicles must be defined if you want to analyze your bridge for vehicle live loads. In SAP2000 vehicles loads are applied to the structure through lanes.

If you plan to use a moving load type analysis case then you must also define one or more vehicle classes. A vehicle class is simply a group of one or more vehicles for which a moving load analysis is performed (one vehicle at a time).

Numerous standard vehicle definitions are built into the program. In addition the General Vehicle feature can be used to create your own vehicle definition. Each vehicle definition consists of one or more concentrated and/or uniform loads.

Under the Bridge menu, select vehicles and click the Add Vehicle button. Add an HSN-44-1 type vehicle and click OK. From the Bridge menu again, select the Vehicle Classes option. Click Add New Class and select the HSN-44-1 vehicle and click Add.

Step 12: Analysis Cases

Although any analysis case type can be used when analyzing your bridge, there are several analysis options that are specialized for analysis of vehicle live loads.

Moving load analysis cases compute influence lines for various quantities and solve all permutations of lane loading to obtain the maximum and minimum response quantities.

Multi-step static and multi-step dynamic (direct integration time history) analysis cases can be used to analyze one or more vehicles moving across the bridge at any speed. These multi-step analysis cases are defined using special moving load type load cases that define the direction, starting time and speed of vehicles moving along lanes.

Under the Define menu>Analysis Cases> click add new case. Under the analysis case type, select Moving Load and add the VECL1 vehicle class and click OK.

Creep and Shrinkage

Under the Define>Material Properties> select Concrete and click Modify/Show Properties. Select the Time Dependent Properties and click on the Creep and Shrinkage option.

- Time Dependent Properties for Concrete. Use the Time Dependent Properties for Concrete form to specify the *Strength and Modulus of Elasticity/CEB-FIP* coefficient, as well as the *Creep* and *Shrinkage* factors for the material definition.

After the time dependent properties have been specified as part of the material property definition, the time dependent properties are applied to the model by assigning the material property definition (that includes the time dependent properties) to a selected area(s), frame(s), cable(s), tendon(s) or solid(s).

Results

The influence lines can be displayed for the various displacements, reactions, forces, moments, shears, torsion or axial loads on joints, frames, shells, planes, solids, solids, and links resulting from a unit load on a defined bridge lane in the structure.

After lanes have been defined and a moving analysis case has been defined and run, select a column and click the Display menu > Show Influence Lines/Surfaces command to display the Show Influence Lines/Surfaces form. See Figure 4-8:

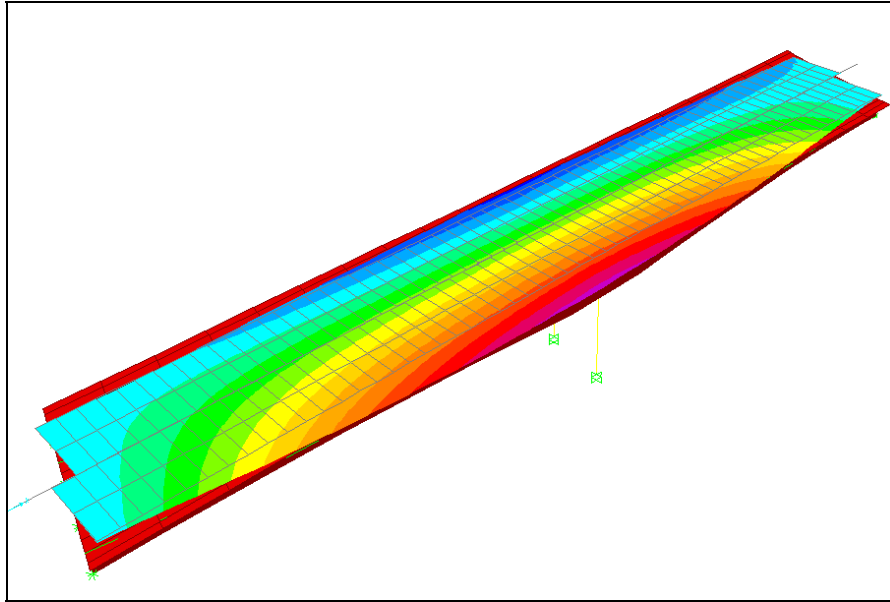


Figure 4-8 *Influence lines on Deck*

You can view Bridge forces and stresses for any load case. Go to Display>Show Bridge Forces/Stresses to display the objects response display form. Click the stresses option, select the prestress load case and select the longitudinal stress –Top and Bottom Right – S11. See Figure 4-9 to see what is shown:

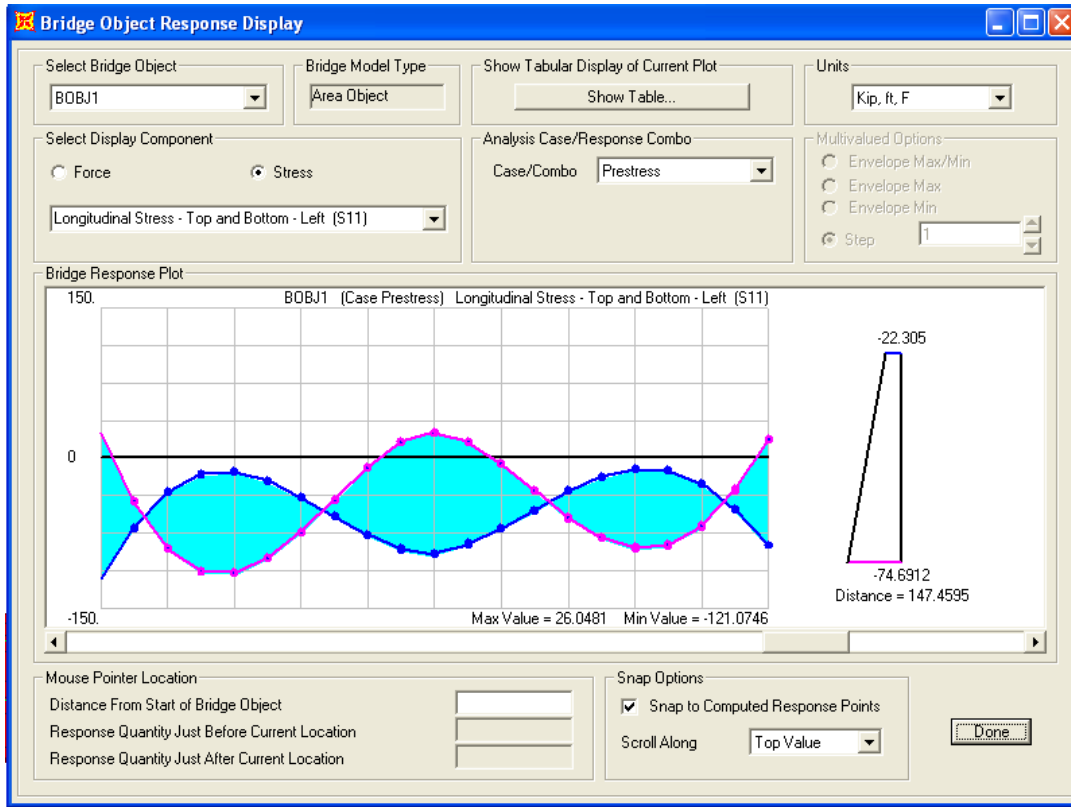


Figure 4-9 *Bridge Object Response Display*

Section Cut Forces

There are two options available to define Section Cuts:

1. The first option is to define the location of the cut. Use the Define menu > Section Cuts command to obtain resultant forces acting at section cuts through a model. Define section cuts before or after an analysis is run; however, it is safest to wait until after the analysis has been run. Typically do not define section cuts, and more importantly, the groups used in the section cut definition, until all manual meshing of the model (if any) has been completed. If the groups are defined before manual meshing, some of the point objects that should be in the group may not yet be created.
2. The second option is to manually draw the section cut on any portion of the model. This can be by utilizing the Draw>Draw Section Cut option. You must make sure that the model has been analyzed and you are viewing a member force/stress diagram. This can be found under Display> Show Member Force/Stress Diagram> select either frame or shell forces. This second option is used in this example to obtain section cuts forces.

To obtain shell stress forces on the bridge deck, go to Draw > Draw Section Cut. Draw a line parallel to a Grid line. The flashing line represents the section cut. Section Cut forces will now be visible on the screen. Please refer to Figure 4-10:

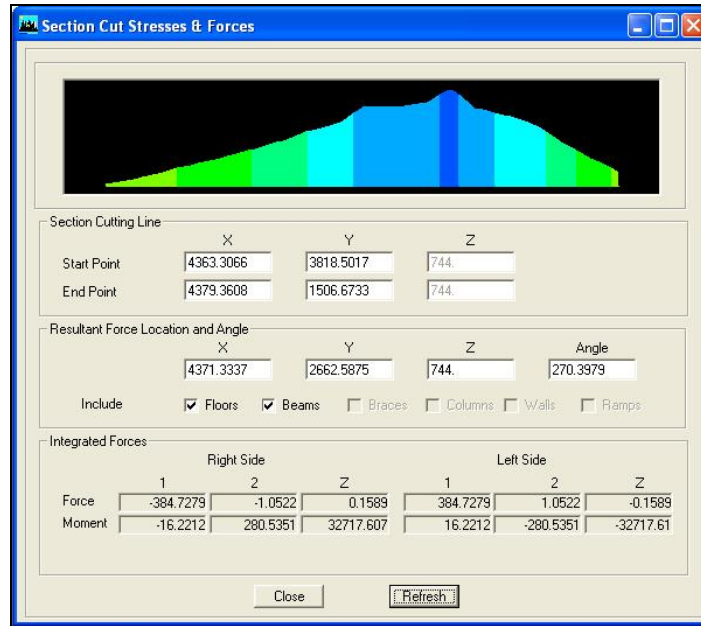


Figure 4-10 Section Cuts force output

EXAMPLE V **Advanced Modeling Techniques – Eight Story Parking Garage Structure**

Description

This is an eight-story parking garage structure with curved ramps located at each level. Each floor has a 12 ft story height. Each bay is 24 ft. long in the X-direction and 36 ft. long in the Y-direction. The structure is laterally supported by a curved shear wall system located at the edge of the ramp system.

Significant Options of SAP2000 Exemplified

- ❑ Replication Features
- ❑ Extrude Lines to Areas Feature
- ❑ Straight and Spiral Parking Garage Ramp Generation
- ❑ Draw Developed Elevations
- ❑ Creation of OpenGL Views
- ❑ Concrete Floor Diaphragm Shears & Section Cuts
- ❑ Element Property Modification Factors

Computer Model Definition

Grid and Line Generation

The structure is an 11-column line, 5-bay system. Kip-inch-second units are used. Spacing in the X and Y direction is 24 and 36 ft. respectively. It is an 8-story structure with 12 ft story heights.

To create this model, go to X-Z plane at $y=864$. Draw a line from Grid line A to Grid line C. Select this line, go to Edit> Replicate> $dz = -288\text{in}$ and under increment data, enter 3. Draw a line from Grid line I to Grid line K and replicate this line on all *odd* story levels. Next, draw a line starting at level 8 diagonally down to level 7 connecting each line at every level. (In this example, it does not matter which beam size you select). Currently, the model should look like Figure 5-1:

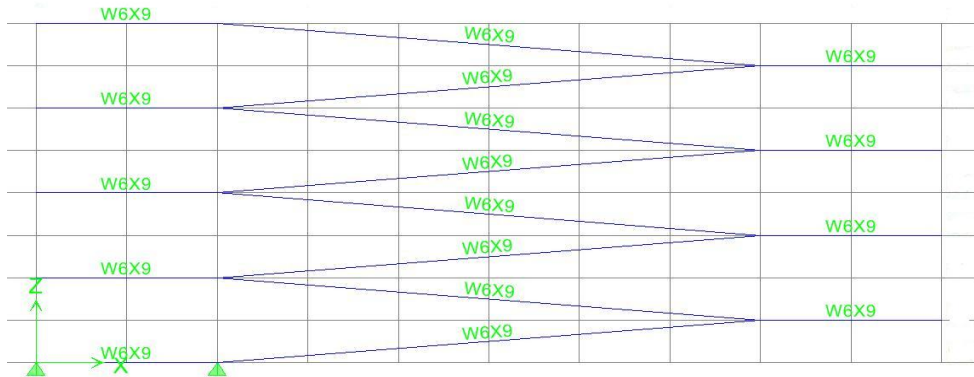


Figure 5-1 *Elevation of Ramp Structure*

Ramp Generation

Next, select all flat beams on every level and every *other* inclined beam element. Go to Edit > Extrude Lines to Areas > Linear. In the Increment Data box, enter 36 ft. for dy and number of replications = 2. Now, select all of the flat beams again and the inclined beams that were *not* selected the first time. Perform the same extrusion application except use -36ft for dy. Your model should now look like 5-2:

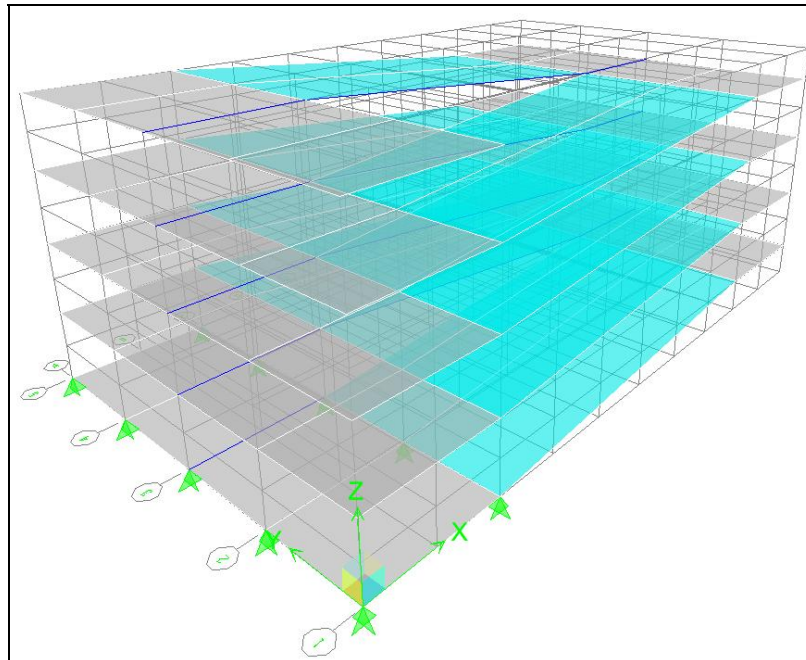


Figure 5-2 *3-D View of Ramp Structure*

Circular Edge Ramp Generation

Now, we are ready to create our circular ramp. Go to Elevation A, and draw a line from a point in between Grid line 3 and 4 to Grid line 5. (Make sure snap to lines and edges is activated under Draw> Snap to > Line and Edges). Replicate this line on all even floors. Select all lines in Elevation A, go to Edit> Extrude lines to Areas> Radial tab. The ramp is a semi-circle so enter 10-degree angles extruded 18 times creating a 180-degree curved surface. The total drop of the ramp is 2 stories or 288 in.

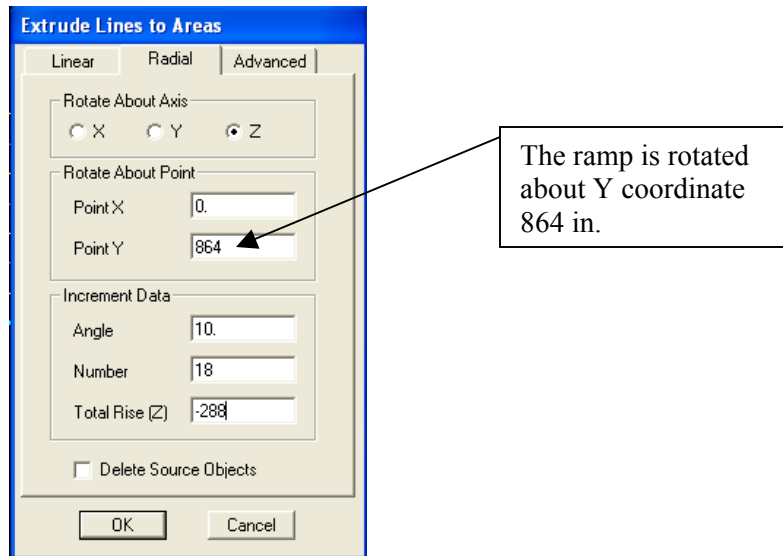


Figure 5-3 *Extrude Lines to Areas Box*

After completing this step, your model should look like Figure 5-4:

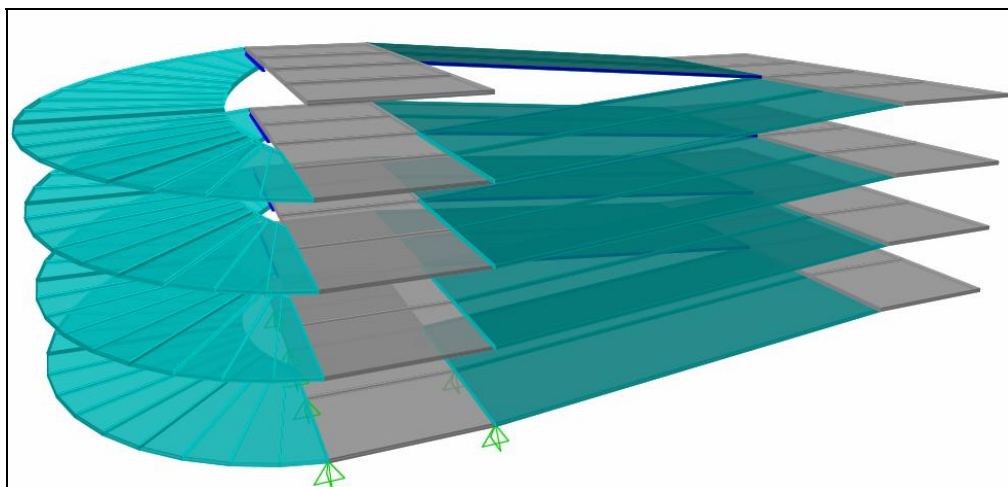


Figure 5-4 *Circular Ramp Structure*

Developed Elevation

Developed elevations can simultaneously show multiple faces of the building in a single "unfolded" elevation view. To draw a developed elevation, from the menu select Draw> Draw Developed Elevation Definition, type a name for the developed elevation in the edit box in the Developed Elevations area of the form and click on 'Add New Name'. Note: Make sure to set the 2D view-cutting plane to 300 in. This is located under the Options>Preferences menu. Next, in floor plan 8, click on the edge joints of the ramp until you've created a semi-circle shaped developed elevation. To view the elevation, click the View menu > Set elevation view and select the name of your Developed Elevation.

Note: If the Options menu > Window command is set to two or more windows and those windows are set to Plan or 3D View, the location of the developed elevation is illustrated with a cyan line (default) in the Plan and 3D view windows.

Curved Shear Wall Generation

Now, we are interested in creating a curved shear wall surface around the edge of the circular ramp surface, which will run the full height of the structure. Go to the View>Set Named View menu to open the developed elevation. Next, select the Draw>Quick draw area button. Draw the shear wall over the entire length and width of the developed elevation. Click on a couple of wall elements and delete them. This will create an opening in the curved shear wall element. The 3-D structure should look like Figure 5-5:

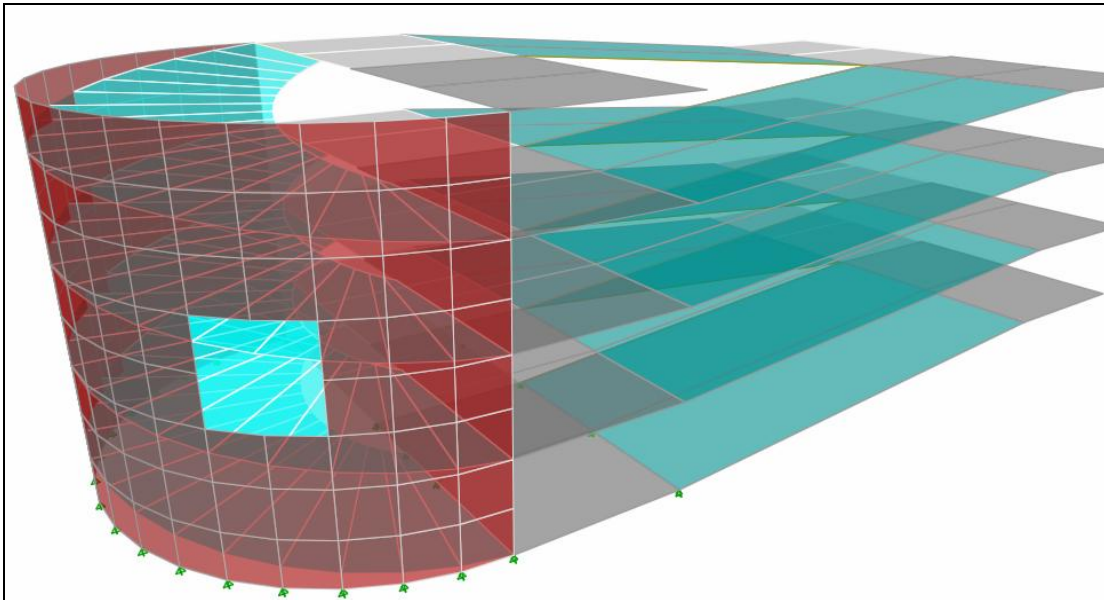


Figure 5-5 *Circular Ramp Structure with Curved Shear Wall*

EXAMPLE VI 2-D Frame Analysis

Description

This is a two-story frame analysis, subjected to lateral static loading. There are 2 bays with 12 ft. spacing in the X-Z plane. The brace elements are W18x35 steel sections. To see the system geometry, please refer to Figure 6-2:

Significant Options of SAP2000 Exemplified

- ❑ Tension/ Compression Limits
- ❑ Section Designer Module
- ❑ Quick Draw Brace Elements
- ❑ Nonlinear Load Cases

Computer Model Definition

The 2-D frame is two stories high with a story height of 12 ft. Each bay is spaced 24 ft. apart. Kip-inch-second units are used. The modulus of elasticity used for steel is 29000 ksi.

Other parameters associated with the structure are as follows:

Coefficient of Thermal Expansion	= 6.500E-06
Poisson's ratio	= 0.3
Shear Modulus	= 11153 ksi
Bending Reinforcement Yield Stress, f_y	= 60 ksi

Beam/Brace Properties:

W18x35 Wide Flange Steel Section

Building Loads:

Lateral Load on Roof level (Load applied in x direction)	= 10 kips
---	-----------

To create this model, go to the File menu->New Model and select the 2-D frame option. Fill in the 2-D frame input as shown in Figure 6-1. Click OK and the model should appear on the screen.

The image shows a software dialog box titled "2D Frames". It is divided into several sections. On the left, there is a "2D Frame Type" dropdown menu set to "Portal" and a small diagram of a portal frame. The "Portal Frame Dimensions" section contains input fields for "Number of Stories" (2), "Story Height" (144), "Number of Bays" (2), and "Bay Width" (288). Below these is a checkbox for "Use Custom Grid Spacing and Locate Origin" and an "Edit Grid..." button. The "Section Properties" section has dropdown menus for "Beams" and "Columns", both set to "W18x35". At the bottom, there is a checked checkbox for "Restraints", and "OK" and "Cancel" buttons.

Figure 6-1 2-D Frame Input Data Form

Drawing Brace Elements

1. Click on the Quick Draw Braces button or use the Draw menu >Quick Draw Braces command to display the Properties of Object form and switch to the drawing mode. Select the Property (W18x35), Moment Release (Pinned) and Bracing Type (X-bracing) from the form.
2. Click in a grid space, bounded by 4 grid lines, to draw a quick brace.
3. Click on all bays to create X-bracing in all 4 locations. The model should look like Figure 6-2:

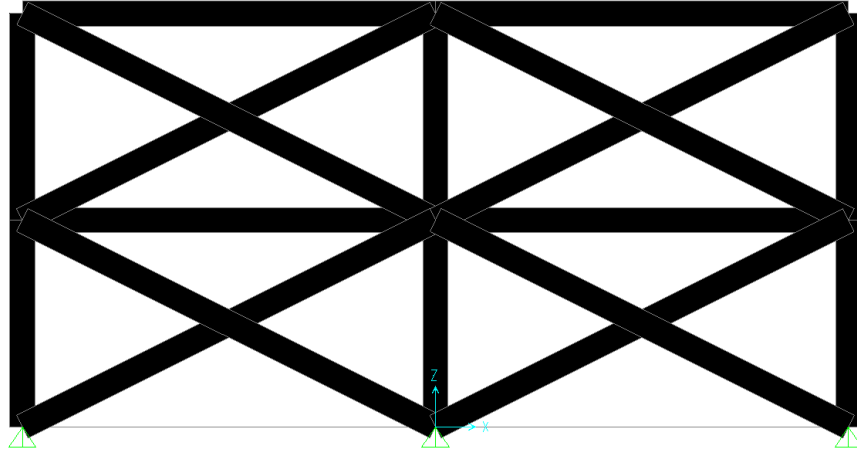


Figure 6-2 Extruded View of 2-D Frame

Load and Analysis Case Definition

Go to Define>Load Cases and create a load case called LATX with an OTHER load type and a Self Weight Multiplier of 0. Click Add New Case and click OK.

Next, go to Define>Analysis cases and click on the LATX load case and select Modify/Show Case. Under the Analysis Type box, select the Nonlinear option. The analysis case data box should look like Figure 6-3:

Load Type	Load Name	Scale Factor
Load	LATX	1.
Load	LATX	1.

Figure 6-3 Analysis Case Data Form

Select the top left joint of the 2-D frame. Go to Assign>Joint Loads>Forces and select the LATX load case and assign a 10 kip load in the X-direction.

Tension/Compression Limits

An upper limit on the amount of tension and compression force supported by a frame/cable/tendon object can be assigned. This is used primarily to model tension-only cables and braces. The behavior modeled is nonlinear but elastic. For example, assume a compression limit of zero has been set. If the object tries to go into axial compression, it will shorten without any stiffness. If the load reverses, it will recover its shortening with no stiffness, and then engage with full stiffness when it reaches its original length.

1. Click on the top left brace (top left to bottom right) in both bays.
2. Use the Assign menu > Frame/Cable/Tendon > Tension/Compression Limits command to access the Frame T/C Limits form:
 - **Tension Limit.** Specify a tension limit by checking the *Tension Limit* box and entering a value that is zero or positive.
 - **Compression Limit.** Specify a compression limit by checking the *Compression Limit* box and entering a value that is zero or negative.

Note: If zero is specified for both the tension and compression limit, the element will carry no force in a nonlinear analysis.

Click on the Compression Limit option and enter a 0 kip limit. This makes these braces tension-only elements.

Tension/Compression limits are nonlinear. This means:

- They will be activated in a nonlinear static or nonlinear direct-integration time-history analysis only. In general, use nonlinear analysis when tension/compression limits are present in the model.
- If a linear analysis is performed using the stiffness at zero, the full axial stiffness (AE/L) of the element will be used as if there were no limits.
- If a linear analysis is performed using the stiffness at the end of a nonlinear analysis, the axial stiffness will be zero or AE/L, depending on whether the object exceeded its limits at the end of the nonlinear case.

This feature is useful for modeling cables and braces that can reasonably be represented by a single straight object when the analysis is focused more on the effect of the

cable/brace on the structure than on the detailed behavior of the cable/brace itself.

To model the deformation of the cable or brace in detail, break the cable/brace into several sub-objects, and use large-displacements analysis without compression limits. Under compression, the cable/brace will buckle out of the way in a more realistic representation of the true behavior.

RESULTS

We are now ready to run the analysis. Go to Analyze>Run Analysis and click Run Now. Click OK and the deflected shape for the DEAD load is shown. We are interested in viewing the axial load diagram for the frame elements. Under Display>Show Forces/Stresses>Frames, select the LATX load case and Axial Force component. The force diagram should look like Figure 6-4:

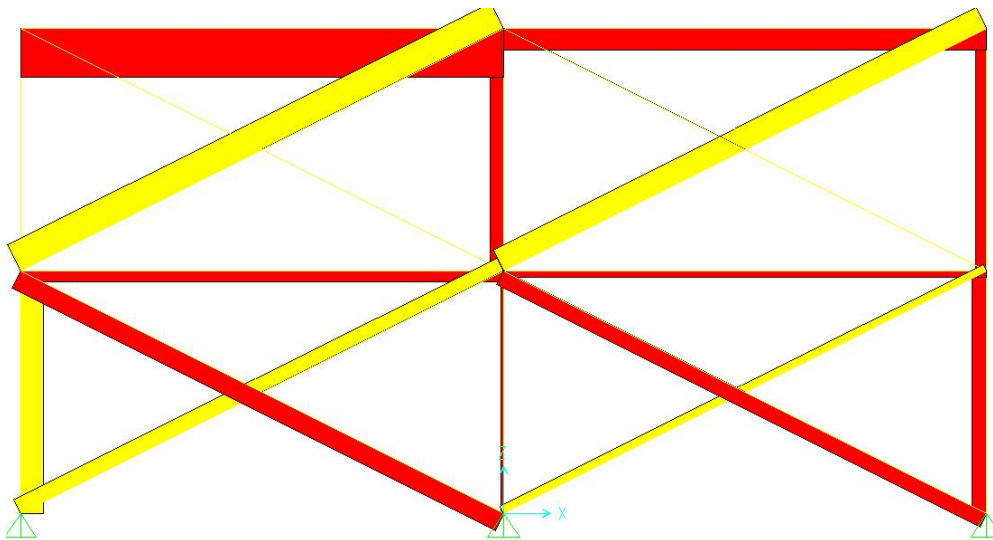


Figure 6-4 Axial Force Diagram

You can see the left brace forces in the top level are not taking any compression loads. The same braces in the bottom story are distributing compression loads.

EXAMPLE VII Nonlinear Time History Analysis

Description

This example consists of 3 simple 2-D frames, subjected to a nonlinear Time History Analysis with the use of link elements. The first frame has no link properties assigned to it. The second frame contains damper link properties, as its lateral force resisting system. The third frame contains isolator link elements. The frame geometry is shown in Figure 7-1.

Significant Options of SAP2000 Exemplified

- Time History Function Definition
- Time History Case Definition
- Damper/ Isolator Link Properties
- Creation of .AVI Files
- Time History Traces
- Response Spectrum Curves

Computer Model Definition

These 3 frames contain one bay that is spaced 24 ft. apart. Each frame is 3 stories high. Each story level is 12 ft. in height. Kip-inch-second units are used. The modulus of elasticity is 29000 ksi. A point mass of .25 is assigned to each joint in the X and Y-directions.

Link Element Definition

Damper and Isolator Link Element

Click the Define menu > Link Properties command to bring up the Define Link Properties form. Click the Add New Property button to access the NL link property data form. Select the Damper or Isolator link element from the pull-down menu Enter the desired values for Mass, Weight, and Rotational Inertia. Note that the mass is for the

entire length of the element and is not mass per unit volume. Typically, link elements can have two sets of properties assigned to them:

- **Linear properties** can be assigned to linear link elements only. The linear property that you specify for each of the six degrees of freedom of a linear link element is an effective stiffness. This is simply a spring stiffness.
- **Nonlinear dynamic properties**, which are used for nonlinear dynamic (time history) analysis. Note that you must have the nonlinear version of SAP2000 to use the nonlinear dynamic link properties.

The linear properties that you specify for each of the six degrees of freedom of all types of link elements other than linear link elements are an effective stiffness and effective damping. In such cases, the effective stiffness is, again, simply a spring stiffness. The effective damping specifies dashpot-type damping; it is not a specification of percent critical damping.

In a linear analysis SAP2000 converts the specified effective damping for a link element to modal damping. It then adds the modal damping calculated for all link elements in the model that have effective damping specified to any modal damping already specified for the structure as a whole to get the final modal damping. SAP2000 reports this final modal damping in the printed analysis output for building modes. To get this output, click the File menu > Print Tables > Analysis Output command and check the Building Modal Info check box.

Link Element Assignment

Damper Link Element

Go to Draw>Draw 2 joint link with the Damper property. Draw a diagonal line on every level of the middle frame.

Isolator Link Element

Go to Draw>Draw 1 joint link with Isolator property. Draw the link at the bottom story of the 3rd frame.

The model should look like Figure 7-1:

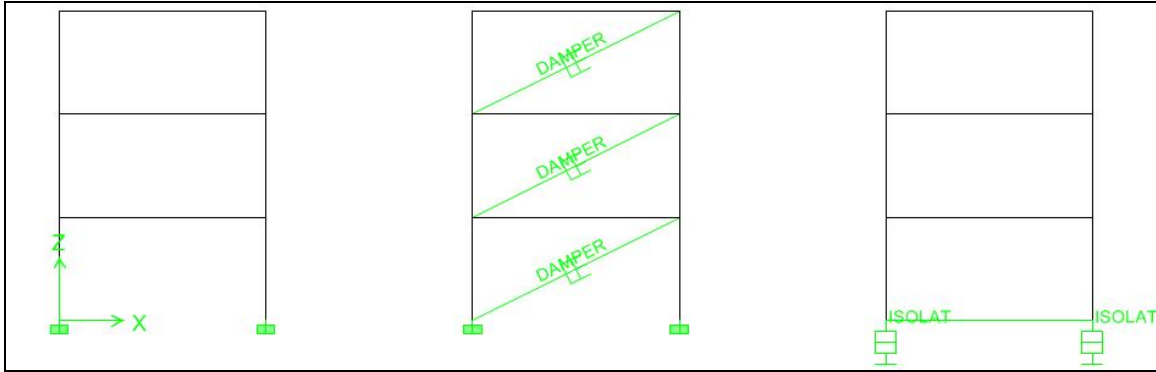


Figure 7-1 *Three-Frame Time History Model*

Time History Function and Case Definition

In this example, we are interested in defining a pre-defined time history function from a text file. Go to Define>Functions>Time History and select Function from File. Click the Browse button and select the time history text file. You can view time history graph by clicking on the Display Graph button. Fill in the Function File information:

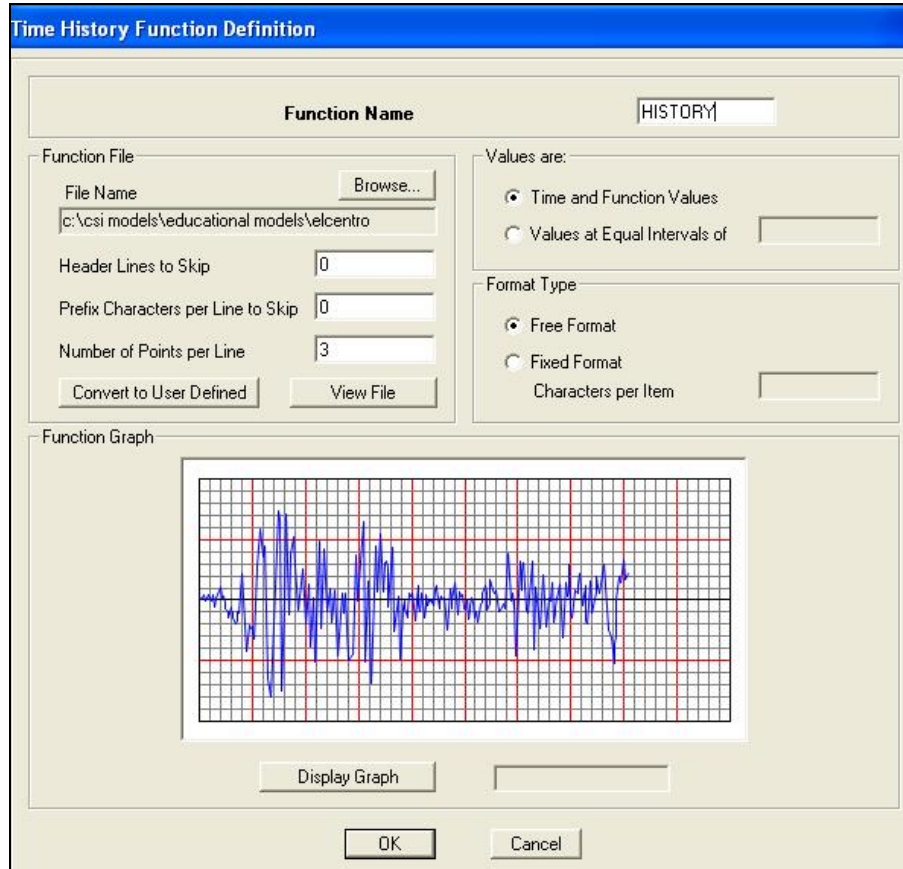


Figure 7-2 *Time History Function Definition*

Next, we define the Time History case data. Go to Analysis Cases>Add New Case>Time History Cases. There are many options available including Number of Output Time Steps and Output Time Step Size. The output time step size is the time in seconds between each of the equally spaced output time steps. Do not confuse this with the time step size in your input time history function. The number of output time step size can be different from the input time step size in your input time history function. The number of output time steps multiplied by the output time step size is equal to the length of time over which output results are reported.

We are interested in assigning the time history acceleration in the x-direction. The time history is a 12-second record, so enter 6000 time steps and time step size of .002 seconds.

Results

Creation of .AVI video files

After running the analysis (under menu>Analyze> Run Analysis), SAP2000 has the capability of creating an .AVI animation file. Use the File menu > Create Video> Time History Animation command to create videos in SAP2000 showing the movement of the structure during any time history analysis that has been run. Enter the input data for the video file:

The Start Time, End Time, Time Increment boxes must be filled to create the video file. Use the default or define start and end times and time increments. The time increment controls how many different pictures (frames) of the deformed shape of the structure are created. For example, a time increment of 0.1 means a picture (frame) of the deformed shape is created for every one-tenth second of the time history.

Response Spectrum Curves

After running a time history analysis, select one or more point objects and click the Display menu > Show Response Spectrum Curves command to access the Response Spectrum Generation form. Use the form to specify the appropriate data to plot the various response spectra.

Important Note: The response spectrum that you plot using this command is based on a time history that you have previously run. It has *nothing* to do with any response spectrum analysis that you may have run.

There are many options available when viewing Response Spectrum Curves. These include Axes, Damping and Period/Frequency definition. See Figure 7-3 for a sample Response Spectrum Curve.

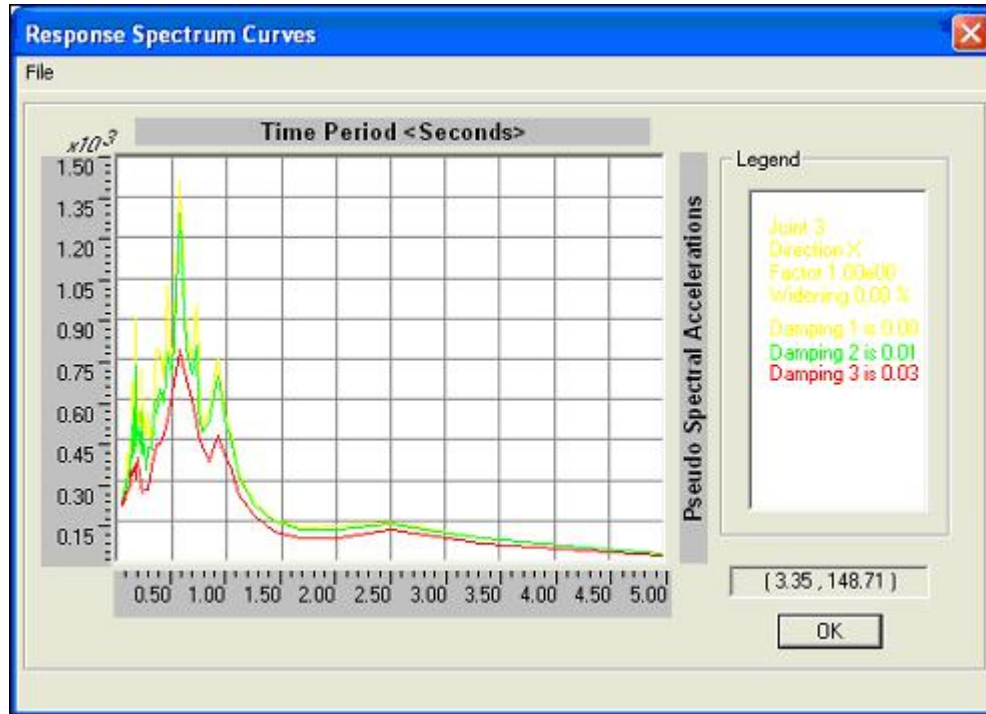


Figure 7-4 Response Spectrum Curves Plot

About the Speakers

ROBERT TOVANI

Robert Tovani has twenty-five years of experience in structural analysis, design, project management, and construction administration. He is currently president of Engineering Analysis Corporation and a Principal at Degenkolb Engineers.

Mr. Tovani has developed an extensive background in computer-aided analysis and design. His analysis background includes work on a variety of structures using linear and nonlinear analysis of new and existing structures in static and dynamic loading environments. He has developed computer models on high rise structures in excess of 100 stories and has provided design work on a variety of structural framing systems including base isolation and other complex framing systems. Mr. Tovani has been using the SAP and ETABS computer programs for over twenty-five years and has worked with CSI on example models and documentation.

ATIF HABIBULLAH

Atif Habibullah has worked as a Software Support Engineer at CSI for the past 4 years. He has extensive experience in modeling of a variety of structural systems, solving special modeling problems and interpretation of analysis results. Prior to working at CSI, Atif has worked in a design office for 6 years using CSI products, particularly in the design of multi-story steel and concrete building structures such as hospitals, office buildings, towers, bridges, stadiums and dams.

MESH TRANSITIONING AND COMPATIBILITY THE AUTOMATED LINE CONSTRAINT IN ETABS & SAP2000

Ashraf Habibullah, S.E., President & CEO, Computers & Structures, Inc.

M. Iqbal Suharwardy, S.E., Ph.D., Director of Research & Development, Computers & Structures, Inc.

In the application of the Finite Element Analysis Method, the most time consuming task is usually the creation and modification of the finite element mesh of the system. Not to mention the fact that creation of mesh transitions from coarse to fine meshes can be very tedious. Also matching up node points to create compatible meshes at intersecting planes, such as walls and floors can be very labor intensive. And even if the mesh generation is automated the mesh transitioning usually produces irregular or skewed elements that may perform poorly. This may have adverse effects on the design, especially in regions of stress concentration, such as in the vicinity of intersecting planes.

The object based modeling environment of ETABS & SAP2000 clearly addresses these time-consuming shortcomings of the Finite Element Method.

In the object-based modeling environment the Engineer generates the structural model by creating only a few large area objects that physically define the structural units such as wall panels, floors or ramps. The finite element mesh is not explicitly created by the user, but is automatically generated by assigning meshing parameters to the area objects. These parameters may include variables, such as mesh size, mesh spacing and mesh grading among others. With this capability the engineer can study the effects of mesh refinement by just defining a few control parameters. The new model with the desired level of refinement is thus created with minimal effort.

If the meshes on common edges of adjacent area objects do not match up, automated line constraints are generated along those edges. These Line Constraints enforce displacement compatibility between the mismatched meshes of adjacent objects and eliminate the need for mesh transition elements.

What makes this technology really powerful is that while making modifications to the model the Engineer need only be concerned about the few large physical objects of the structure. The modified finite element analytical model gets recreated automatically with any changes to the base objects.

The following examples are designed to illustrate the power and practicality of this technology.

EXAMPLE 1

Simply Supported Plate (Mismatched Meshing)

As illustrated in Figure 1, this is a model of a simply supported plate, which has been modeled in two different ways. In one case the mesh is uniform across the plate and in the other case the mesh is fine on one half of the plate and coarse on the other half of the plate. In the latter case, an interpolating line constraint is automatically generated to enforce displacement compatibility between the adjacent halves of the plate where the mesh does not match. As shown in the figure, correlation between the two models is very good.

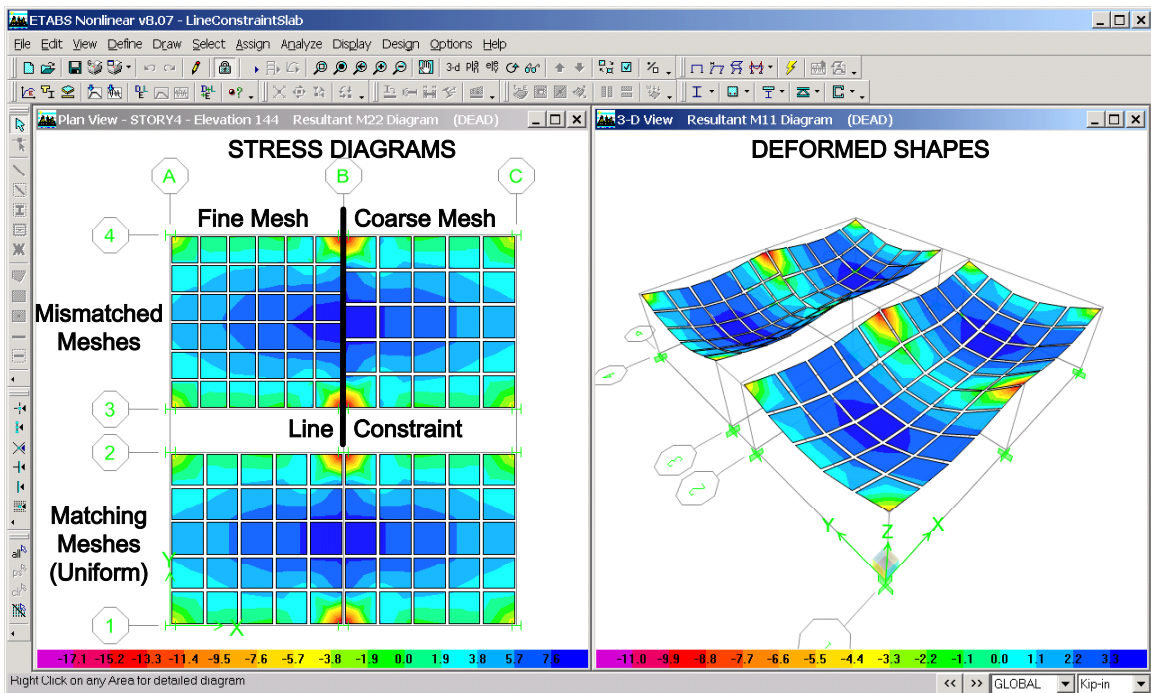


Figure 1: Simply Supported Plate with Mismatching Edges

EXAMPLE 2

Curved Ramp Supported by Curved Wall

This example, Figure 2, illustrates the use of Line Constraints to capture the interaction of a curved shear wall supporting a curved ramp. Notice that there are no joints at the points where the ramp element edges intersect the wall element edges. Displacement compatibility along the lines of intersection of the ramp and the wall is enforced automatically by the generation of Line Constraints along those lines. Notice how the application of Line Constraints allows the wall and ramp mesh to retain a simple rectangular (or quadrilateral) configuration. A conventional finite element model would be very irregular because it would need all the additional joints (and corresponding elements) to allow for the ramp element and wall element edge intersections.

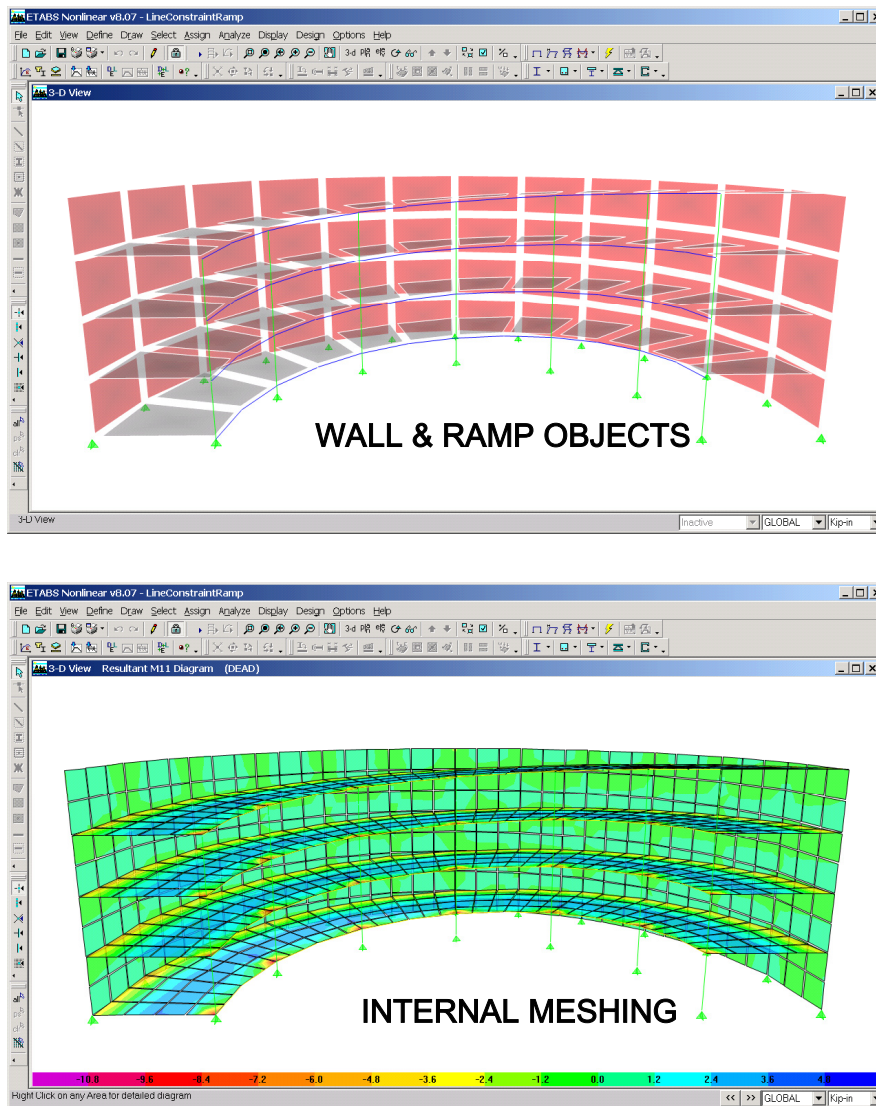


Figure 2: Curved Ramp Supported by Curved Wall

EXAMPLE 3

Floor Slab - Shear Wall Compatibility

This example, Figure 3, illustrates a 3D Concrete Flat Plate Building with shear walls and an elevator core. Again, in this model, Line Constraints automatically appear at the lines where the floor and wall objects intersect. This, of course, as in previous examples, will enforce displacement compatibility when mesh geometries do not match. As shown in the deformed shape of the Elevator Core, in many places the wall meshing does not match the floor meshing. All elements meeting at common edges, however, still show no displacement incompatibilities, even though the element nodes do not coincide.

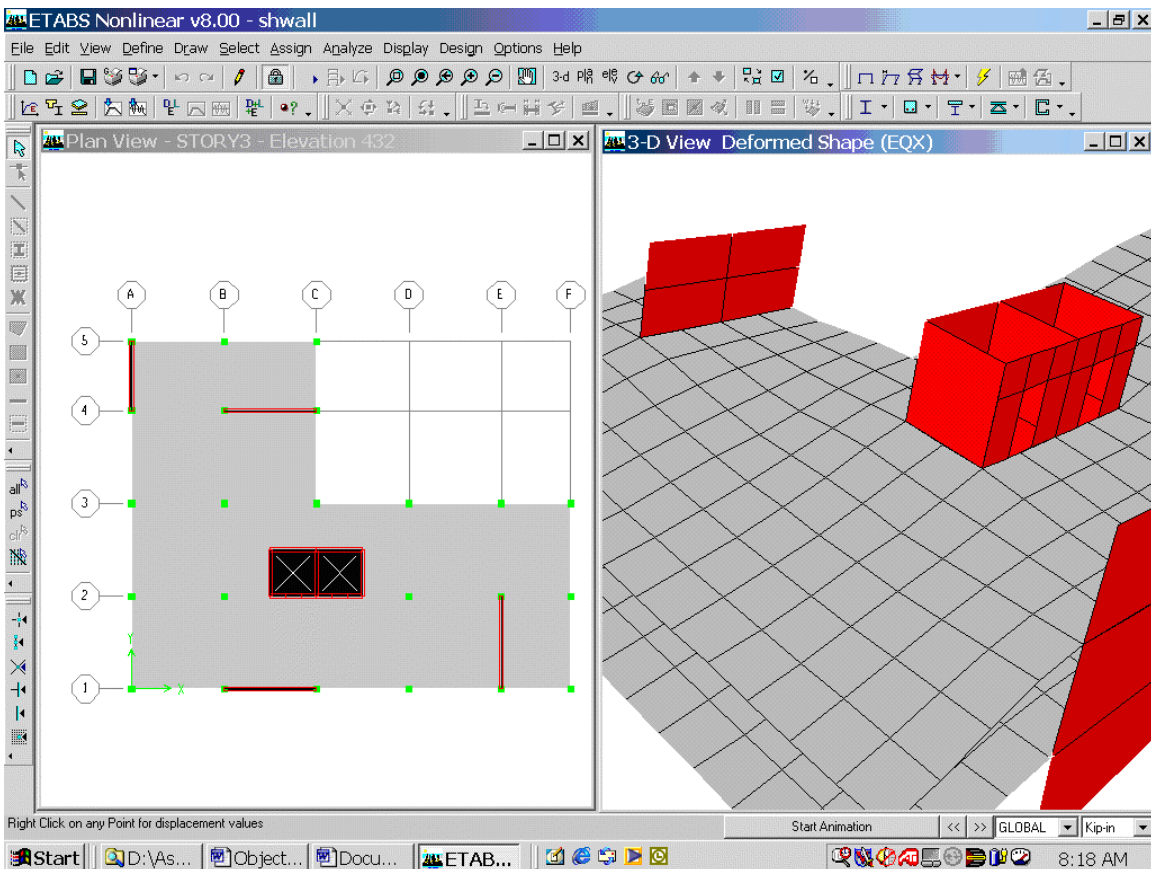


Figure 3: Floor Slab - Shear Wall Compatibility

EXAMPLE 4

Shear Wall – Spandrel Transition

This example, Figure 4, models a Shear wall – Spandrel System, illustrating mesh transitioning from the spandrel to the shear wall. Line Constraints are generated as needed in any direction. In this case the Line Constraints are vertical as well as horizontal.

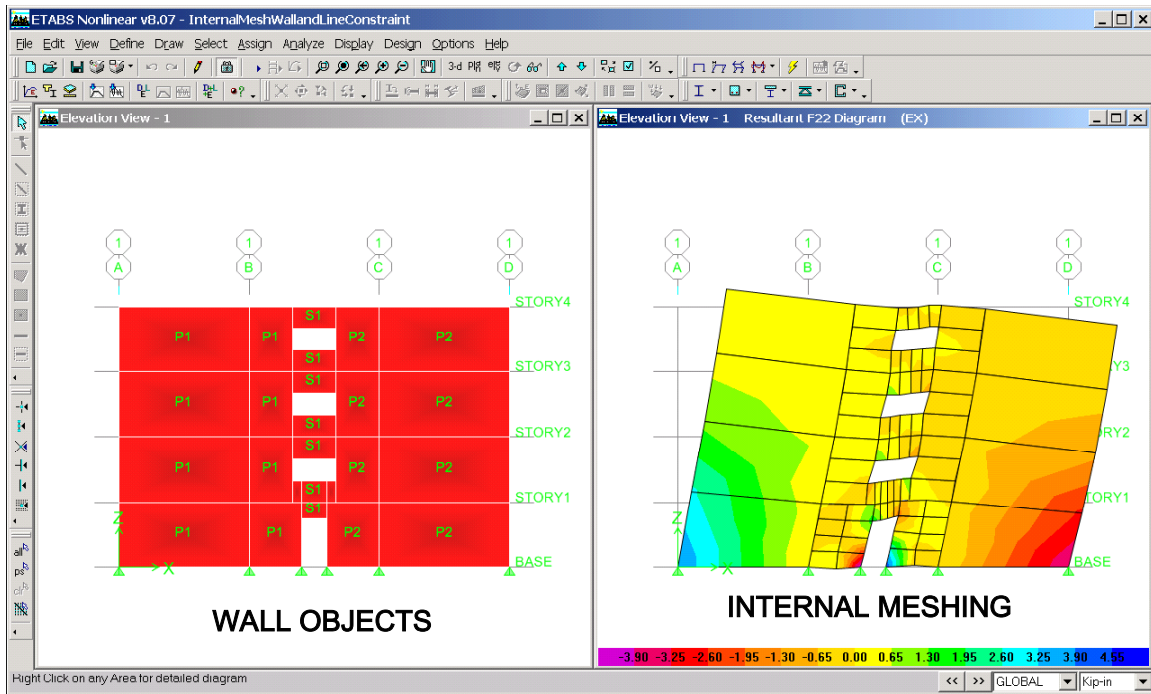


Figure 4: Shear Wall - Spandrel Transition

BIBLIOGRAPHY

1. *Computers & Structures, Inc., "SAP2000 – Integrated Software for Structural Analysis & Design, Technical Reference Manual"*
2. Computers & Structures, Inc Website: www.computersandstructures.com