

SAP2000®

**Linear and Nonlinear
Static and Dynamic
Analysis and Design
of
Three-Dimensional Structures**

INTRODUCTORY TUTORIAL



**Computers and Structures, Inc.
Berkeley, California, USA**

Version 8.0
June 2002

Copyright

The computer program SAP2000 and all associated documentation are proprietary and copyrighted products. Worldwide rights of ownership rest with Computers and Structures, Inc. Unlicensed use of the program or reproduction of the documentation in any form, without prior written authorization from Computers and Structures, Inc., is explicitly prohibited.

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

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

Phone: (510) 845-2177

FAX: (510) 845-4096

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

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

web: www.csiberkeley.com

© Copyright Computers and Structures, Inc., 1978-2002.
The CSI Logo is a registered trademark of Computers and Structures, Inc.
SAP2000 is a registered trademark of Computers and Structures, Inc.
Windows is a registered trademark of Microsoft Corporation.
Adobe and Acrobat are registered trademarks of Adobe Systems Incorporated.

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAP2000. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

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

— |

| —

— |

| —

Contents

Introductory Tutorial for SAP2000

1	Introduction	
	Using This Manual	1-1
	Overview of the Program	1-2
	Using this Tutorial	1-2
2	An Introductory Tutorial	
	The Project	2-2
	The Interface	2-2
Step 1	Begin a New Model	2-3
	Define an Auto Select Section List	2-6
Step 2	Add Frame Objects	2-10
	Draw Frame Objects	2-10
	Replicating Objects	2-11
	Trimming Objects	2-14
	Assigning Member End Releases	2-17
	Save the Model	2-19
Step 3	Add Area Objects	2-19
	Define the Area Sections	2-19
	Draw the Area Object	2-20
	Mesh the Area Object	2-22
Step 4	Add Restraints	2-23

Introductory Tutorial for SAP2000 Version 8

Step 5	Define Load Cases	2-25
Step 6	Assign Gravity Loads	2-26
Step 7	Assign Area Stiffness Modifiers	2-28
Step 8	Run the Analysis	2-29
Step 9	Graphically Review the Analysis Results	2-30
Step 10	Design the Steel Frame Objects	2-34

Chapter 1

Introduction

Using this Manual

This manual introduces you to SAP2000 Version 8. The step-by-step instructions guide you through development of your first model. The intent is to demonstrate the fundamentals and to show how quickly and easily a model can be created using this program. This tutorial is intended to give you hands-on experience working with SAP2000, which for most people, is the quickest way to become familiar with the program.

SAP2000 is an extremely versatile and powerful program with many features and functions. This manual does not attempt to fully document all of those capabilities. Rather, we briefly show how to work with the program, providing some commentary along the way. To grasp the full value of SAP2000, you should use this introductory tutorial manual in conjunction with the other SAP2000 documentation.

We hope that you enjoy using this tutorial, and that you find it beneficial as a starting point in your exploration of this powerful and comprehensive version of SAP2000.

1

Overview of the Program

SAP2000 is a stand-alone finite-element-based structural program for the analysis and design of civil structures. It offers an intuitive, yet powerful user interface with many tools to aid in the quick and accurate construction of models, along with the sophisticated analytical techniques needed to do the most complex projects.

SAP2000 is object based, meaning that the models are created with members that represent the physical reality. A beam with multiple members framing into it is created as a single object, just as it exists in the real world, and the subdividing needed to ensure that connectivity exists with the other members is handled internally by the program. Results for analysis and design are reported for the overall object, and not for each sub-element that makes up the object, providing information that is both easier to interpret and more consistent with the physical structure.

Using this Tutorial

The example in this tutorial provides a step-by-step description of how to use SAP2000. We recommend that you actually perform these steps in SAP2000 while reading this manual.

The SAP2000 program must be installed on your computer before you can begin the tutorial. It would also be a good idea to peruse the other SAP2000 documentation prior to starting this tutorial, or at least have them readily available if needed.

If you are viewing this tutorial manual as a .pdf file, we strongly recommend that you print it out before starting the tutorial. It will not be practical to use the SAP2000 program while trying to read this manual on your computer screen.

During the course of this tutorial, we will explore many of the basic features of SAP2000. Prepare to spend at least one hour going through this example, and if at any time you need to stop, save your model so that you may continue at a later time.

Welcome to SAP2000.

Chapter 2

An Introductory Tutorial

This chapter provides step-by-step instructions for building a basic SAP2000 model. Each step of the model creation process is identified, and various model construction techniques are introduced. At the completion of this chapter, you will have built the model shown in Figure 1.

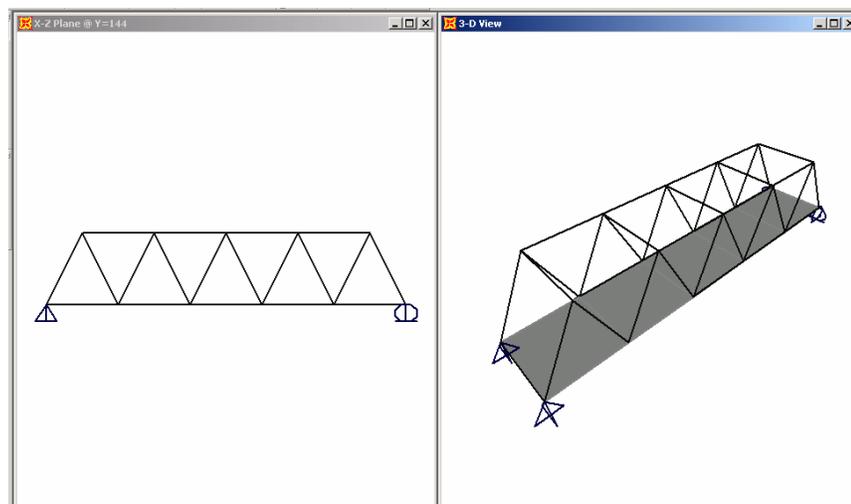


Figure 1
*The Tutorial
Model*

The Project

2

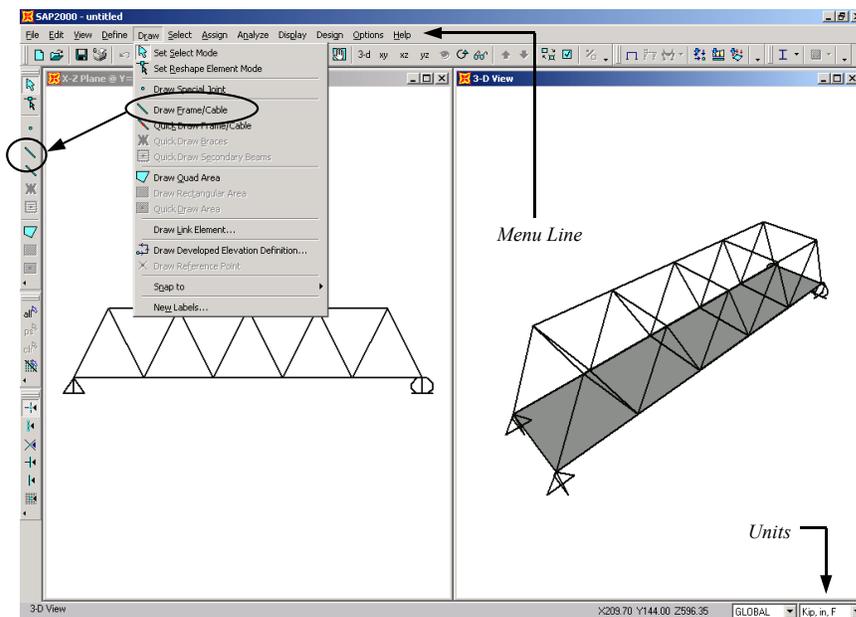
The tutorial project is a five panel, sloped truss bridge. The bridge spans 60 feet, and has a width and height of 12 feet each. The supports are rollers at one end, and pins at the other.

The trusses and cross members are to be constructed of 2L4X4's, while the deck will be a concrete slab 5 inches thick. The bridge will be analyzed for static loads only, and the deck will be loaded with a Dead Load = 10 pounds per square foot (psf) and a Live Load = 100 psf.

The Interface

The top menu line contains all of the commands and options available to SAP2000, including Define, Draw, Select, Assign, Analyze, Display and Design. These listed menus contain the commands that will be needed most often when using SAP2000, and many of the most frequently used commands are accessible as a single click button in the screen regions surrounding the drawing areas. The availability of a button is indicated in the main menus by the existence of an icon to the left of the command. The lower right corner shows the current unit selection. Figure 2 shows the layout of the interface.

Figure 2
The Interface

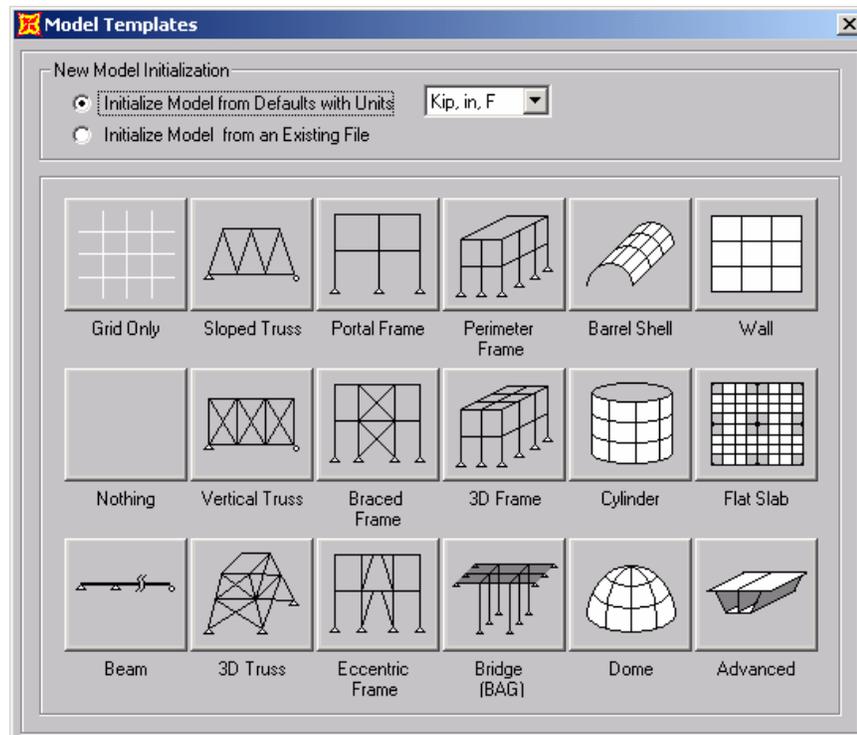


Step 1 Begin a New Model

In this Step, the basic grid will be defined which will serve as a template for developing the model. Then a list of double angle sections will be selected for the truss Auto Select list.

- A. Click the **File menu** > **New Model** command or the New Model button . The form shown in Figure 3 will display. Verify that the default units are set to Kip-in.

*Figure 3
Model
Templates*



- B. The Model Template form allows for the quick generation of numerous model types using parametric generation techniques. However, in this tutorial the model will be started using only the grid generation. When laying out the grid, it is important that the geometry defined accurately represents the major geometrical aspects of the

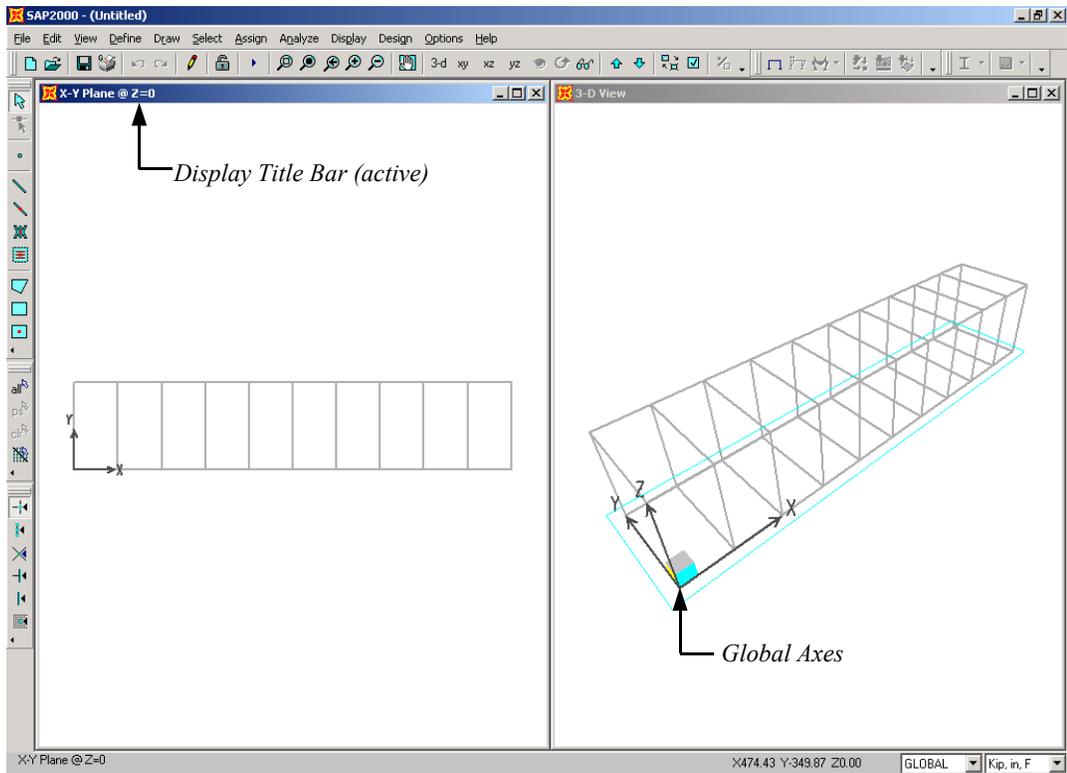
2

model, so it is advisable to spend time carefully planning the number and spacing of the grid lines. Select the **Grid Only** button, and the form shown in Figure 4 will display.

Figure 4
New Coord/Grid
System form

The screenshot shows a dialog box titled "New Coord/Grid System". It has two tabs: "Cartesian" (which is selected) and "Cylindrical". The "System Name" field contains the text "GLOBAL". There are two main sections: "Number of Grid Spaces" and "Grid Spacing". In the "Number of Grid Spaces" section, the "X direction" field has the value "10", the "Y direction" field has "1", and the "Z direction" field has "1". In the "Grid Spacing" section, the "X direction" field has "72", the "Y direction" field has "144", and the "Z direction" field has "144". At the bottom of the dialog are "OK" and "Cancel" buttons.

- C. The New Coord/Grid System form is used to specify the grids and spacing in the X, Y and Z direction. Set the number of grid spaces to 10 for the X direction, and to 1 for the Y and Z directions. Type **6 ft** into the X direction spacing edit box and press the Enter key on your keyboard. Note that the program automatically converts the 6 ft to 72 to be consistent with the default units of inches. Enter **12 ft** or 144 for both the Y and Z direction spacing.



2

Figure 5
The SAP2000
windows

D. Click the **OK** button to accept the changes, and Figure 5 will appear.

The grids appear in two view windows tiled vertically, a X-Y “Plan” View on the left and a 3-D View on the right, as shown above. The number of view windows may be changed by selecting the **Options menu > Windows** command.

Notice that the “Plan” view is active in Figure 5. When the window is active, the display title bar is highlighted. Set a view active by clicking anywhere in the view window.

Note that the Global Axes are displayed as well, and that Z positive is in the “up” direction. When SAP2000 refers to the direction of gravity, this is in the negative Z direction, or “down”.

Define an Auto Select Section List

An auto select section list is simply a list of sections, which for this tutorial will be a set of double angles (2L4X4's). Auto select section lists are assigned to frame objects in the same manner as an individual section property. When an auto select section list is assigned to a frame object, the program can automatically select the most economical, adequate section from the list when designing the member. When performing the initial analysis, the program will assign the median section from the list for the analysis properties.

For this particular tutorial, the program will analyze and design from a set of double angles, which will be chosen from an auto select sections list created now.

- A. Click the **Define menu > Frame/Cable Sections** command, which will display the Frame Properties form shown in Figure 6.

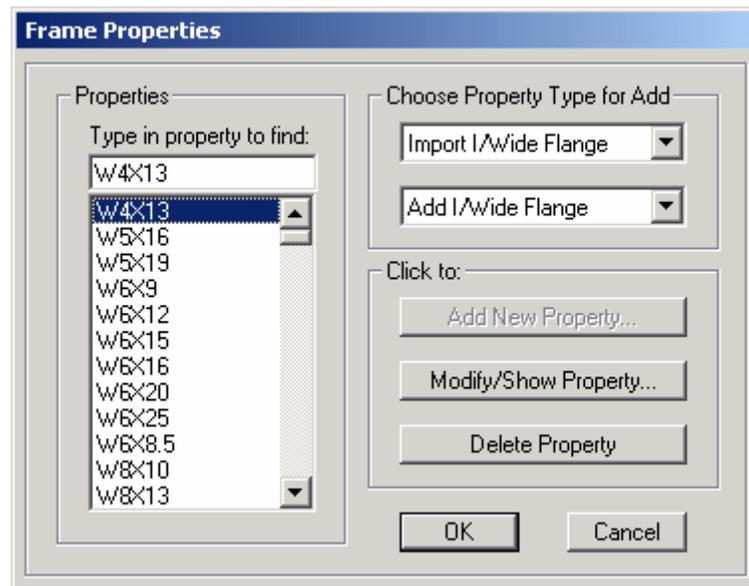
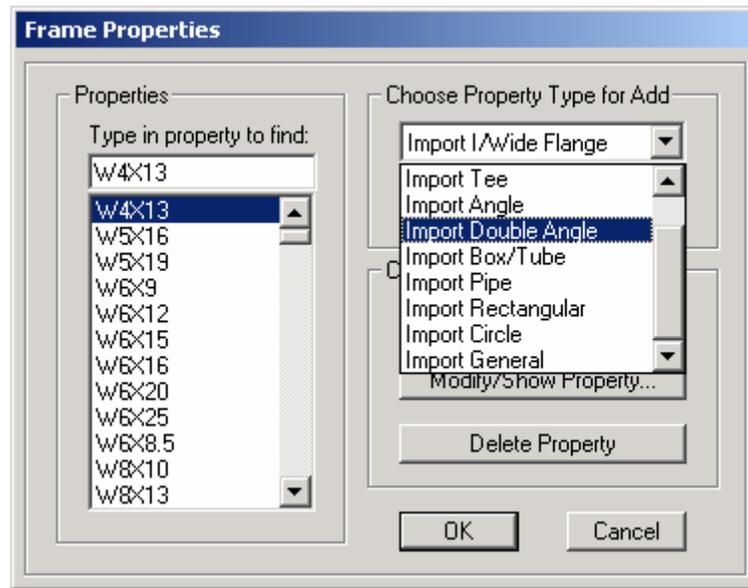


Figure 6
The Frame Properties form

- B. Scroll down the sections listed under Properties to see if the list contains 2L4X4's, and if it does, skip ahead to Step D. Otherwise, proceed to Step C.

- C. Click the drop-down box that reads “*Import I/Wide Flange*” in the Choose Property Type for Add area of the form. Scroll down the list of import options until you find *Import Double Angle* – see Figure 7. Single click on it.

Figure 7
Import Double Angle

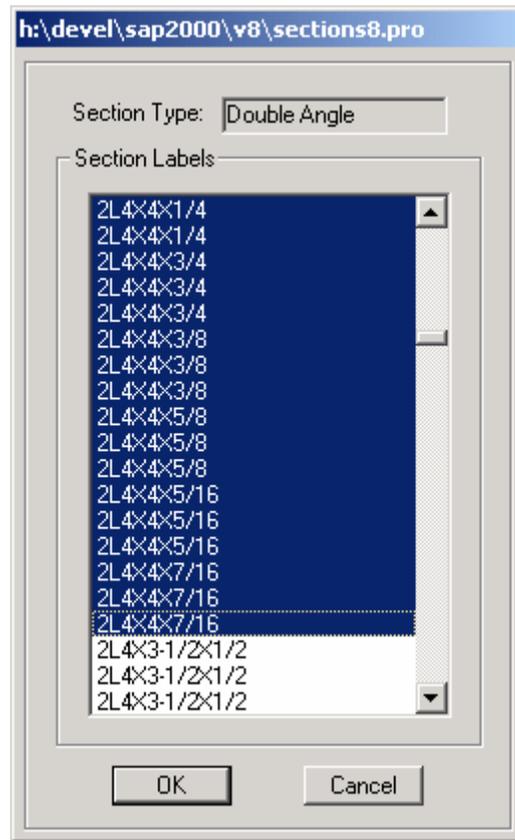


- D. In the Click to area of the Frame Properties form, click the **Add New Property** button, which will open the Section Property File form.
- E. Select and open the file named *SECTIONS8.PRO* from the Section Property File form, as this file contains the properties of the double angles to be used in the model. The Sections8.pro sections list form shown in Figure 8 appears.
- F. Scroll down the list of double angles in the Sections Labels area until you find the first 2L4X4. Click once on that member to highlight it.
- G. Scroll further down the list until you find the last 2L4X4. Hold down the Shift key on your keyboard and click once on the last 2L4X4X7/16 – all of the 2L4X4’s should now be highlighted.

- H. Click the **OK** button, and then click the **OK** button in the Double Angle Section form to add the angles selected to the list in the Properties area on the Frame Properties form.

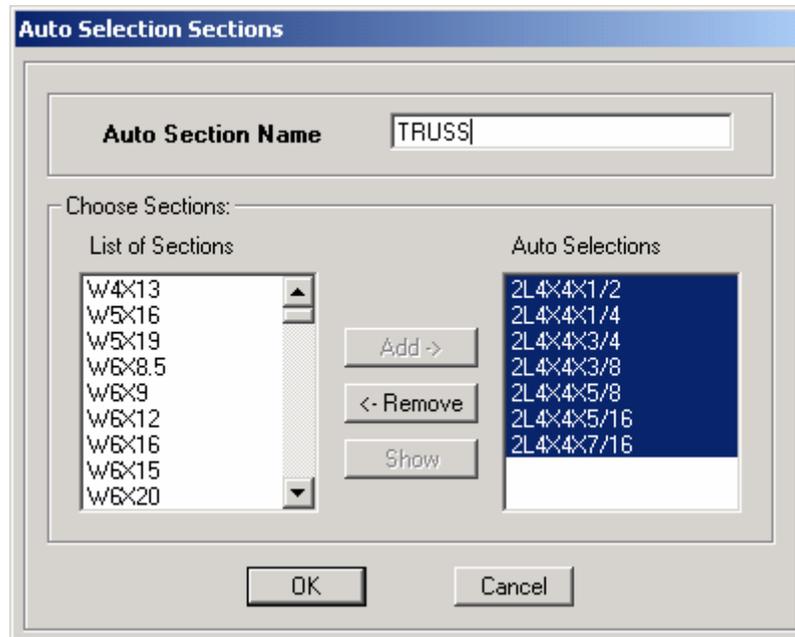
2

Figure 8
Sections8.pro
sections
list



- I. Click the drop-down box that reads “Add I/Wide Flange” in the Choose Property Type for Add area of the Frame Properties form and scroll down until you locate *Add Auto Select*. Single click on it.
- J. In the Click To area of the Frame Properties form, click the **Add New Property** button, which will open the Auto Selection Sections form shown in Figure 9.
- K. Type **TRUSS** in the Auto Section Name edit box.

Figure 9
Auto
Selection
Sections
form



- L. Scroll down the List of Sections to find the *2L4X4X1/2* double angle, and click once to highlight it.
- M. Continue down the list until you find the last double angle, *2L4X4X7/16*, and while holding down the shift key on the keyboard, click once on this section. All of the 2L4X4's should now be highlighted.
- N. Click the **Add** button to move the selected list to the Auto Selections edit box on the right side of the form.
- O. Click the **OK** button and then click the **OK** button on the Frame Properties form to accept your changes and add the TRUSS auto select list to the Properties edit box.

Step 2 Add Frame Objects

In this Step, Frame objects with the associated TRUSS sections list are drawn using the grids and snap-to options, and generated with Edit menu commands.

Draw Frame Objects

Make sure that the X-Y Plane @ Z=0 view (plan at lowest elevation) is active (see Step1-D for directions on how to make a view active and page 2-14 for setting the view). This view should be in the left window. Also check that the **Snap to Points and Grid Intersections** command is active. This will assist in accurately positioning the frame objects. This command is active when its associated button  is depressed. Alternatively, use the **Draw menu > Snap to > Points and Grid Intersections** command. By default, this command is active.

- A. Click the **Draw Frame/Cable**  button or use the **Draw menu > Draw Frame/Cable** command. If you accessed the Draw Frame/Cable command via the Draw menu, the Draw Frame/Cable button will depress verifying your command selection. The Properties of Object pop-up box for frames will appear as shown in Figure 10.

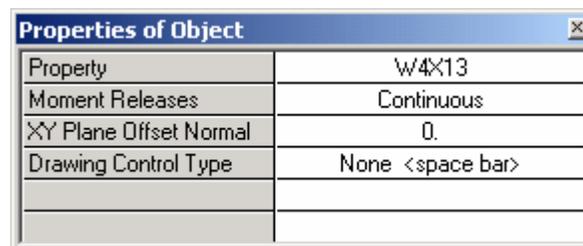


Figure 10
Properties of
Object box

If the Properties of Object box is covering any part of the model in either view, drag it out of the way.

- B. Click in the Property edit box on the Properties of Object form and scroll down to *TRUSS*. Single click on it to assign the auto select list TRUSS to the members you will draw.

- C. To draw the first frame object, left click once in the X-Y Plane view at the X-Y origin, and then click again at the far right end along the same horizontal grid line ($x=720$, $y=0$). The cursor location is indicated in the lower right-hand corner of the interface. A frame line should appear in both views (plan and 3D). After clicking to define the end point of the frame object, a right click will “lift the pen” so you will no longer be actively drawing, but will leave the Draw Frame/Cable command active so that you may add additional elements.

If you have made a mistake while drawing this object, click the **Select Object**  button, to leave the Draw mode and go to the Select mode. Then click the **Edit menu > Undo Frame Add** command, and repeat Items A-C.

- D. Repeat Item C, drawing an additional frame object parallel to the first member from ($x=0$, $y=144$) to ($x=720$, $y=144$). These members form the bottom chords of the trusses. Right click to stop drawing.
- E. Left click at ($x=0$, $y=0$) and then at ($x=0$, $y=144$) to draw the first transverse member.
- F. Click on the **Select Object**  button, or Press the Esc key on the keyboard to exit the Draw Frame/Cable command.

Replicating Objects

Make sure that the program is in the Select mode.

- A. Select the transverse member spanning between the longitudinal chords by a left click directly on the member, or left click to the right of the object, and while holding the left mouse button down, drag the mouse across the member. See Figure 12 for selection options.
- B. Click the **Edit menu > Replicate** command to bring up the form shown in Figure 11.
- C. On the Linear tab, type **144** into the dx edit box.
- D. Type **5** into the Number edit box.

- E. Click the **OK** button. Note that transverse members have been generated at every other grid line.

2

Figure 11
Replicate
box

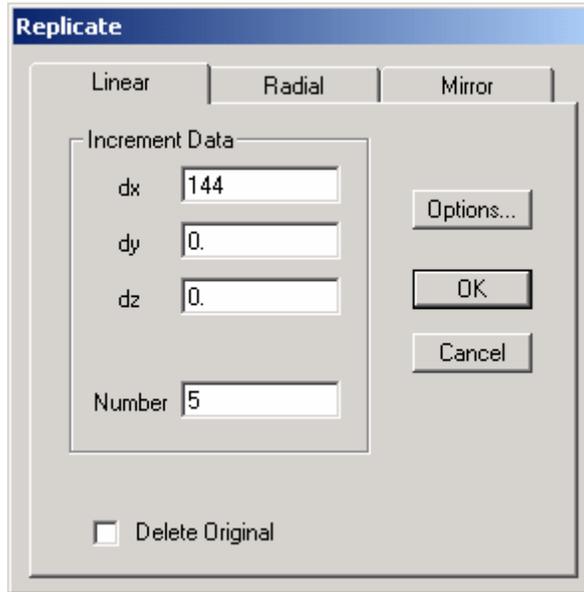
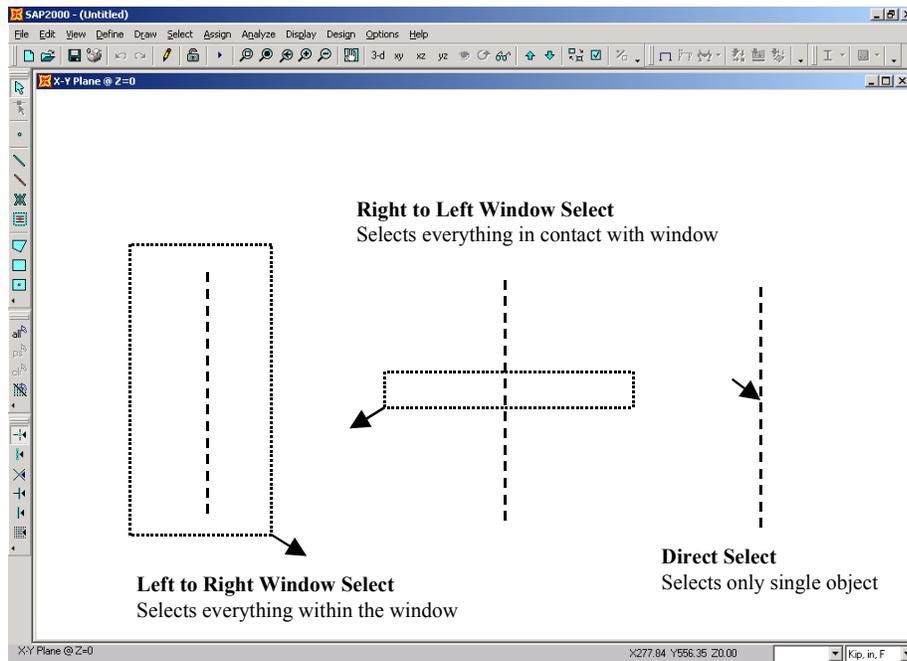


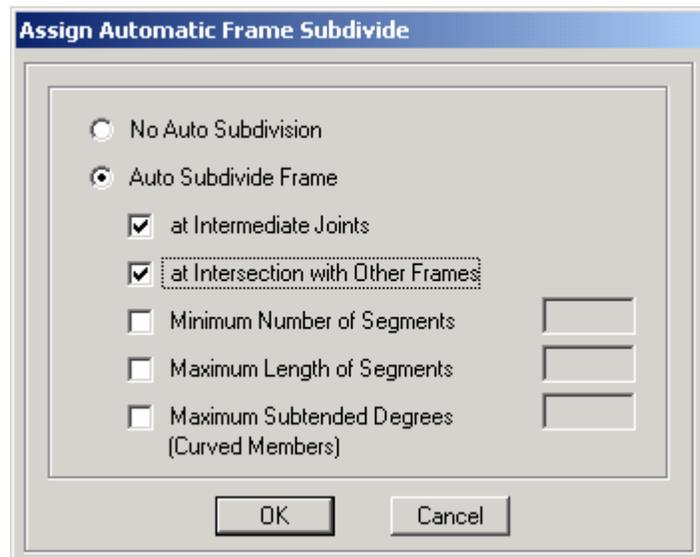
Figure 12
Graphical
Selection
Options



- F. Left click once on each of the longitudinal chord members to select them.
- G. Click on the **Assign menu > Frame/Cable > Automatic Frame Subdivide** command to bring up the form in Figure 13. Select the *Auto Subdivide Frame* option and check the *at Intermediate Joints* and *at Intersection with Other Frames* check boxes, and click **OK**.

This subdivision is necessary to ensure connectivity between the chords and the other members because the chords were drawn as single “physical” objects. From an analytical standpoint, the chords will now be connected to all of the elements framing into them, but for design and selection they will remain as single objects.

Figure 13
Assign
Automatic
Frame
Subdivide
form



- H. Click the **Select All**  button or use the **Select menu > Select > All** command to select all of the objects currently in the model.
- I. Click the **Edit menu > Replicate** command to bring up the Replicate form.
1. Type **72** into the dx edit box, **0** into the dy box, and **144** into the dz box.

2. Type **1** into the Number edit box.
3. Click **OK** to accept the changes.

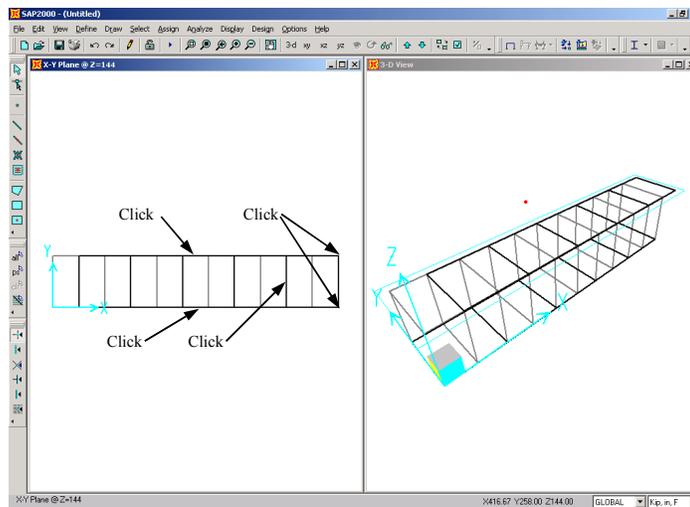
The framing at the bottom plan will be replicated at the top level with a shift of 72 inches in the X direction.

Trimming Objects

Make sure that the program is in the select mode, and that the X-Y view is active.

- A. Click the **View menu > Set 2D View** command
 1. In the Set 2D View form click on the *X-Y plane* option.
 2. Type **144** into the Z= edit box to display the plan view at the upper elevation.
- B. Click the **Assign menu > Clear Display of Assigns** command to remove the Frame Subdivide identifiers.
- C. Click on both top chords, the next to last transverse member to the right, and the two point objects at the far right ends of both chords, as shown in Figure 14. The selected objects should be shown as dashed lines.

Figure 14
Select mode for Trim



D. Click the **Edit menu > Trim/Extend Frames** command, which brings up the Trim/Extend Selected Frames form.

1. Select the *Trim Frames* option, and click **OK**.

2

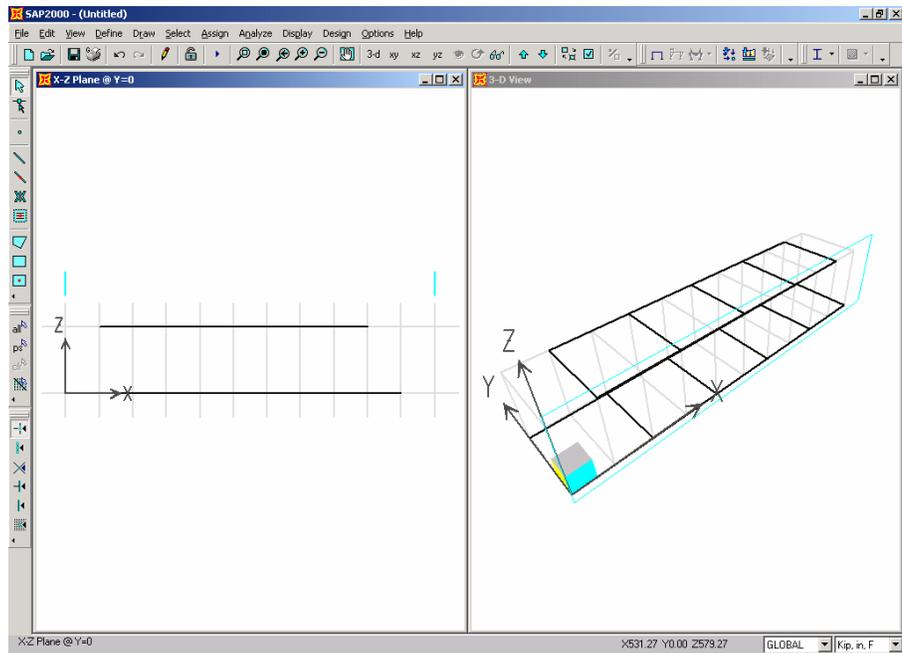
Selecting the Trim Frames option will trim the two top chords beyond the next to last transverse member. To trim a Frame member, select the member, select a member to be used as the trim location, and select a point object on the side to be trimmed.

E. Click on the “orphaned” transverse frame member on the far right, and go to the **Edit menu > Delete** command, or Press the Delete key on your keyboard.

F. Make sure that the plan view is active and click the **XZ View** button.

Your model now appears as shown in Figure 15.

*Figure 15
Model after
frame ob-
jects have
been added
in plan*



- G. Click the **Draw Frame/Cable**  button or use the **Draw menu > Draw Frame/Cable** command. The Properties of Object pop-up box for frames will appear.
- H. Make sure that the Property item on the Properties of Object form is set to *TRUSS*.
- I. To draw the first diagonal, left click once in the X-Z Plane view at the X-Z origin, and then click again at the nearest end of the top chord ($x=72, z=144$). Without clicking on the right mouse button, add a second diagonal by doing a left click at point ($x=144, z=0$).

Diagonals for one bay are now drawn.

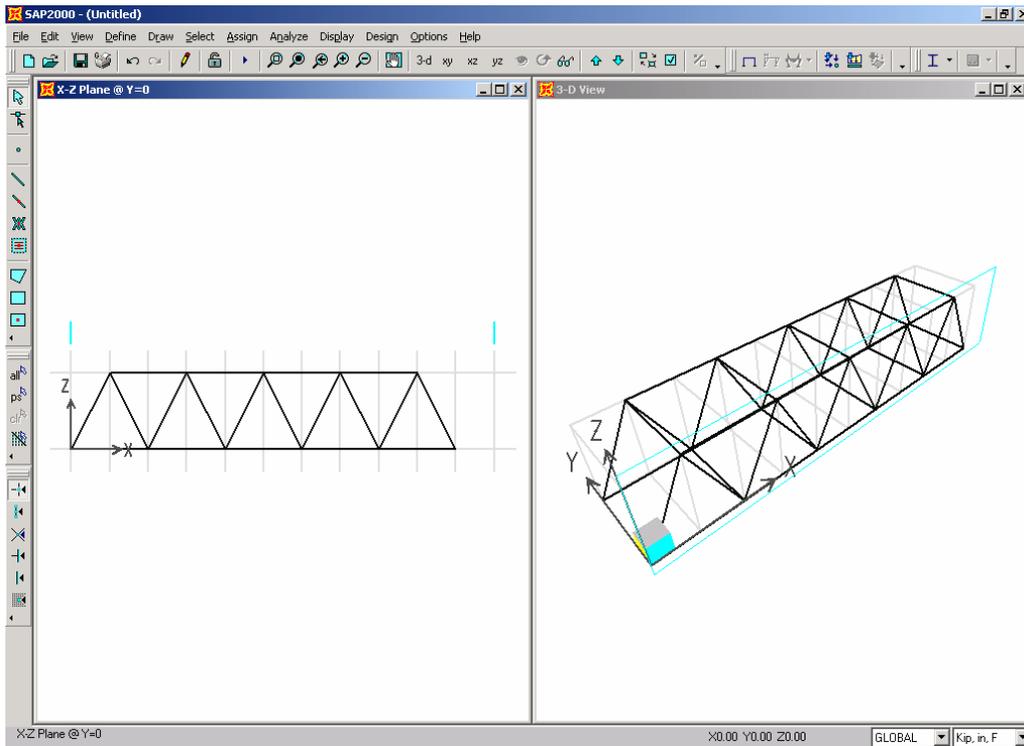
- J. Right click and then click on the **Select Object**  button, or Press the Esc key on the keyboard to exit the Draw Frame/Cable command.
- K. Draw a Selection Box from Right to Left across the two diagonals just drawn to select both diagonals. See figure 12 for selection options.
- L. Click the **Edit > Replicate** command to bring up the Replicate form.
 - 1. Type **144** into the dx edit box, **0** into the dy box, and **0** into the dz box.
 - 2. Type **4** into the Number edit box.
 - 3. Click **OK** to accept the changes.

All of the diagonals for one truss have been drawn.

- M. Draw a Selection Box from Right to Left across all of the diagonals.
- N. Click the **Edit > Replicate** command to bring up the Replicate form.
 - 1. On the Linear tab, type **0** into the dx edit box, **144** into the dy box, and **0** into the dz box.
 - 2. Type **1** into the Number edit box.

3. Click **OK** to accept the changes.

The model now appears as shown in Figure 16.



2

Figure 16
Model after all frame
objects have been added

Assigning Member End Releases

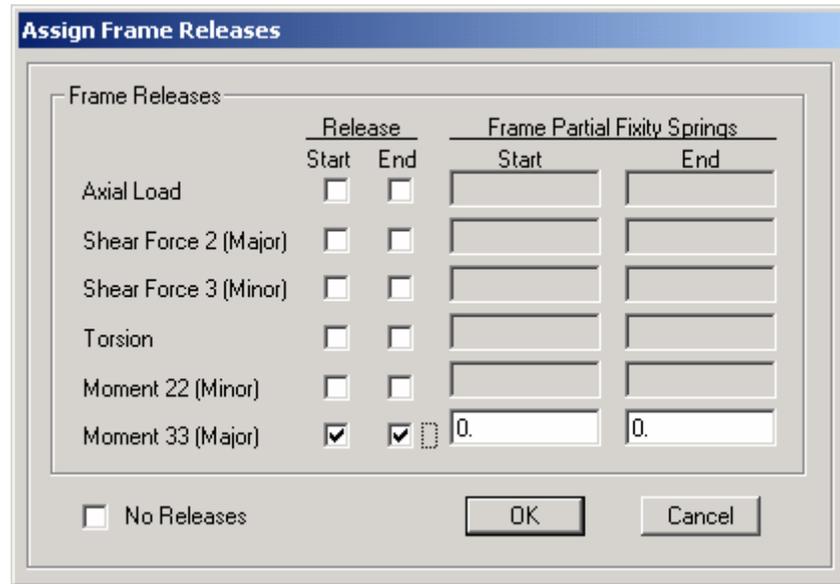
Make sure that the program is in the select mode, and that the X-Z view is active.

- A. Draw a Selection Box from Right to Left across all of the diagonals.
- B. Click on the **Assign** menu > **Frame/Cable** > **Releases/Partial Fixity** command to bring up the form shown in Figure 17. Check the *Moment 33 (Major)* check boxes for both the *Start* and *End* Releases.

By releasing the moments in the major direction, the diagonals in the trusses will behave as pinned elements.

2

Figure 17
Assign Frame Releases form



- C. Click **OK** to accept the changes and return to the select mode.
- D. Click the **View menu > Set 2D View** command. In the Set 2D View form click on the *X-Z plane* option and type **144** into the Y= edit box to display the second elevation view. Alternatively, use the **Move Up in List**  button to toggle to the other elevation.
- E. Draw a Selection Box from Right to Left across all of the diagonals.
- F. Click on the **Assign menu > Frame/Cable > Releases/Partial Fixity** command to bring up the Assign Frame Releases form and make sure that the *Moment 33 (Major)* check boxes for both the *Start* and *End* Releases are checked. Click **OK** to accept changes.
- G. Click the **Assign menu > Clear Display of Assigns** command to remove the Frame Releases identifiers.

Save the Model

During development, save the model often. Although typically you will save it with the same name, on occasion you may want to save it with a different name to record your work at various stages of development.

- A. Click the **File menu** > **Save** command, or the **Save**  button, to save your model. Specify the directory in which you want to save the model and, for this tutorial, specify the file name Truss.

Step 3 Add Area Objects

In this step, a concrete deck is added to the model.

Define the Area Sections

Make sure that the X-Z view is active. Now switch to a “plan” view, and define the properties for the concrete deck.

- A. Click the **XY View**  button to display the lowest plan view.
- B. Click the **Define menu** > **Area Sections** command. The Area Sections form will appear.
- C. Click the **Add New Sections** button in the Click to area of the form. The Area Section form shown in Figure 18 appears.
 1. Type **DECK** into the Section Name edit box.
 2. Set the Thickness (both Membrane and Bending) to **5** to indicate that the concrete deck is 5 inches thick.

By definition, a Shell object has both Membrane and Bending behavior.

3. Click the **OK** button and then click the **OK** button in the Area Sections form to accept your changes.

*Figure 18
Area Section
form*

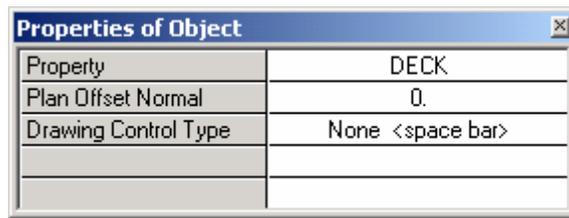
Draw the Area Object

Make sure that the X-Y Plane @ Z=0 view is active. Now draw an area object to represent the deck using the following Action Items.

- A. Click the **Draw Quad Area Element** button, or go to the **Draw menu > Draw Quad Area** command. The Properties of Object pop-up box for areas will appear as shown in Figure 19.

Make sure that the Property item in this box is set to *DECK*. If it is not, click once in the edit box opposite the Property item to activate the drop-down menu and select *DECK* from the list.

Figure 19
Properties of
Object box



2

- B. Check that the **Snap to Points and Grid Intersections** command is active. This will assist in accurately drawing the area object.
- C. Click once at point (x=0,y=0). Then moving clockwise around the model, click once at these object points in this order to draw the outline of the deck: (x=0,y=144), (x=720,y=144) and (x=720,y=0).
- D. Click on the **Select Object**  button, or Press the Esc key on the keyboard to exit the Draw Quad Area command.
- E. To better view the deck addition, click the **Set Display Options**  button. When the form appears, check the *Fill Objects* check box and the *Apply to All Windows* check box, as shown in Figure 20.

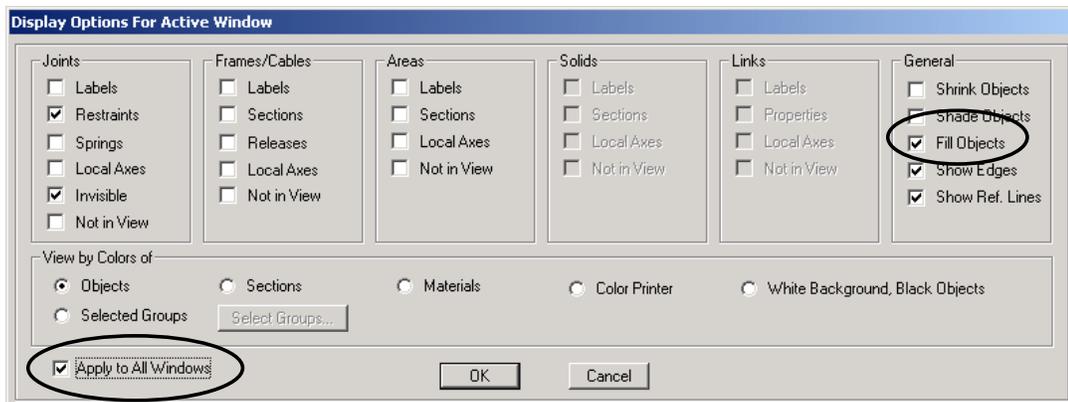


Figure 20
Display Options for
Active Windows form

- F. Click **OK** to accept the changes, and the model now appears as shown in Figure 21.

2

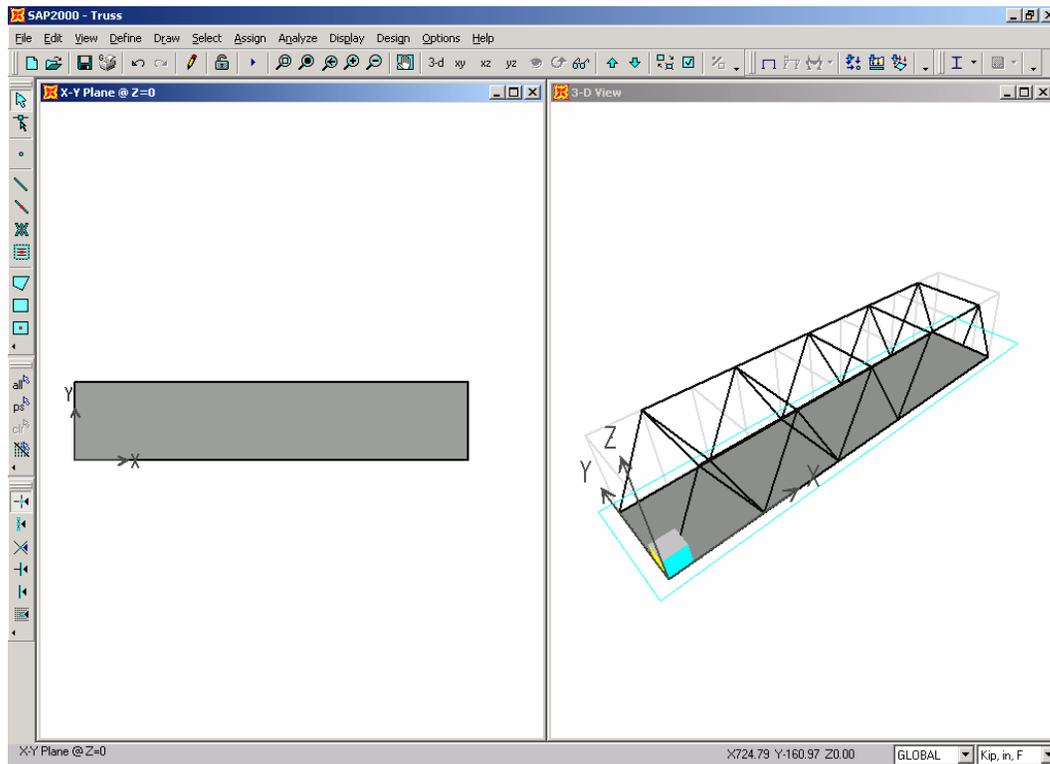


Figure 21
*Model after the area
objects have been drawn*

Mesh the Area Object

Make sure that the X-Y Plane @ Z=0 view is still active. The area object will now be meshed rather than subdivided as done for the frame objects.

- A. Draw a Selection Box from Left to Right around each of the bottom chords to select all of the points on each of these frame objects.
- B. Click anywhere on the area object to select the deck. Items currently selected are indicated in the lower left corner of the interface.
- C. Click the **Edit menu > Mesh Areas** command to bring up the Mesh Selected Shells form.

The area object representing the deck was drawn as a single object, but needs to be re-meshed into additional objects so that there will be

connectivity between the deck and the intermediate points along the chord elements. Meshing creates new objects – subdividing does not.

- D. Click on the *Mesh using selected Joints on edges* option, and then click **OK**. The model now appears as shown in Figure 22.

2

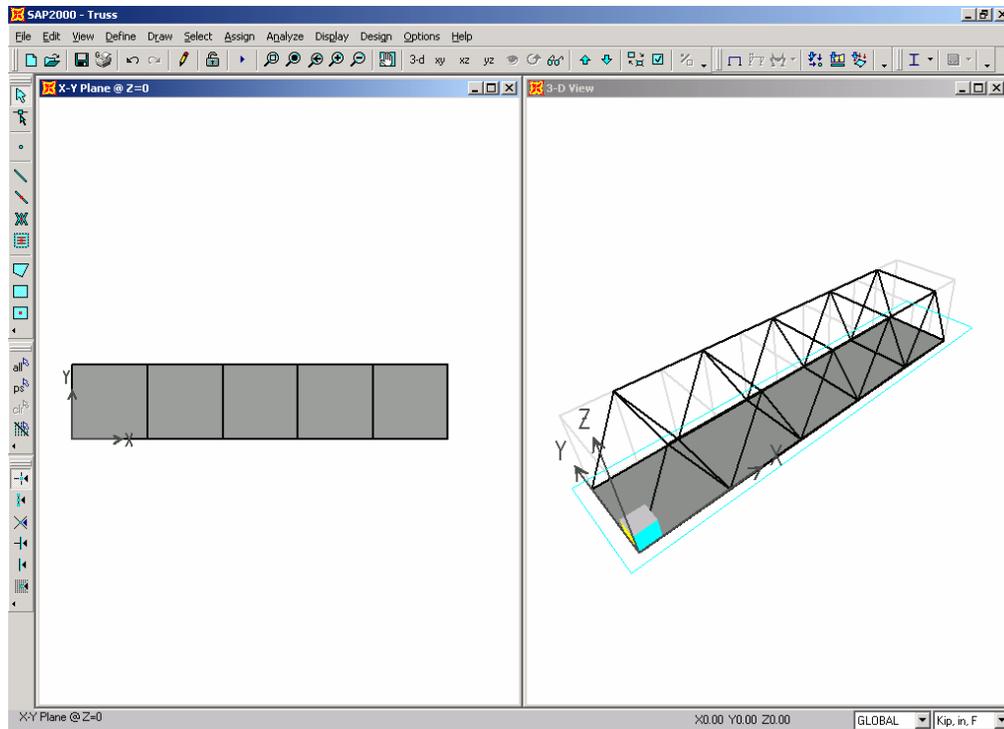


Figure 22
Model after re-mesh
of DECK area object

Step 4 Add Restraints

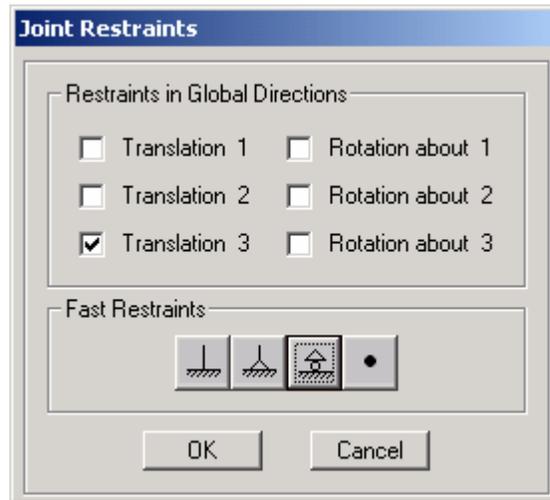
In this step, supports for the truss bridge are defined. Make sure that the X-Y Plane @ Z=0 view is still active, and that the program is in the select mode.

- A. Click on the two joints marking the right ends of the two bottom chords.

- B. Click on the **Assign menu > Joint > Restraints** command to bring up the Joint Restraints form as shown in Figure 23.

2

*Figure 23
Joint
Restraints
form*



- C. Click on the roller  button to assign restraints in the Translation 3 direction for these two joints. Click **OK** to accept the changes.
- D. Click on the two joints marking the left ends of the two bottom chords. The lower left-hand corner of the interface should indicate “2 Points selected”.
- E. Click on the **Assign menu > Joint > Restraints** command to bring up the Joint Restraints form.
- D. Click on the pinned  button to assign restraints in the Translation 1, 2, & 3 directions for these two joints. Click **OK** to accept the changes.
- F. Click the **File menu > Save** command, or the **Save**  button, to save your model.

Step 5 Define Load Cases

The loads used in this tutorial consist of dead and live static loads acting in the gravity direction.

For this example, assume that the dead consists of the self weight of the bridge plus an additional 10 pounds per square foot (psf) applied to the concrete deck. The live load is taken to be 100 psf applied to the deck.

- A. Click the **Define menu > Loads** command to bring up the Define Loads form shown in Figure 24. Note there is only a single default load case defined, which is a dead load case with self weight (DEAD).

Note that the self weight multiplier is set to 1 for the default case. This indicates that this load case will automatically include 1.0 times the self weight of all members.

Load Name	Design Type	Self Weight Multiplier	Auto Lateral Load
DEAD	DEAD	1	
DEAD	DEAD	1	

Figure 24
Define Loads form

In SAP2000, both Load Cases and Analysis Cases exist, and they may be different. However, the program automatically creates a corresponding analysis case when a load case is defined, and the analysis cases are available for review at the time the analysis is run.

- B. Click in the edit box for the Load Name column. Type the name of the new load, **LIVE**. Select a Type of load from the pull down menu;

in this case, select *LIVE*. Make sure that the Self Weight Multiplier is set to zero. Click the **Add New Load** button to add the LIVE load to the load list.

The Define Loads form should now appear as shown in Figure 25. Click the **OK** button in that form to accept all of the newly defined static load cases.

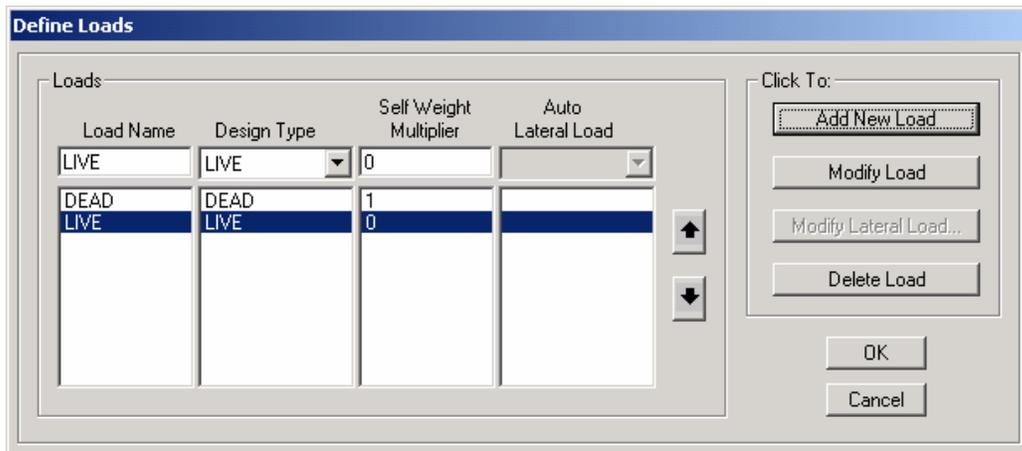


Figure 25
The Define Loads form after all load cases have been defined

Step 6 Assign Gravity Loads

In this Step, the dead and live gravity loads will be applied to the model. Make sure that the X-Y Plane @ Z=0 view is still active, and that the program is in the select mode.

- A. Draw a Selection Box from Right to Left across the entire deck to select all of the deck objects. The status bar in the lower left-hand corner should show “5 Areas Selected”. If you make a mistake in selecting, press the **Clear Selection**  button, and try again.
- B. Click the **Assign menu > Area Loads > Uniform (Shell)** command. This brings up the Area Uniform Loads form. Select *DEAD* from the Load Case Name drop-down box as shown in Figure 26.

Figure 26
Area Uniform
Loads form

The screenshot shows the 'Area Uniform Loads' dialog box with the following settings:

- Load Case Name:** DEAD
- Units:** Kip, in, F
- Uniform Load:**
 - Load:** 0.
 - Coord System:** GLOBAL
 - Direction:** Gravity
- Options:**
 - Add to Existing Loads
 - Replace Existing Loads
 - Delete Existing Loads

2

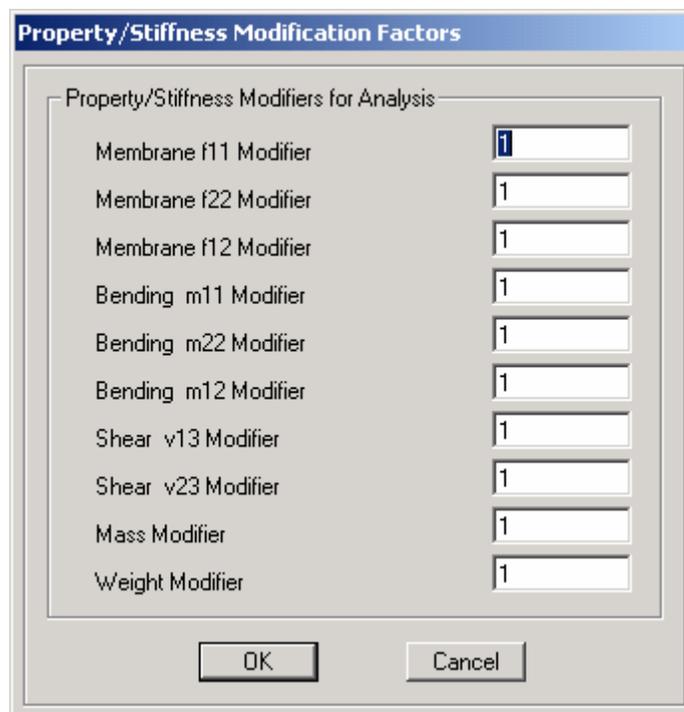
1. Select *lb-ft* from the Units drop-down box.
 2. Type **10** in the Load edit box in the Uniform Load area.

Again, remember that the *Gravity* Direction is in the negative Global Z direction.
 3. Click the **OK** button to accept the dead load.
- C. Draw a Selection Box from Right to Left across the entire deck, or click **Select menu > Get Previous Selection** command, or click the **Get Previous Selection**  button. These actions select all of the deck objects.
- D. Click the **Assign menu > Area Loads > Uniform (Shell)** command. This brings up the Area Uniform Loads form. Select *LIVE* from the Load Case Name drop-down box.
1. Select *lb-ft* from the Units drop-down box.
 2. Type **100** in the Load edit box in the Uniform Load area.
 3. Click the **OK** button to accept the live load.
- E. Click the **Assign menu > Clear Display of Assigns** command to clear the display of the assigned loads.

Step 7 Assign Area Stiffness Modifiers

In this Step, the membrane properties of the Area objects are modified to prohibit the deck from acting as a flange for the bottom chords of the trusses. Make sure that the X-Y Plane @ Z=0 view is still active, and that the program is in the select mode.

- A. Draw a Selection Box from Right to Left across the entire deck, or click **Select menu > Get Previous Selection** command, or click the **Get Previous Selection**  button. These actions select all of the deck objects.
- B. Click the **Assign menu > Area > Area Stiffness Modifiers** command to bring up the Property/Stiffness Modification Factors form shown in Figure 27.



The dialog box titled "Property/Stiffness Modification Factors" contains a list of modifiers for analysis. Each modifier has a corresponding input field. The "Membrane f11 Modifier" field is currently set to "1" and is highlighted with a blue selection box. The other modifiers are also set to "1".

Property/Stiffness Modifiers for Analysis	Value
Membrane f11 Modifier	1
Membrane f22 Modifier	1
Membrane f12 Modifier	1
Bending m11 Modifier	1
Bending m22 Modifier	1
Bending m12 Modifier	1
Shear v13 Modifier	1
Shear v23 Modifier	1
Mass Modifier	1
Weight Modifier	1

Buttons: OK, Cancel

Figure 27
Property/Stiffness
Modification
Factors form

1. Type **0** in the Membrane f11 Modifier edit box.

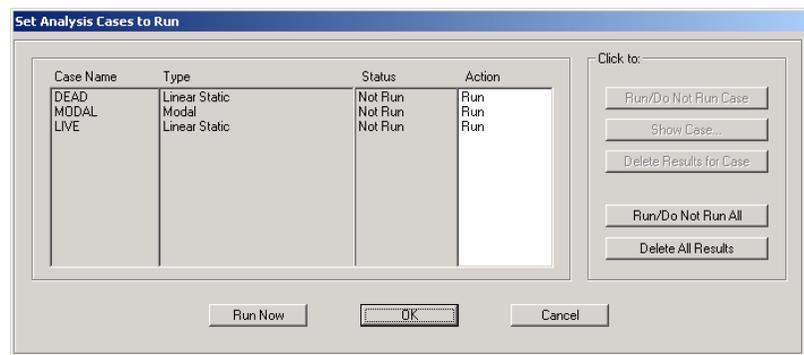
2. Type **0** in the Membrane f22 Modifier edit box.
 These actions will prohibit the deck objects from carrying in-plane axial loads.
 3. Click **OK** to accept these changes.
- C. Click the **Assign menu > Clear Display of Assigns** command to clear the display of the stiffness modifiers.
 - D. Make the 3-D View active by clicking anywhere in the window, and click the **View menu > Show Grid** command. This will shut off the grid lines in the 3-D View providing a less cluttered image of the model.
 - E. Click the **File menu > Save** command, or the **Save**  button, to save your model.

Step 8 Run the Analysis

In this Step, the analysis will be run.

- A. Click the **Analyze menu > Run Analysis** command or the **Run Analysis**  button, to bring up the Set Analysis Cases to Run form as shown in Figure 28.

Figure 28
Set
Analysis
Cases
to Run
form



Note that the program has automatically defined three different analysis cases: DEAD, MODAL and LIVE based on the load cases defined previously, as well as the assumption that the program may need modal properties for some analysis options, even though no dynamic functions have been defined.

1. Select *MODAL* from the Case Name box.
2. Click the **Run/Do Not Run Case** button to set the action for MODAL to *Do Not Run*, as we intend to run only a static analysis.
3. Click the **Run Now** button.

The program will create the analysis model from your object-based SAP2000 model, and will soon display an "Analyzing, Please Wait" window. Data will scroll in this window as the program runs the analysis. This information may be accessed at a later time by going to the **File menu > Show Input/Output Text Files** command and selecting the file with the *.LOG* extension.

- B. When the analysis is finished, the message "ANALYSIS COMPLETE" will display. Click **OK** to close the analysis window. The program automatically displays a deformed shape view of the model, and the model is locked. The model is locked when the Lock/Unlock Model button, , appears depressed. Locking the model prevents any changes to the model that would invalidate the analysis results.

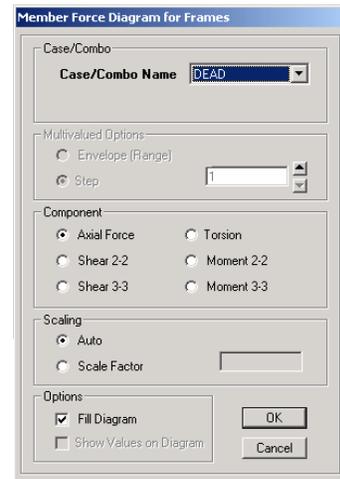
Step 9 Graphically Review the Analysis Results

In this Step, the analysis results will be reviewed using graphical representation of the results.

- A. Make sure that the X-Y Plane @ Z=0 view is active. Then click on the **XZ View**  button to reset the view to an elevation.
- B. Click the **Show Frames/Cables Forces/Stresses** button, , or the **Display menu > Show Forces/Stresses > Frames/Cables** command

to bring up the Member Force Diagram for Frames form shown in Figure 29.

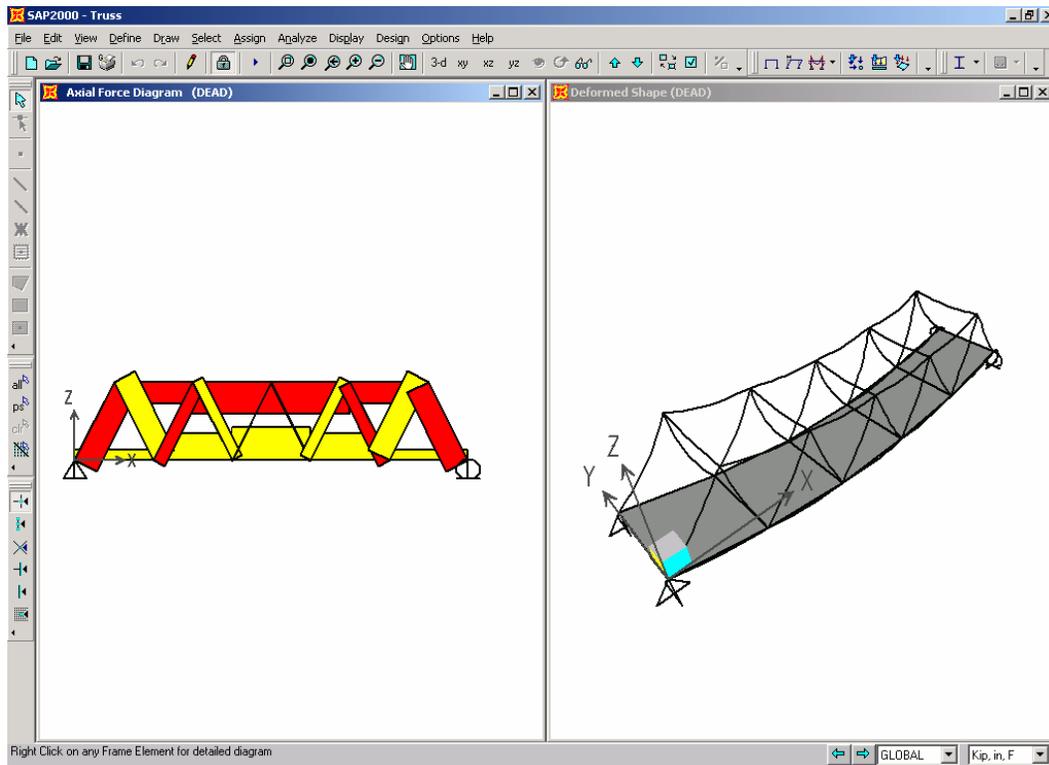
1. Select *DEAD* from the Case/Combo drop-down box.
2. Select the *Axial Force* option.
3. Check the *Fill Diagram*.
4. Click the **OK** button to generate the axial force diagram shown in Figure 30.



2

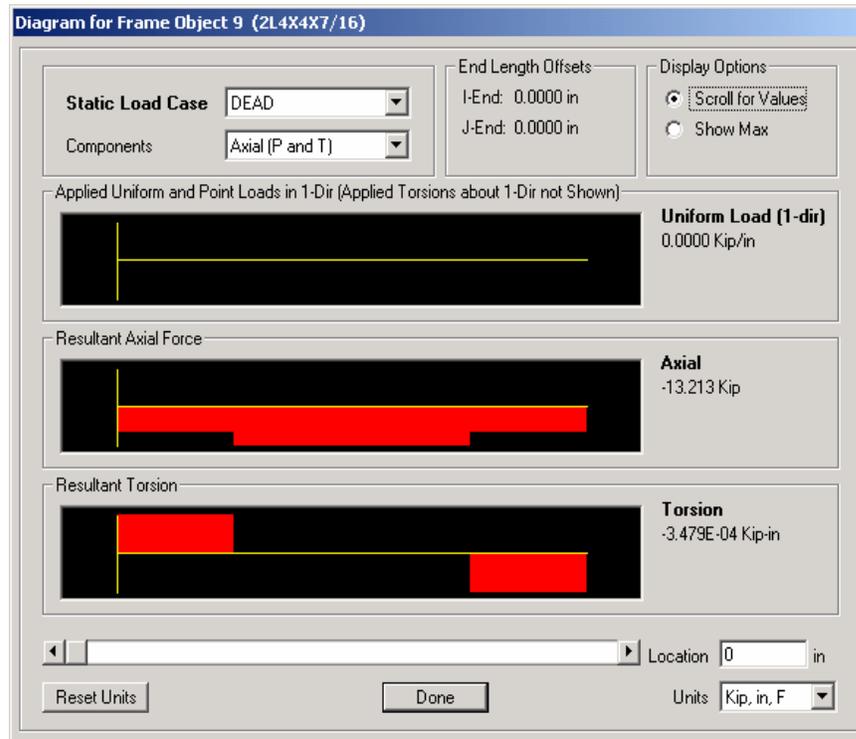
Figure 30
Axial force diagram in an elevation view

Figure 29
Member Force Diagram for Frames form



- C. Right click on the top chord member in the X-Z view to bring up the Diagram for Frame Object form shown in Figure 31.

Figure 31
Force details obtained by right-clicking top chord of truss in the elevation view in Figure 29



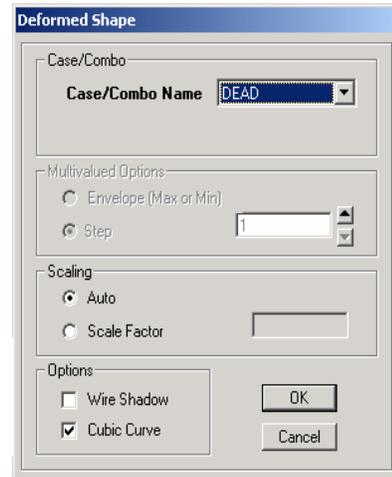
Note that the program displays the force diagrams for the entire top chord object just as it was drawn, even though the program has automatically subdivided the frame object into smaller elements for analysis.

1. Click the *Scroll for Values* option and a scroll bar appears at the bottom of the form. Drag the scroll bar with your mouse to see values at different locations along the beam.
 2. Click the **Done** button to close the form.
- D. Make sure that the X-Z View is active and then click the **Display menu > Show Deformed Shape** command or the **Show Deformed**

Shape  button, to bring up the Deformed Shape form shown in Figure 32.

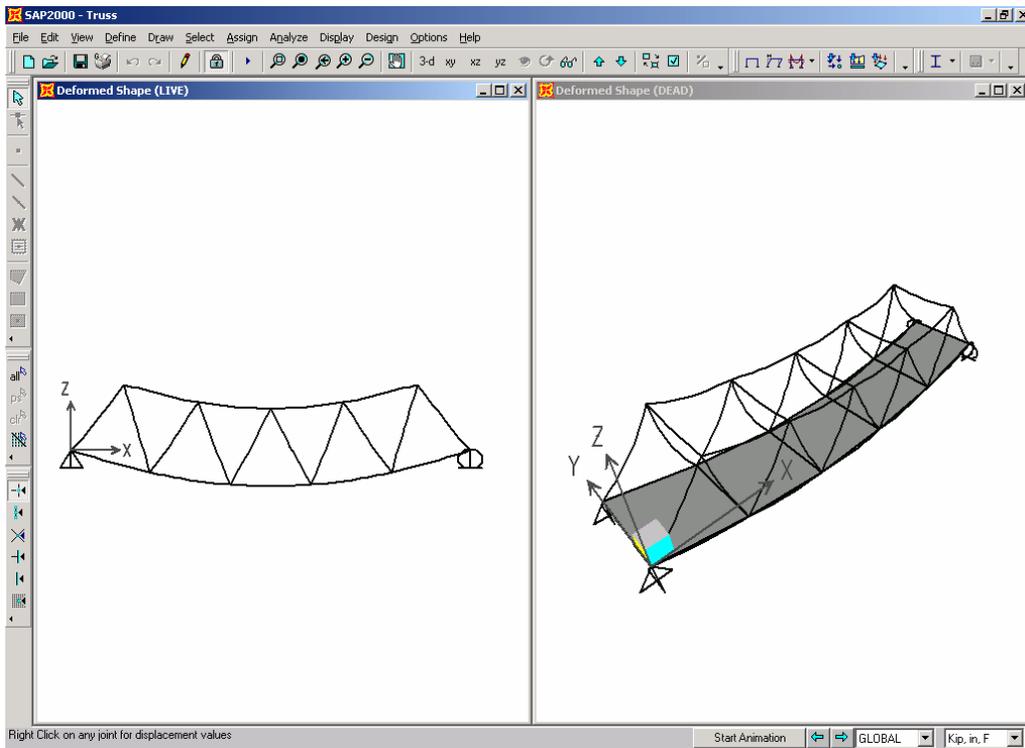
1. Select *LIVE* from the Case/Combo Name drop-down box.
2. Check the *Cubic Curve*.
3. Click the **OK** button to generate the deformed shape shown in Figure 33.

2



*Figure 32
Deformed
Shape form*

*Figure 33
Deformed Shape
in an elevation
view*



- E. Right click on the middle joint on the top chord object in Figure 33 to display the Joint Displacements results form shown in Figure 34.

2

Figure 34
Joint Displacements
 obtained by right-clicking a joint shown in the elevation view in Figure 32



Joint ID	19		
	1	2	3
Trans	0.02697	6.693E-05	-0.13149
Rotn	-1.121E-05	0.00000	0.00000

Note that local object axis 3 is in the positive global Z direction.

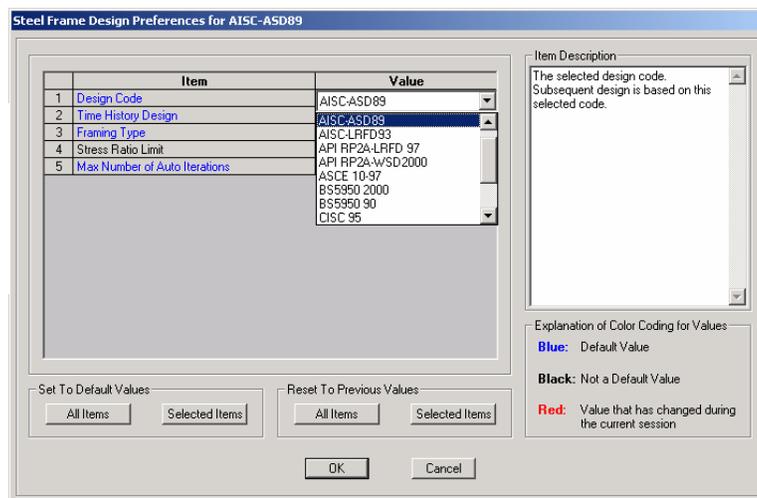
- F. Close the Joint Displacements form.

Step 10 Design the Steel Frame Objects

In this Step, the steel frame members of the trusses will be designed. Note that the analysis should be run before completing the following Action Items.

- A. Click the **Options menu > Preferences > Steel Frame Design** command. The Steel Frame Design Preferences form shown in Figure 35 appears.

Figure 35
Steel Frame Design Preferences form



Item	Value
1 Design Code	AISC-ASD89
2 Time History Design	AISC-ASD89
3 Framing Type	AISC-LRFD93
4 Stress Ratio Limit	API RP2A-LRFD 37
5 Max Number of Auto Iterations	ASCE 10-97

Item Description: The selected design code. Subsequent design is based on this selected code.

Explanation of Color Coding for Values:
Blue: Default Value
Black: Not a Default Value
Red: Value that has changed during the current session

1. Click in the Design Code Values edit box to see the available design codes. Select the *AISC-ASD89* code.
 2. Review the information contained in the other edit boxes and then click **OK** to accept the selections.
- B. Click the **Design menu > Steel Frame Design > Start Design/Check of Structure** command or the **Start Steel Design/Check of Structure** button, to start the steel frame design process. The program designs the steel members, selecting the optimum member size from the TRUSS auto select section list assigned to them when they were drawn.

When the design is complete, the selected sizes are displayed on the model. The model appears as shown in Figure 36.

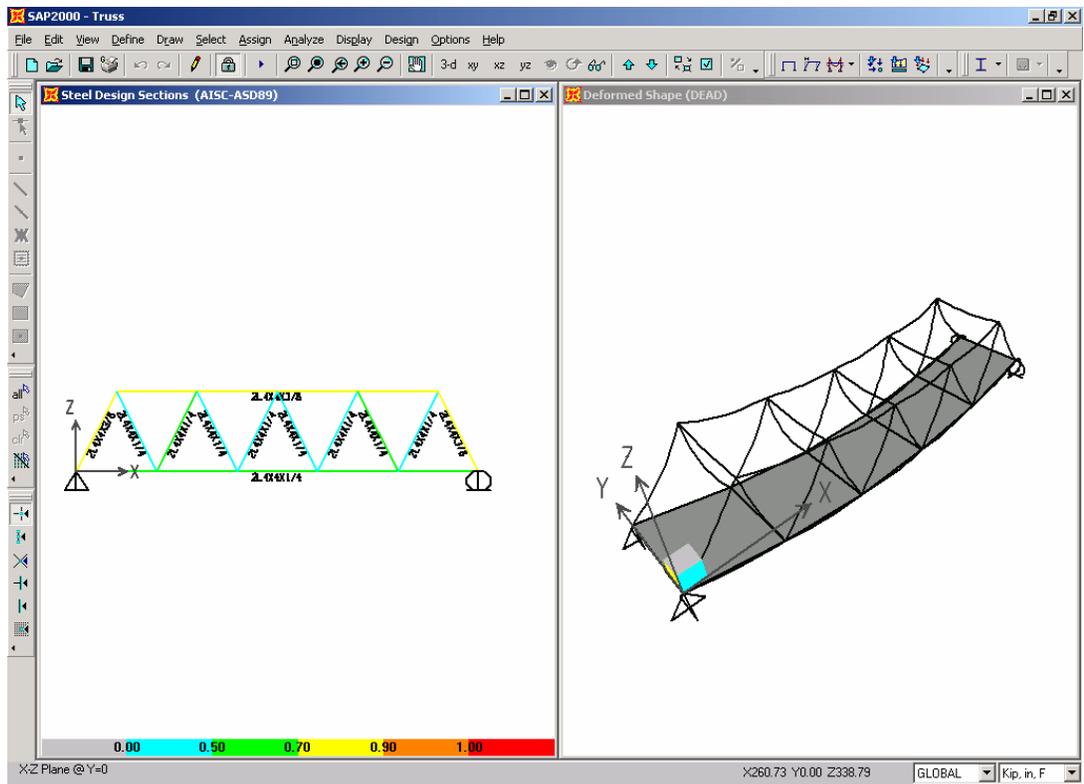
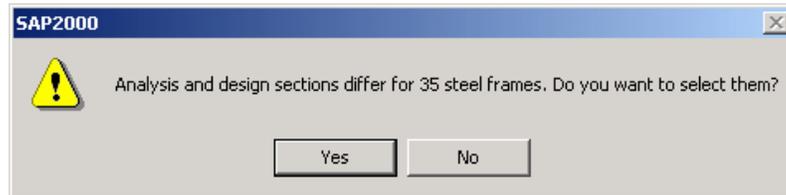


Figure 36
Model after the initial steel frame design

Figure 37
*Analysis vs Design
Section warning
message*



- C. Click the **Design menu > Steel Frame Design > Verify Analysis vs Design Section** command. A message similar to the one in Figure 37 appears. Click the **No** button to close the form.

In the initial analysis (Step 8), the program used the median section by weight from the TRUSS auto select section list. During design (this Step), the program selected different sections than those that were used in the analysis. The message in Figure 37 indicates that the analysis and design sections are different.

The goal is to repeat the analysis and design process until the analysis and design sections are all the same. Note that when the bridge is reanalyzed, SAP2000 will use the current design sections (i.e., those selected in Step 10) as new analysis sections for the next analysis run.

- D. Right click on one of the truss top chord member in the X-Z view shown in Figure 36. The Steel Stress Check Information form shown in Figure 38 appears. Note that the reported analysis and design sections are different.

The main body of the form lists the design stress ratios obtained at various stations along the frame object for each design load combination. Note that the program automatically created code-specific design load combinations for this steel frame design.

Also note that the program designed the chord as a single physical member, just as it was drawn as a single object, even though the program has automatically subdivided the frame object into smaller elements for analysis.

Steel Stress Check Information

Frame ID: 9
Design Code: AISC-ASD89
Analysis Section: 2L4x4x7/16
Design Section: 2L4x4x3/8

COMBO ID	STATION LOC	MOMENT RATIO	INTERACTION CHECK	MAJ-SHR RATIO	MIN-SHR RATIO
DSTL1	432.00	0.254 (C)	= 0.216 + 0.037 + 0.000	0.003	0.000
DSTL1	504.00	0.252 (C)	= 0.216 + 0.036 + 0.000	0.000	0.000
DSTL1	576.00	0.269 (C)	= 0.216 + 0.052 + 0.000	0.003	0.000
DSTL2	0.00	0.551 (C)	= 0.451 + 0.098 + 0.001	0.004	0.000
DSTL2	72.00	0.508 (C)	= 0.451 + 0.056 + 0.000	0.000	0.000
DSTL2	144.00	0.468 (C)	= 0.451 + 0.017 + 0.000	0.003	0.000
DSTL2	144.00	0.785 (C)	= 0.677 + 0.108 + 0.000	0.003	0.000

Buttons: 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 38
Steel Stress Check Information form

Click the **Details** button on the Steel Stress Check Information form. The Steel Stress Check Information AISC-ASD89 form shown in Figure 39 appears. Note that you can print this information using the File menu on the form.

Figure 39
Steel Stress Check Information AISC-ASD89 form

Steel Stress Check Information AISC-ASD89

File Edit View Help

AISC-ASD89 STEEL SECTION CHECK
Combo : DSTL2
Units : Kip, in, F

Frame : 9
X Mid : 368.000
Y Mid : 0.000
Z Mid : 194.000
Length : 576.000
RLFF : 1.000

Design Sect: 2L4x4x3/8
Design Type: Beam
Frame Type: Moment Frame
Sect Class: Non-Compact
Major Axis: 0.000 degrees counterclockwise from local 3

Area : 5.710
IMajor : 8.640
IMinor : 15.346
Ixg : 0.000

SMajor : 3.010
SMinor : 3.937
rMajor : 1.290
rMinor : 1.639

AMajor : 3.000
AMinor : 3.000
E : 29000.000
Fy : 36.000

STRESS CHECK FORCES & MOMENTS							
Location	P	M33	M22	U2	U3	T	
144.000	0.000	0.000	0.000	0.000	0.000	0.000	

PHI DEMAND/CAPACITY RATIO							
Equation (H1-1)	Total Ratio	P Ratio	MMajor Ratio	MMinor Ratio	Ratio Limit	Status	Check
(H1-1)	0.785	= 0.677	+ 0.000	+ 0.108	0.950	OK	

AXIAL FORCE DESIGN							
	P	Fa	Fa	Fc			
Force		Stress	Allowable	Allowable			
Axial	-41.374	7.246	10.699	21.600			

MOMENT DESIGN							
	M	Fb	Fb	Fc	Cb	K	L
Moment		Stress	Allowable	Allowable	Factor	Factor	Factor
Major Moment	-2.760	0.917	21.600	10.897	0.850	1.000	0.250
Minor Moment	0.011	0.003	21.600	19.355	0.850	1.000	0.250

SHEAR DESIGN							
	U	Fv	Fv		Status	T	
Force		Stress	Allowable	Allowable	Check	Factor	
Shear	0.000	0.000	61.330	61.330	OK		

Units: Kip, in, F

Click the **X** in the upper right-hand corner of the Steel Stress Check Information AISC-ASD89 form to close it.

Click the **Cancel** Button to close the Steel Stress Check Information form.

- E. To rerun the analysis with the new analysis sections, click the **Analyze menu > Run Analysis** command or the **Run Analysis** button, and then click the **Run Now** button on the Set Analysis Cases to Run form.
- F. When the analysis is complete, click the **OK** button to close the analysis window. Click the **Design menu > Steel Frame Design > Start Design/Check of Structure** command or the **Start Steel Design/Check of Structure** button, to start the steel frame design process.
- G. When the design is complete, click the **Design menu > Steel Frame Design > Verify Analysis vs Design Section** command. A message similar to the one in Figure 40 appears.

*Figure 40
Analysis vs Design
Section message*



The message in Figure 40 indicates the number of analysis sections that differ from the design sections. Click the **No** button if sections do not match, or the **OK** button if they do match, to close the form.

Repeat Action Items E through G until the message received indicates that all analysis and design sections match. This may take numerous iterations depending upon the complexity of the model.

- H. When the analysis and design sections are the same, click the **Design menu > Steel Frame Design > Verify all Members Passed** command. A form similar to that shown in Figure 41 should appear indicating that all members passed.

Figure 41
*Stress/capacity
check message*

**2**

Note that members not passing at this stage is an indication of inadequate sections in the auto select list. The program would have used the largest section in the auto select list for both the analysis and design, so the message stating that members do not pass indicates that the auto select section list needs modification. In that case, either add more sections to the auto select sections list or assign larger sections to the members that did not pass and continue the analysis and design iteration.

- I. Click the **File menu** > **Save** command, or the **Save**  button, to save your model.

This introductory tutorial for SAP2000 Version 8 is now complete.