

SAP2000[®]

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

INTRODUCTORY TUTORIAL



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

Version 11.0
October 2006

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., 1978-2006.

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

Chapter 1	Introduction	
	Using This Manual	1-1
	Overview of the Program	1-1
	Using this Tutorial	1-2
Chapter 2	An Introductory Tutorial	
	The Project	2-2
	The Interface	2-2
Step 1	Begin a New Model	2-3
	Define a Material	2-6
	Define an Auto Select Section List	2-7
Step 2	Add Frame Objects	2-11
	Draw Frame Objects	2-11
	Replicate Objects	2-13
	Trim Objects	2-16
	Assign Member End Releases	2-19
	Save the Model	2-20

Step 3	Add Area Objects	2-21
	Define the Area Sections	2-21
	Draw the Area Object	2-22
	Mesh the Area Object	2-24
Step 4	Add Restraints	2-26
Step 5	Define Load Cases	2-27
Step 6	Assign Gravity Loads	2-29
Step 7	Assign Area Stiffness Modifiers	2-31
Step 8	Run the Analysis	2-32
Step 9	Graphically Review the Analysis Results	2-34
Step 10	Design the Steel Frame Objects	2-38

Chapter 1

Introduction

Using this Manual

This manual introduces you to SAP2000 Version 11. 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 the program. Completing the tutorial will give you hands-on experience working with SAP2000, which for most people is the quickest way to become familiar with the program.

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

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 using 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 meshing 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.

Overview of the Tutorial

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

We recommend that you perform each step of the tutorial as you read the manual. Therefore, the program should be installed on your computer before you begin. Prepare to spend at least one hour going through the example. If at any time you need to stop, save your model so that you may continue at a later time.

During the course of the tutorial, we will explore many of the basic features of SAP2000. We hope that you enjoy and find this approach helpful as a starting point in your use of this powerful and comprehensive version of SAP2000.

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.

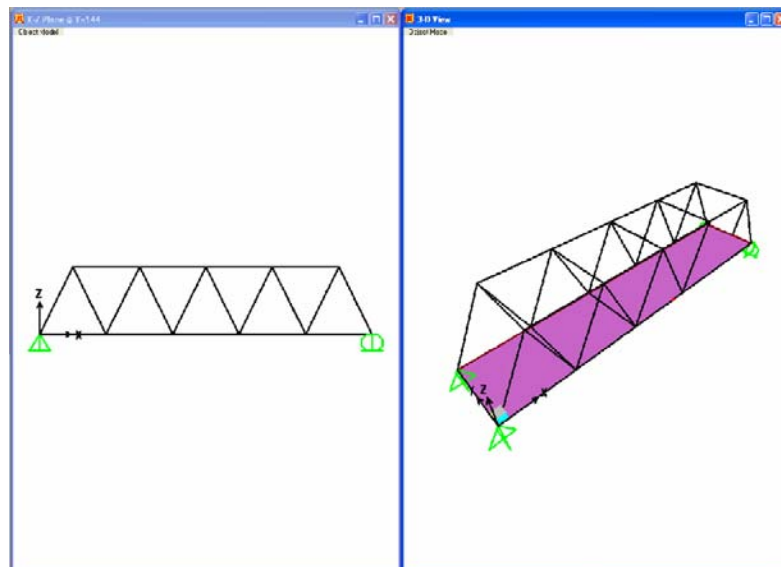


Figure 1
The Tutorial
Model

The Project

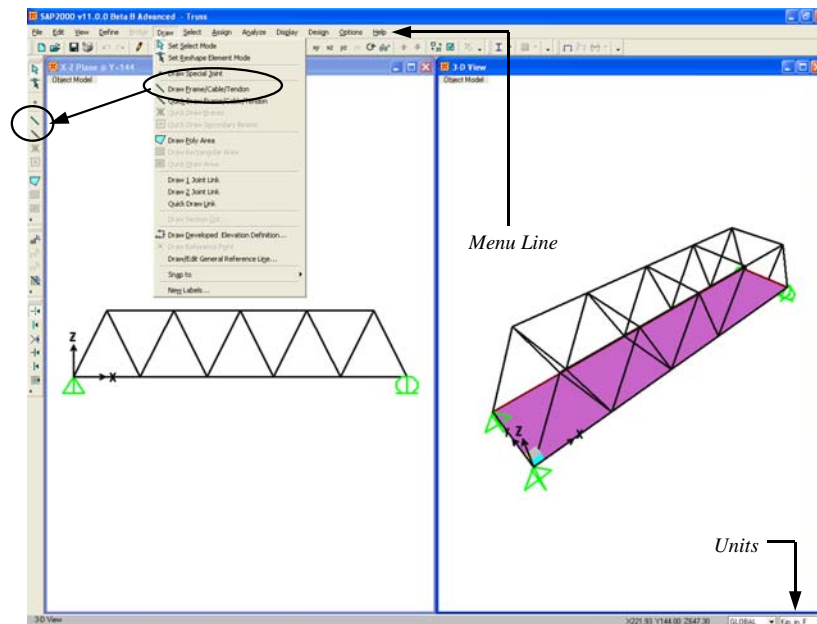
The tutorial project is a five panel, sloped truss bridge. The bridge spans 60 feet, and both the width and height of the panels are 12 feet. The supports are rollers at one end, and pins at the other.

The trusses and cross members are to be constructed of 2L4X4s, 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 that will serve as a template for developing the model will be defined. Then a material will be defined and a list of double angle sections will be selected for the truss Auto Select Section 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.

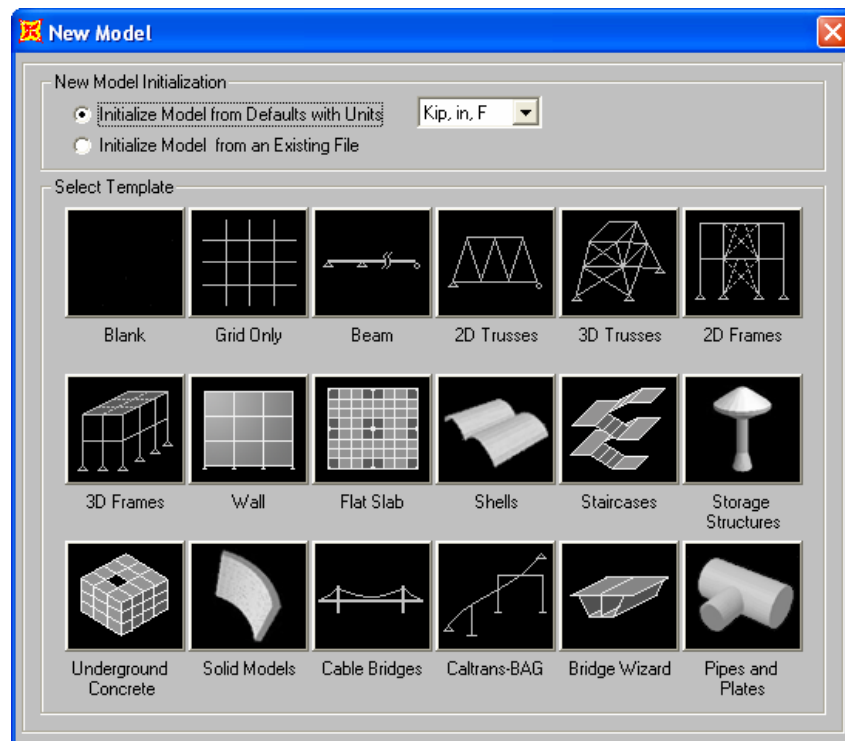


Figure 3
New
Model
form

- B. The New Model 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 model, so it is advisable to spend time carefully planning the number and spac-

ing of the grid lines. Select the **Grid Only** button, and the form shown in Figure 4 will display.

Figure 4
Quick Grid Lines form

The screenshot shows the 'Quick Grid Lines' dialog box with the following settings:

Section	Direction	Value
Coordinate System Name	Coordinate System Name	GLOBAL
	Number of Grid Lines	
	X direction	11
Grid Spacing	X direction	72
	Y direction	144
	Z direction	144
First Grid Line Location	X direction	0.
	Y direction	0.
	Z direction	0.

Buttons: OK, Cancel

- C. The Quick Grid Lines form is used to specify the grids and spacing in the X, Y and Z directions. Set the number of grid lines to 11 for the X direction, and to 2 for the Y and Z directions. Type **6 ft** (including the 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. The values specified in the First Grid Line Location area locate the origin of the grid lines – make sure that these values are all set to zero for this tutorial.

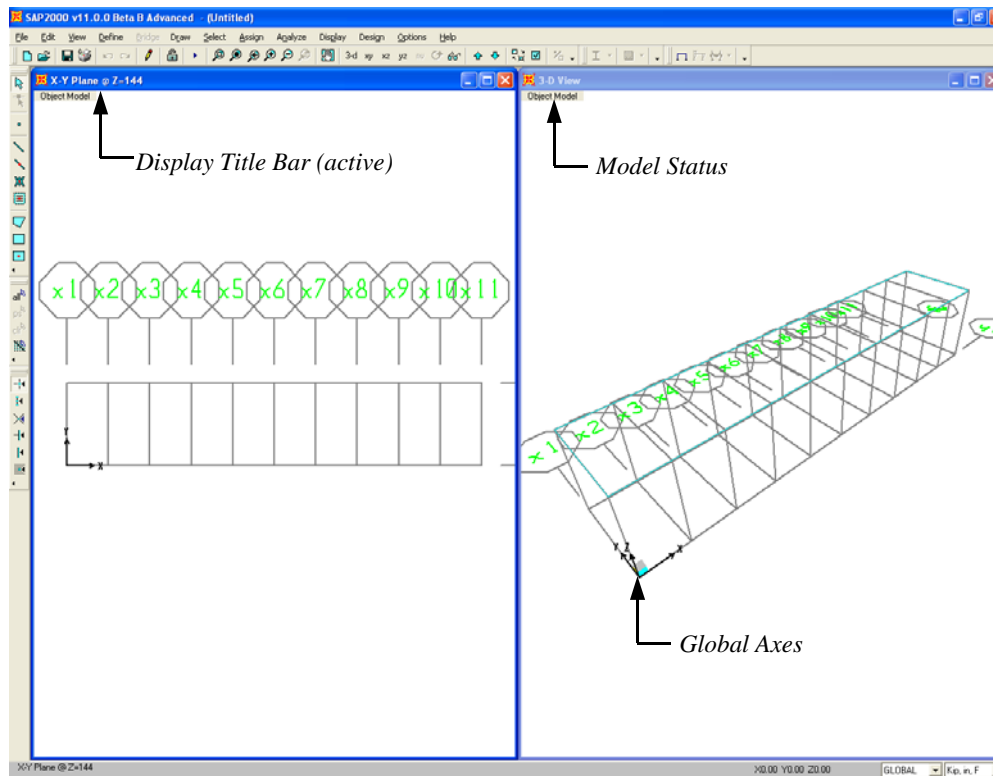


Figure 5
The SAP2000 windows

- D. Click the **OK** button to accept the changes, and the program will appear as shown in Figure 5. Note that the grids appear in two view windows tiled vertically, an X-Y “Plan” View on the left and a 3-D View on the right. The number of view windows may be changed using the **Options menu > Windows** command.

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.

Directly under the display title bar is an indicator showing the model status. This indicator will display either “Object Model” or “Analysis Model” – objects represent physical members and are typically what

the user draws in SAP2000, while the analysis model shows the elements that are generated by meshing those objects.

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

- E. To make viewing our model easier, we will reduce the size of the grid bubbles. Click the **Define menu > Coordinate Systems/Grids** command. The Coordinate/Grid Systems form will display.
1. Make sure that the Systems item on the Coordinate/Grid Systems form has *Global* highlighted and click the **Modify/Show System** button. The Define Grid Data form will appear.
 2. Type **36** into the Bubble Size edit box.
 3. Click the **OK** buttons to close the Define Grid Data and Coordinate/Grid Systems forms.

Define a Material

Two default material properties are predefined; one for concrete and one for steel. A third material property will be added for the double angle sections. Varying levels of sophistication may be used to define the materials, including inputting advanced nonlinear stress-strain curve data. For this tutorial, the “Quick” material definition option will be used.

- A. Click the **Define menu > Materials** command to display the Define Materials form shown in Figure 6.

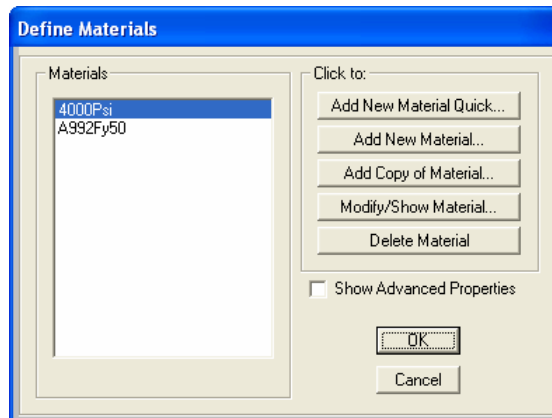
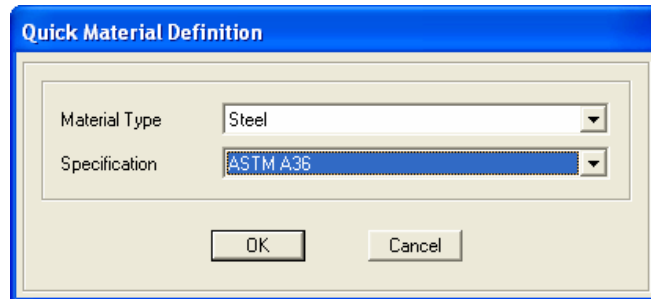


Figure 6
Define Materials form

- B. Click the **Add New Material Quick** button to display the Quick Material Definition form shown in Figure 7.

Figure 7
*Quick Material
Definition form*



- C. The Quick Material Definition form allows for the rapid selection of material types from predefined standards. Select *Steel* from the Material Type drop-down list.
- D. Select *ASTM A36* from the Specification drop-down list; the program has all of the properties needed for this material type already defined.
- E. Click the **OK** buttons to close the Quick Material Definition and Define Materials forms.

Define an Auto Select Section List

An auto select section list is simply a list of sections. 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 (2L4X4s), which will be chosen from an auto select sections list created now.

- A. Click the **Define menu > Frame Sections** command, which will display the Frame Properties form shown in Figure 8.

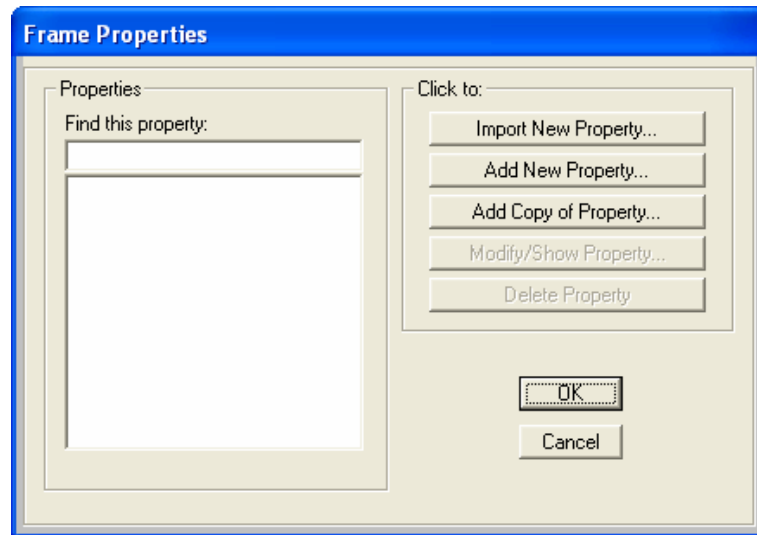


Figure 8
The Frame
Properties
form

- B. Click the **Import New Property** button, which will display the Import Frame Section Property form shown in Figure 9.

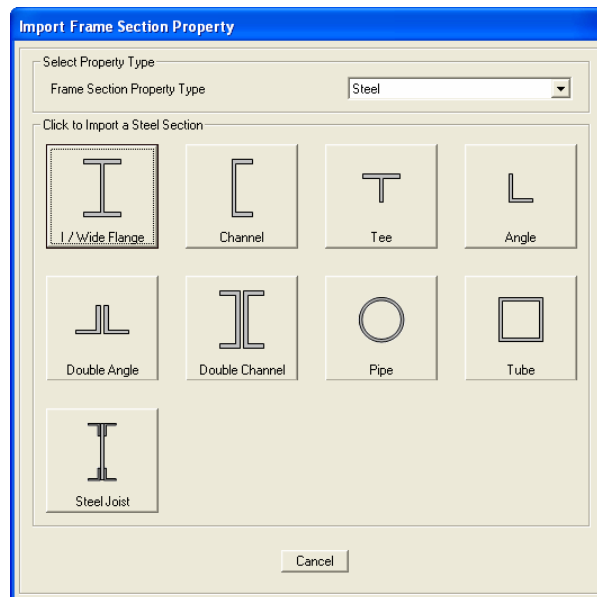
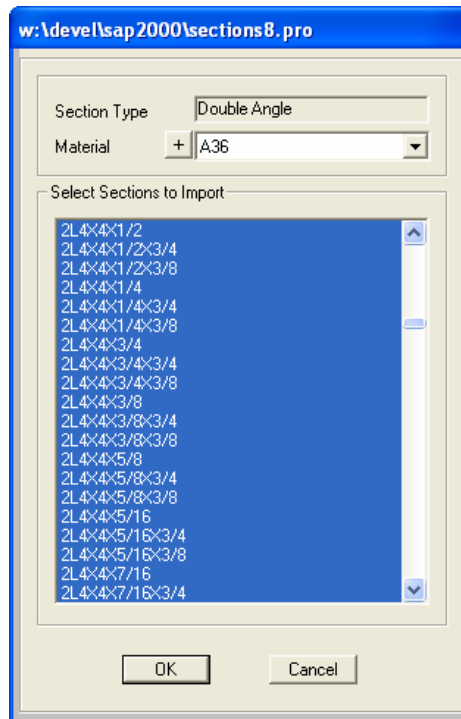


Figure 9
Import Frame
Section
Property form

- C. Verify that *Steel* is showing in the Frame Section Property Type drop-down list and then in the Click to Import a Steel Section area of the Import Frame Section Property form, click the **Double Angle** button, which will open the Section Property File form.
- D. Use the Section Property File form to locate the *SECTIONS8.PRO* file, which contains the properties of the double angles to be used in the model. The SECTIONS8.PRO file likely will be stored with the program files for SAP2000. Highlight the file name and click the **Open** button. The Sections8.pro sections list form shown in Figure 10 displays.
- E. Select A36 from the Material drop-down list – this is the material property defined in the previous section. Clicking on the **+** button will display the Define Materials form where material properties may be altered or added.
- F. Scroll down the list of double angles in the Select Sections to Import area until you find the first 2L4X4. Click once on that member to highlight it.

Figure 10
Sections8.pro
sections
list



- 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 2L4X4X7/16X3/8; all of the 2L4X4s 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.
- I. Click the **Add New Property** button in the Click to area of the Frame Properties form and the Add Frame Section Property form will display.
- J. In the Frame Section Property Type drop-down list, select *Steel*.
- K. Click the **Auto Select List** button to display the Auto Selection Sections form shown in Figure 11.

Figure 11
Auto Selection Sections form

Auto Selection Sections

Auto Section Name TRUSS

Section Notes

Choose Sections:

List of Sections	Auto Selections
	2L4x4x3/8x3/8
	2L4x4x5/8
	2L4x4x5/8x3/4
	2L4x4x5/8x3/8
	2L4x4x5/16
	2L4x4x5/16x3/4
	2L4x4x5/16x3/8
	2L4x4x7/16
	2L4x4x7/16x3/4
	2L4x4x7/16x3/8


Starting Section

- L. Type **TRUSS** in the Auto Section Name edit box.
- M. Locate the *2L4X4X1/2* double angle under the List of Sections, and click once to highlight it.
- N. Continue down the list until you find the last double angle, *2L4X4X7/16X3/8*, and while holding down the shift key on the keyboard, click once on this section. All of the 2L4X4s should now be highlighted.
- O. Click the **Add** button to move the selected list to the Auto Selections edit box on the right side of the form.
- P. 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 list.

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 using **Edit menu** commands.

Draw Frame Objects

Make sure that the X-Y Plane @ Z=144 view is active (see Step 1-D for directions on how to make a view active). 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 **View menu > Set 2D View** command.
 - 1. Click on the *X-Y plane* option.
 - 2. Type **0** into the Z= edit box to display the plan view at the lower elevation, and click **OK**.




- B. Click the **Draw Frame/Cable/Tendon**  button or use the **Draw menu > Draw Frame/Cable/Tendon** command. If you accessed the **Draw Frame/Cable/Tendon** command via the **Draw menu**, the **Draw Frame/Cable/Tendon** button will depress verifying your command selection. The Properties of Object pop-up form for frames will appear as shown in Figure 12.


Figure 12
Properties of Object form

Properties of Object 	
Line Object Type	Straight Frame
Section	TRUSS
Moment Releases	Continuous
X-Y Plane Offset Normal	0.
Drawing Control Type	None <space bar>

If the Properties of Object form is covering any part of the model in either view, click on the blue title bar and drag it out of the way.

- C. Click in the Section drop-down list 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.
- D. 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/Tendon command active so that you may add additional objects.

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 B-D.

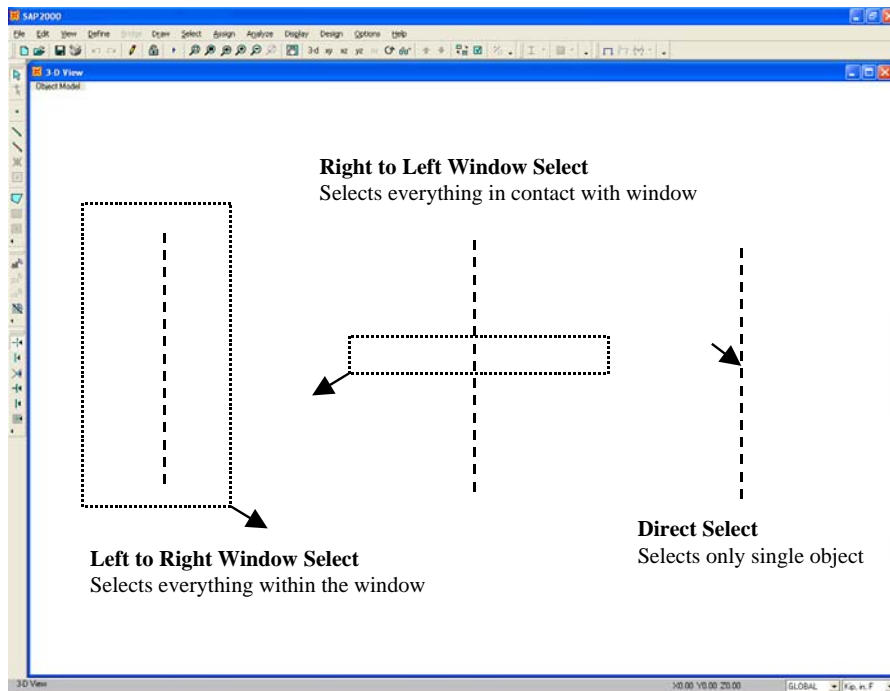
- E. Repeat Item D, drawing an additional frame object parallel to the first member from (x=0, y=144) to (x=720, y=144). These members form the bottom chords of the trusses. Right click to stop drawing.
- F. Left click at (x=0, y=0) and then at (x=0, y=144) to draw the first transverse member.
- G. Click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

Replicate Objects

Make sure that the program is in the Select mode.

- A. Select the transverse member spanning between the longitudinal chords by left clicking directly on the member, or left clicking to the right of the object, holding the left mouse button down, and dragging the mouse across the member. See Figure 13 for selection options.

*Figure 13
Graphical
Selection
Options*



- B. Click the **Edit menu > Replicate** command to access the form shown in Figure 14.

Figure 14
Replicate form

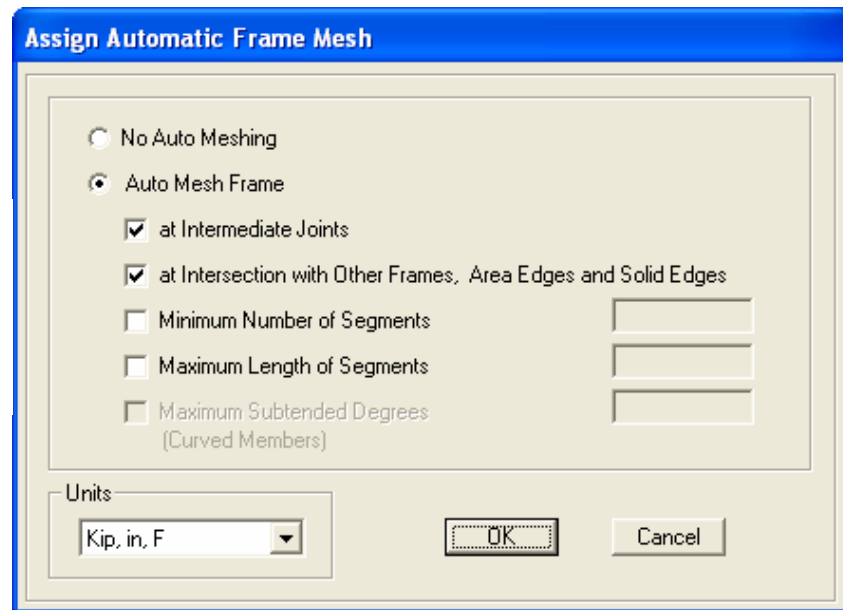
The image shows the 'Replicate' dialog box with the 'Linear' tab selected. The 'Increments' section has dx set to 144, dy to 0, and dz to 0. The 'Increment Data' section has the 'Number' set to 5. The 'Replicate Options' section shows '7 of 7 active boxes are selected' and the 'Delete Original Objects' checkbox is unchecked. The 'OK' and 'Cancel' buttons are at the bottom.


- 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.
- F. Click on the **Select menu > Select > Select Lines Parallel To > Click Straight Line Object** command and left click once on the longitudinal chord member along the X axis. This select command selects the other chord object that is parallel as well.
- G. Click on the **Assign menu > Frame/Cable/Tendon > Automatic Frame Mesh** command to access the form in Figure 15. Select the *Auto Mesh Frame* option and check the *at Intermediate Joints* and *at Intersection with Other Frames...* check boxes, and click **OK**.

This meshing 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 in the Physical model they will remain as single objects.

Figure 15
Assign
Automatic
Frame Mesh
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 access 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.

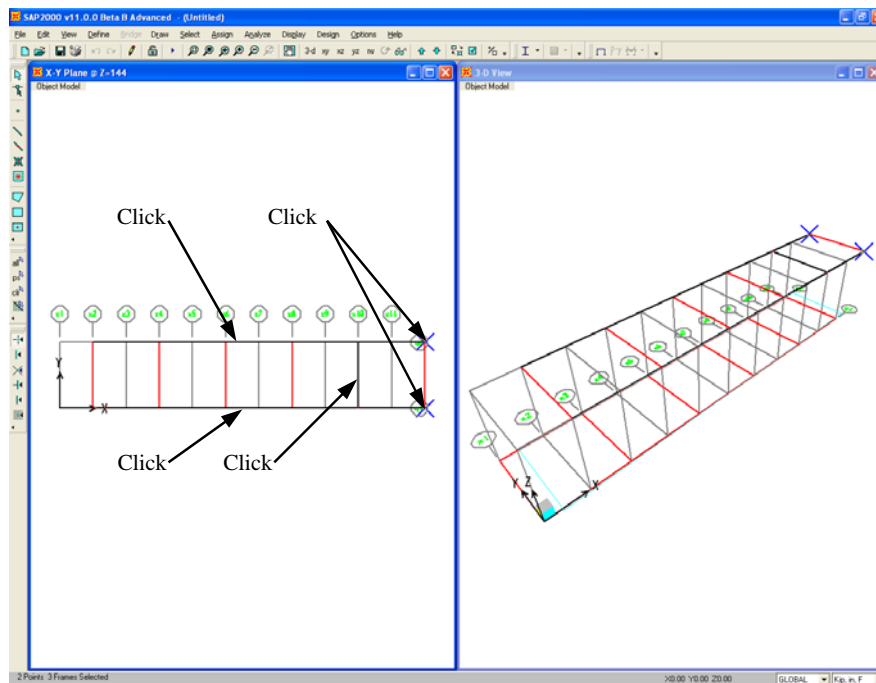
The framing at the bottom plan will be replicated at the top level with a shift of 72 inches in the X direction.

Trim 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.
 3. 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, and click **OK**.
- 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 16. The selected objects should be shown as dashed lines.

Figure 16
Select mode
for Trim



D. Click the **Edit menu > Edit Lines > Trim/Extend Frames** command to access the Trim/Extend Selected Frames form.

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

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.

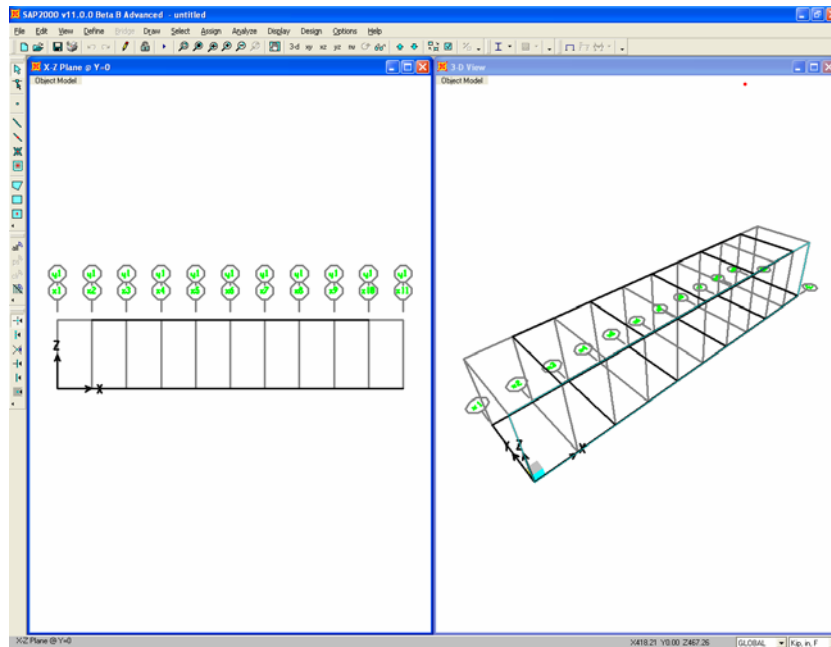
F. Make sure that the plan view is active and click the **View menu > Set 2D View** command.



4. In the Set 2D View form click on the *X-Z plane* option.

2. Type **0** into the Y= edit box and click **OK**.

Your model now appears as shown in Figure 17.

Figure 17
Model after
frame
objects
have been
added in
plan

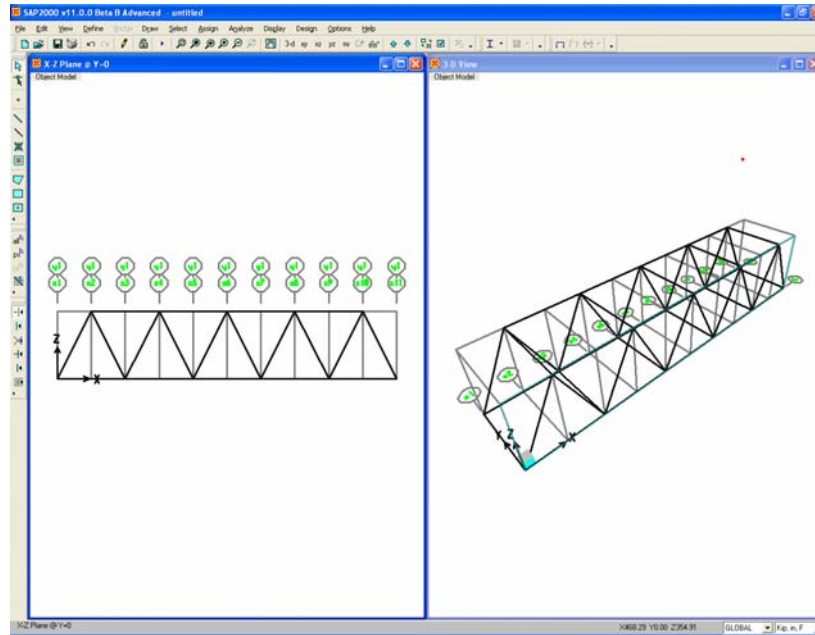


- G. Click the **Draw Frame/Cable/Tendon**  button or use the **Draw menu > Draw Frame/Cable/Tendon** command. The Properties of Object pop-up form for frames will display.
- H. Make sure that the Section item on the Properties of Object form is set to *TRUSS*.
- I. To draw the first diagonal, left click 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 ($x=144$, $z=0$).
- J. Diagonals for one bay are now drawn.
- K. Right click and then click on the **Select Object**  button, or Press the Esc key on the keyboard to exit the **Draw Frame/Cable/Tendon** command.
- L. Draw a Selection Box from Right to Left across the two diagonals just drawn to select both diagonals. See Figure 13 for selection options.
- M. Click the **Edit > Replicate** command to access 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.
- N. Draw a Selection Box from Right to Left across all of the diagonals.
- O. 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 18.

Figure 18
Model after
all frame
objects have
been added



Assign 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 the **Assign menu > Frame/Cable/Tendon > Releases/Partial Fixity** command to access the form shown in Figure 19. 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.

- C. Click the **OK** button to accept the changes and return to the Select mode.

Figure 19
Assign Frame Releases form


Frame Releases	Release		Frame Partial Fixity Springs	
	Start	End	Start	End
Axial Load	<input type="checkbox"/>	<input type="checkbox"/>		
Shear Force 2 (Major)	<input type="checkbox"/>	<input type="checkbox"/>		
Shear Force 3 (Minor)	<input type="checkbox"/>	<input type="checkbox"/>		
Torsion	<input type="checkbox"/>	<input type="checkbox"/>		
Moment 22 (Minor)	<input type="checkbox"/>	<input type="checkbox"/>		
Moment 33 (Major)	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0.	0.

No Releases

- 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 and click **OK**. Alternatively, use the **Move Up in List**  button.
- E. Draw a Selection Box from Right to Left across all of the diagonals.
- F. Click the **Assign menu > Frame/Cable/Tendon > Releases/Partial Fixity** command to access 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 the **OK** button to accept the 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 **View menu** > **Set 2D View** command. In the Set 2D View form click on the *X-Y plane* option, type **0** into the Z= edit box and click the **OK** button.
- B. Click the **Define menu** > **Area Sections** command. The Area Sections form will display.
- C. Make sure that the Select Section Type to Add item is set to *Shell*. Click the **Add New Section** button in the Click to area of the form. The Shell Section Data form shown in Figure 20 displays.
 1. Type **DECK** into the Section Name edit box.
 2. Verify that the Material Name is set to *4000Psi* in the Material area. Clicking on the **+** button will display the Define Materials form where material properties may be altered or added.
 3. 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.

4. Click the **OK** button and then click the **OK** button in the Area Sections form to complete the deck definition.

Figure 20
Shell Section Data
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 Poly Area** button , or go to the **Draw menu > Draw Poly Area** command. The Properties of Object pop-up form for areas will display as shown in Figure 21.

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

Figure 21
Properties of
Object box

Properties of Object	
Section	DECK
Drawing Control Type	None <space bar>

- 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. Press the Enter key on your keyboard to stop drawing.
- E. Click on the **Select Object**  button, or Press the Esc key on the keyboard to exit the **Draw Poly Area** command.
- F. To better view the deck addition, click the **Set Display Options**  button, or go to the **View menu > Set Display Options** command. When the form appears, check the *Fill Objects* check box and the *Apply to All Window* check box, as shown in Figure 22.

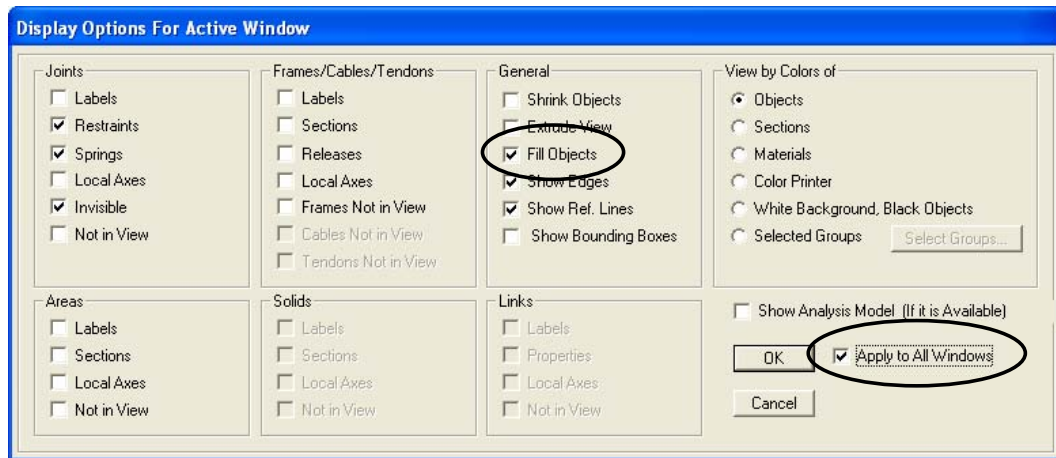


Figure 22
Display Options for Active Window form

- G. Click **OK** to accept the changes, and the model now appears as shown in Figure 23.

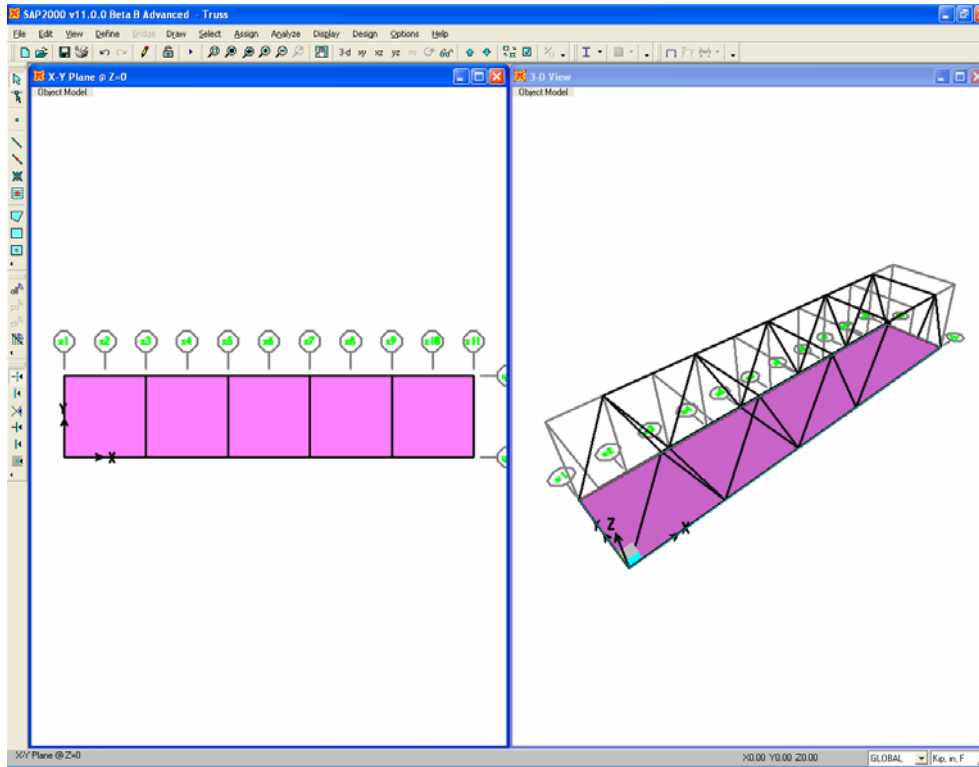


Figure 23
Model after the area object has been drawn

Mesh the Area Object

Make sure that the X-Y Plane @ Z=0 view is still active. The area object will now be meshed similar to the previous meshing of frame objects.

- A. **Right** click anywhere on the area object to display the Object Model – Area Information form as shown in Figure 24.
- B. Click on the Assignments tab on the Object Model – Area Information form.

Figure 24
Object Model –
Area Information
form

Section Property	
Section Name	DECK
Section Type	Shell (Shell-Thin)
Property Modifiers	None
Material Overwrite	None
Thickness Overwrite	None
Joint Offset Overwrite	None
Local Axes	Default
Area Springs	None
Area Mass	None
Automatic Area Mesh	None
Auto Edge Constraint	No
Material Temp	Default
Reference Temp	Default
Group	ALL
Plot Functions	None

- C. Double click in the edit box opposite the Automatic Area Mesh item to display the Assign Automatic Area Mesh form as shown in Figure 25.
- D. Select the *Mesh Area Based On Points On Area Edges* option.
- E. Check the *Intersections of Straight Line Objects In Meshing Group With Area Edges* check box.

The area object representing the deck was drawn as a single object, but needs to be meshed into multiple analysis elements so that there will be connectivity between the deck and the intermediate points along the chord elements. Meshing, unlike dividing, does not create new objects. If the **Edit menu > Edit Areas > Divide Areas** command were to be used, new objects would be created.

- F. Click the **OK** button and then the **OK** button on the Object Model – Area Information form to complete the area object meshing.

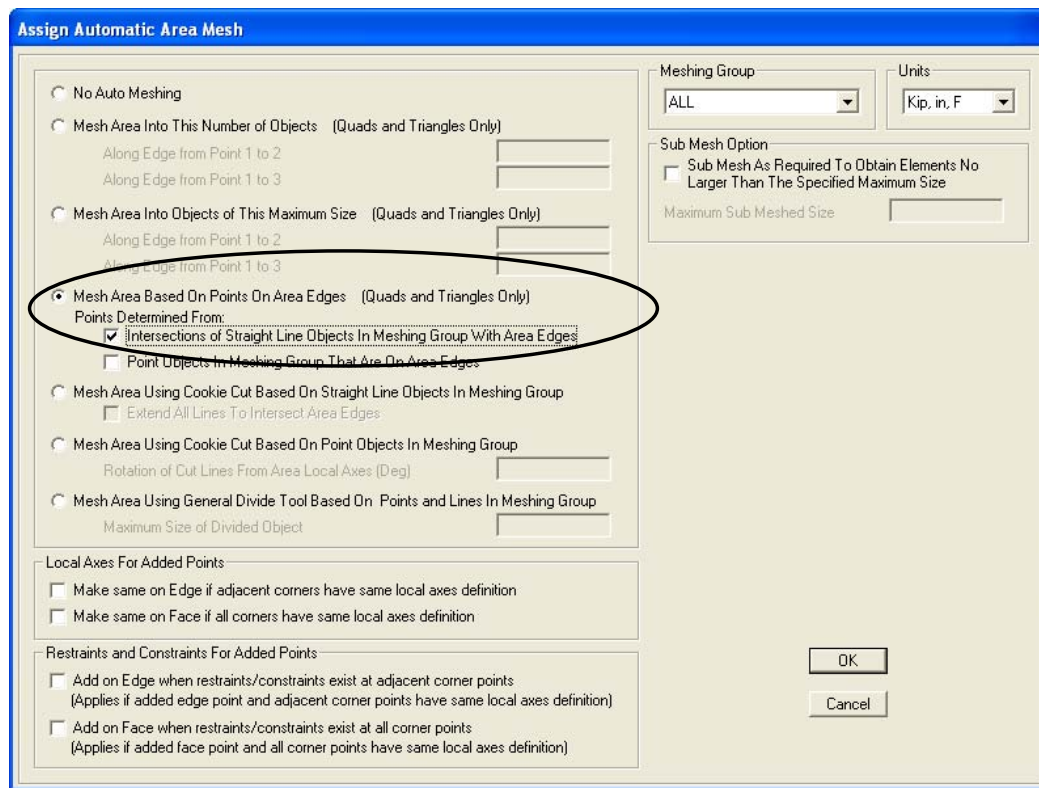


Figure 25
Assign Automatic Area Mesh form

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 access the Joint Restraints form as shown in Figure 26.
- C. Click on the **Roller**  button to assign restraints in the Translation 3 direction for these two joints. Click **OK** to accept the assignment.

Figure 26
Joint
Restrains
form



- 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 access the Joint Restraints form.
- F. Click on the **Pinned**  button to assign restraints in the Translation 1, 2, and 3 directions for these two joints. Click **OK** to accept the assignment.
- G. 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 > Load Cases** command to access the Define Loads form shown in Figure 27. Note there is only a single default load case defined, which is a dead load case with self-weight (DEAD).

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

Figure 27
Define Loads form

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.

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 drop-down list; 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 28. Click the **OK** button in that form to accept the newly defined static load case.

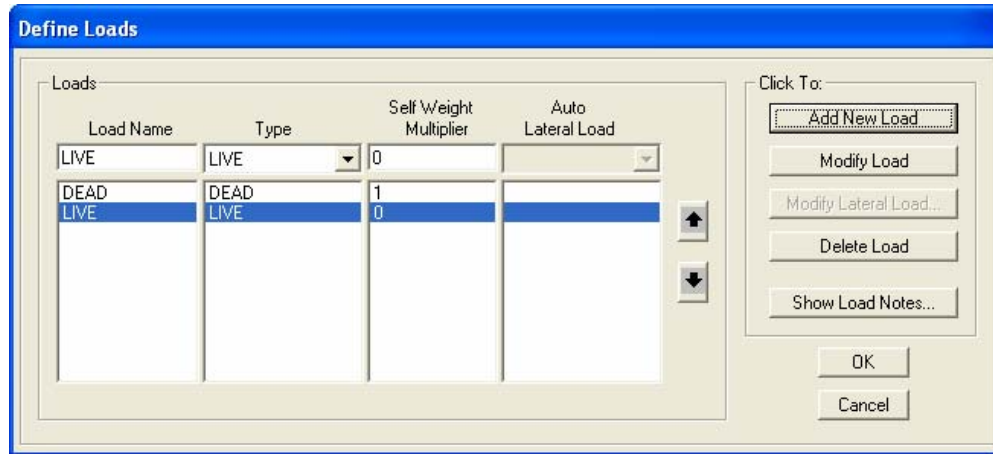


Figure 28
The Define Loads form after the live load case has been defined

- C. Click the **Assign menu > Clear Display of Assigns** command to remove the Joint Restraints identifiers.

Step 6 Assign Gravity Loads

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


- A. Click anywhere on the area object to select the deck. The status bar in the lower left-hand corner should show “1 Areas, 4 Edges Selected.” If you make a mistake in selecting, click the **Clear Selection**  button, and try again.
- B. Select the **Assign menu > Area Loads > Uniform (Shell)** command to access the Area Uniform Loads form. Select *DEAD* from the Load Case Name drop-down list as shown in Figure 29. Clicking on the + button will display the Define Loads form where load cases may be altered or added.

Figure 29
Area Uniform
Loads form


1. Select *lb-ft* from the Units drop-down list.
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. Right click anywhere on the area object to display the Object Model – Area Information form and select the Loads tab.
- D. Double click in the edit box opposite the Force/Area item to display the Area Uniform Loads form. Select *LIVE* from the Load Case Name drop-down list.
1. Select *lb-ft* from the Units drop-down list.
 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 **OK** button on the Object Model - Area Information form.
- F. Click the **Assign menu > Clear Display of Assigns** command to clear the display of the assigned loads.

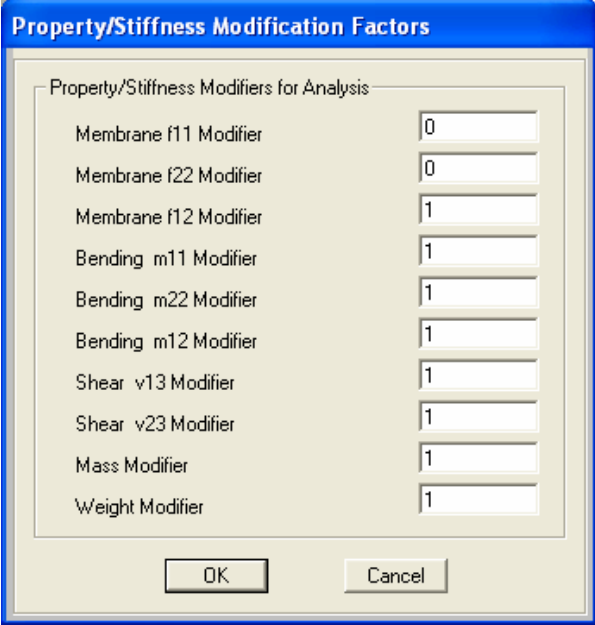
Step 7 Assign Area Stiffness Modifiers

In this Step, the membrane properties of the Area object is 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. Click anywhere on the area object to select the deck, or click **Select menu > Get Previous Selection** command, or click the **Get Previous Selection**  button. These actions select the deck object.
- B. Click the **Assign menu > Area > Area Stiffness Modifiers** command to access the Property/Stiffness Modification Factors form shown in Figure 30.
 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 the assignment.



Property/Stiffness Modifiers for Analysis	
Membrane f11 Modifier	0
Membrane f22 Modifier	0
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

Figure 30
*Property/Stiffness
Modification
Factors form*

- 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 toggle the grid lines off 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 model will be viewed and the analysis will be run.

- A. Click the **Set Display Options**  button. When the form appears, check the *Show Analysis Model* check box as shown in Figure 31.

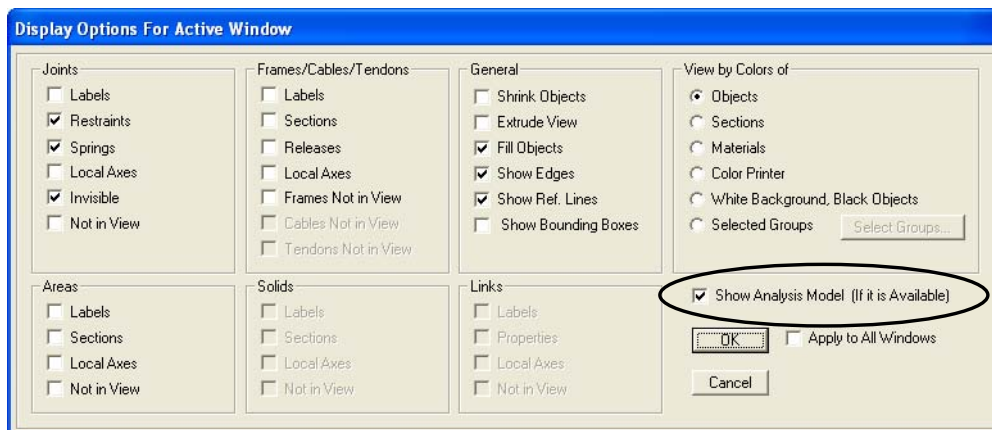


Figure 31
Display Options for Active Window form

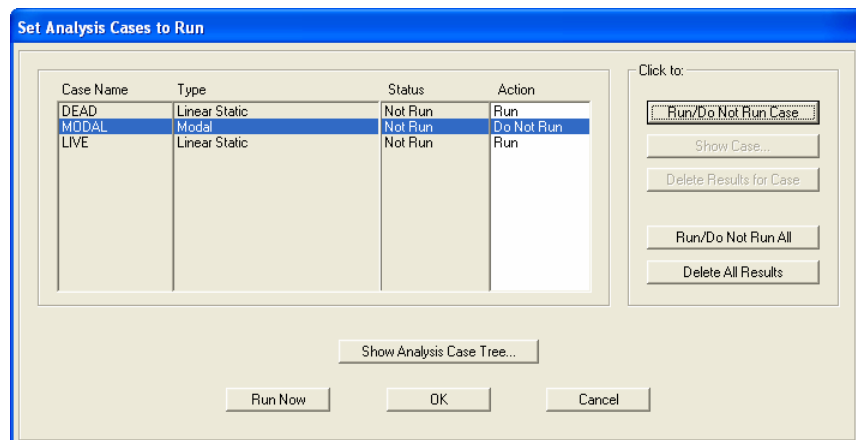
- B. Click the **OK** button to accept the display setting.
- C. If the model has not previously been analyzed, a message similar to the one in Figure 32 displays.

Figure 32
Analysis Model
message



- D. Click the **Yes** button to display the analysis model. Note that under the 3-D View display title bar that the model status has changed to Analysis Model. Take a moment to verify that the element formation is as expected.
- E. Click the **Analyze menu > Run Analysis** command or the **Run Analysis** button, to access the Set Analysis Cases to Run form as shown in Figure 33.

Figure 33
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 list.
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 analysis 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.

- F. 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.



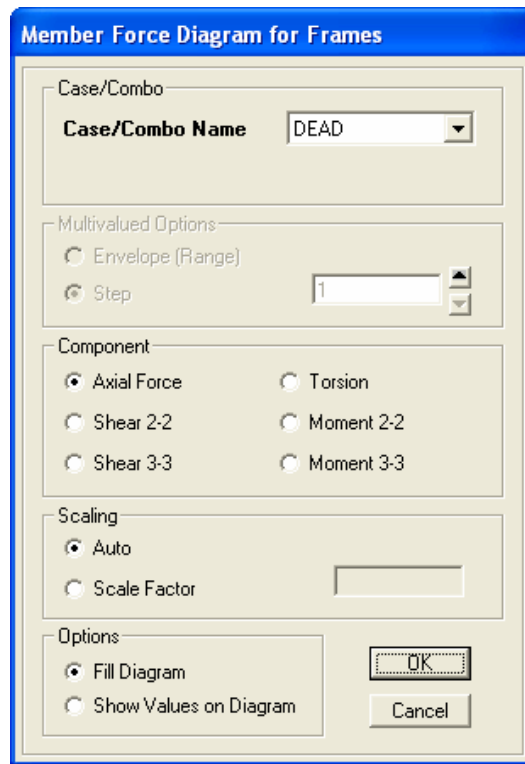
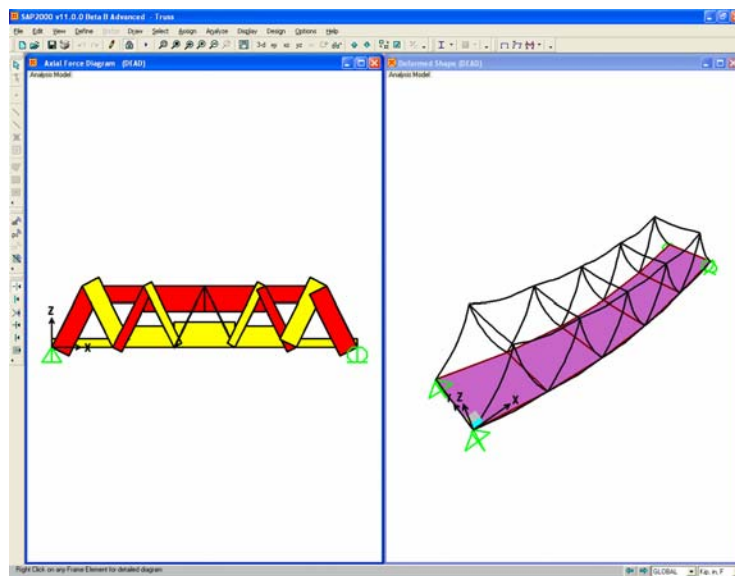
- A. Make sure that the X-Y Plane @ Z=0 view is active. Then click on the **XZ View**  button to reset the view to an elevation.
- B. Click the **Show Forces/Stresses > Frames/Cables** button, , or the **Display menu > Show Forces/Stresses > Frames/Cables** command to bring up the Member Force Diagram for Frames form shown in Figure 34.
 1. Select *DEAD* from the Case / Combo Name drop-down list.
 2. Select the *Axial Force* option.
 3. Select the *Auto* option under the Scaling area.
 4. Select the *Fill Diagram* option.

Figure 34
Member Force Diagram for Frames form



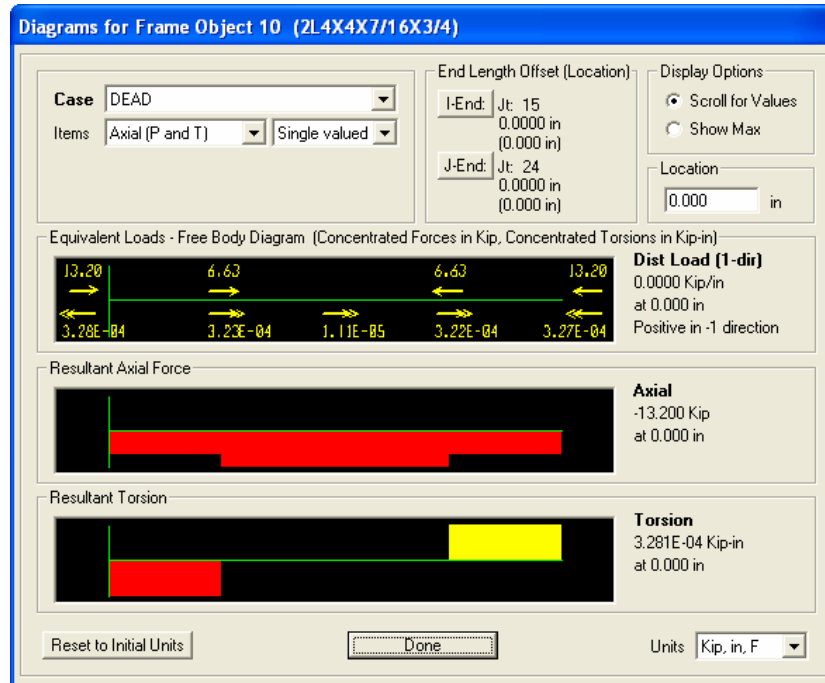
5. Click the **OK** button to generate the axial force diagram shown in Figure 35.

Figure 35
Axial force diagram in an elevation view




- C. Right click on the top chord member in the X-Z view to access the Diagram for Frame Object form shown in Figure 36.

Figure 36
Force details
obtained by
right-clicking
top chord of
truss in the
elevation view
in Figure 35



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 meshed the frame object into smaller elements for analysis.

1. Click the *Scroll for Values* option and you may obtain the values at any location by moving the mouse over the diagrams with the left button held down.
 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 access the Deformed Shape form shown in Figure 37.

1. Select *LIVE* from the Case/Combo Name drop-down list.
2. Check the *Cubic Curve* check box.
3. Click the **OK** button to generate the deformed shape shown in Figure 38.

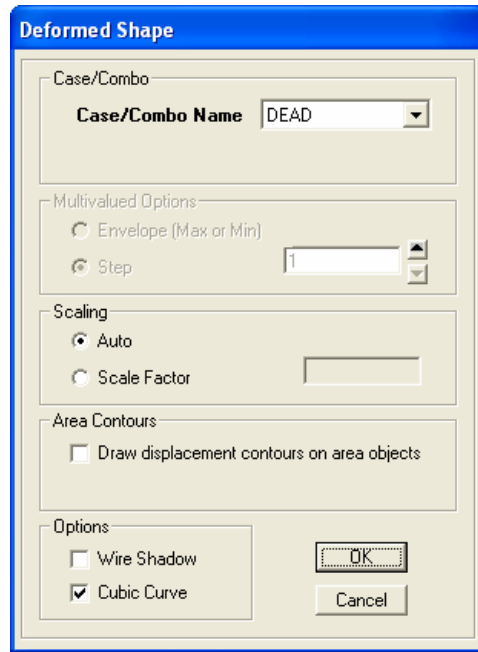
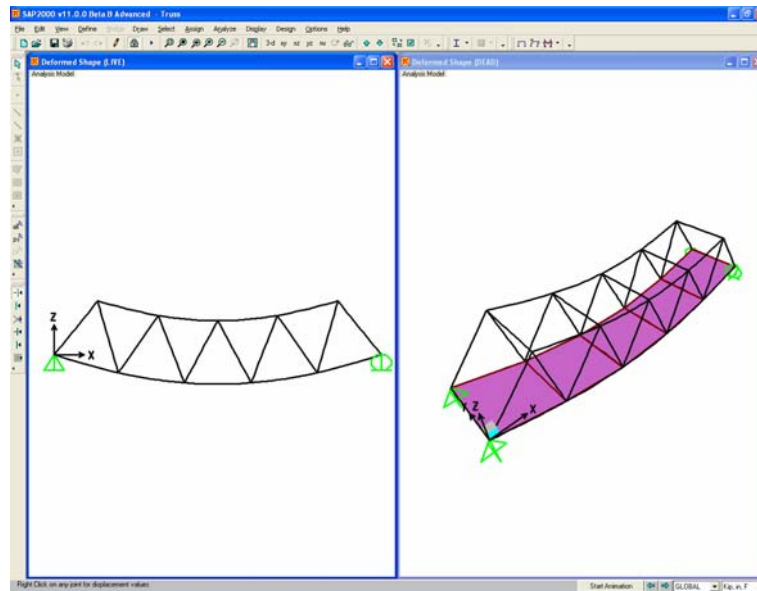


Figure 37
Deformed Shape form

Figure 38
Deformed Shape in an elevation view



- E. Right click on the middle joint on the top chord object in Figure 38 to display the Joint Displacements results form shown in Figure 39.

Figure 39
Joint Displacements obtained by right-clicking a joint shown in the elevation view in Figure 38


Joint Object 20		Joint Element 20	
	1	2	3
Trans	0.02697	1.575E-06	-0.13156
Rotn	1.059E-05	0.00000	0.00000

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

- F. Close the Joint Displacements form by clicking the X in the upper right hand corner of the form, or by clicking anywhere other than on the 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 40 displays.
 1. Click in the Design Code Values drop-down list to see the available design codes. Select the *AISC-LRFD99* code.
 2. Review the information contained in the other items 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.

Item	Value
1 Design Code	AISC-LRFD99
2 Time History Design	envelopes
3 Framing Type	DMF
4 Seismic Design Category	D
5 Phi (Bending)	0.9
6 Phi (Compression)	0.85
7 Phi (Tension-Yielding)	0.9
8 Phi (Tension-Fracture)	0.75
9 Phi (Shear)	0.9
10 Phi (Shear-Torsion)	0.75
11 Phi (Compression, Angle)	0.9
12 Ignore Seismic Code?	No
13 Ignore Special Seismic Load?	No
14 Is Doubler Plate Plug-Welded?	Yes
15 Consider Deflection?	No
16 DL Limit, L /	120.
17 Super DL+LL Limit, L /	120.
18 Live Load Limit, L /	360.
19 Total Limit, L /	240.
20 Total-Camber Limit, L /	240.
21 Pattern Live Load Factor	0.75
22 Stress Ratio Limit	0.95
23 Max Number of Auto Iterations	1

Item Description
Design Code:
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

Set To Default Values: All Items, Selected Items
Reset To Previous Values: All Items, Selected Items
OK, Cancel

Figure 40
Steel Frame Design Preferences form

When the design is complete, the selected sizes are displayed on the model. The model appears as shown in Figure 41.

- C. Click the **Design menu > Steel Frame Design > Verify Analysis vs Design Section** command. A message similar to the one in Figure 42 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 42 indicates that the analysis and design sections are different.

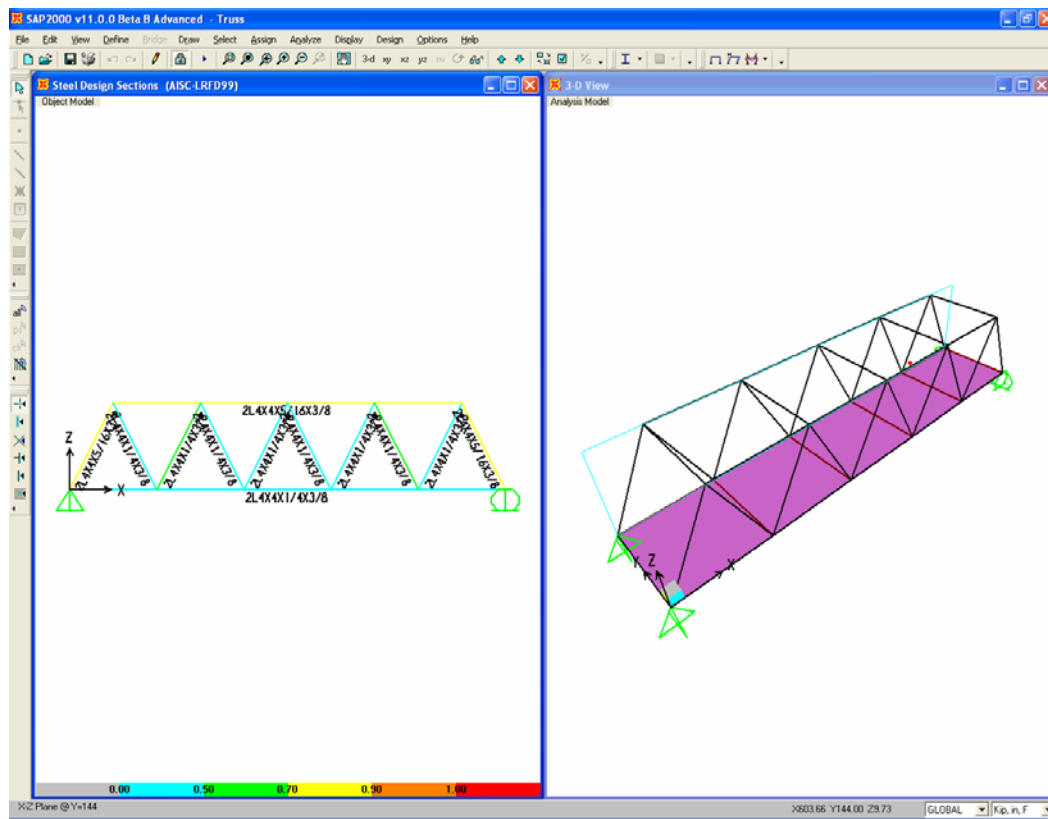
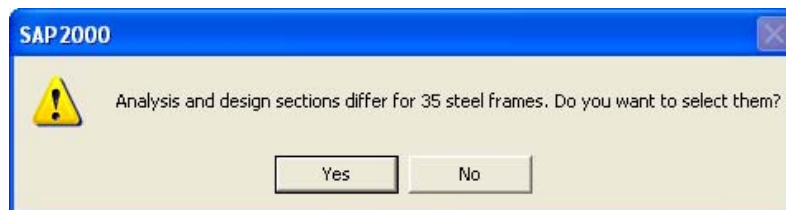


Figure 41
Model after the initial steel frame design

Figure 42
Analysis vs Design
Section warning
message



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 members in the X-Z view (shown in Figure 41). The Steel Stress Check Information form shown in Figure 43 displays. Note that the reported analysis and design sections are different.

Steel Stress Check Information (AISC-LRFD99)

Frame ID: 10
 Design Code: AISC-LRFD99
 Analysis Section: 2L4x4x7/16x3/4
 Design Section: 2L4x4x5/16x3/8

COMBO ID	STATION LOC	----MOMENT RATIO	INTERACTION CHECK	MAJ-SHR RATIO	MIN-SHR RATIO
DSTL2	24.00	0.567 (C)	= 0.539 + 0.027 + 0.000	0.002	0.000
DSTL2	48.00	0.581 (C)	= 0.539 + 0.041 + 0.000	0.000	0.000
DSTL2	72.00	0.582 (C)	= 0.539 + 0.043 + 0.000	0.000	0.000
DSTL2	96.00	0.571 (C)	= 0.539 + 0.031 + 0.000	0.002	0.000
DSTL2	120.00	0.547 (C)	= 0.539 + 0.007 + 0.000	0.003	0.000
DSTL2	144.00	0.585 (C)	= 0.539 + 0.045 + 0.000	0.004	0.000
DSTL2	144.00	0.876 (C)	= 0.810 + 0.066 + 0.000	0.004	0.000

Modify/Show Overwrites: Overwrites
 Display Details for Selected Item: Details
 Display Complete Details: Tabular Data
 Stylesheet: Default
 Table Format File

OK Cancel

Figure 43
 Steel Stress Check Information form

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.

Click the **Details** button on the Steel Stress Check Information form. The Steel Stress Check Data AISC-LRFD99 form shown in Figure 44 displays. Use the File menu on the form to print the data.

Figure 44
Steel Stress
Check
Data
AISC-LRFD99
form

Steel Stress Check Data AISC-LRFD99

AISC-LRFD99 STEEL SECTION CHECK
 Combo : D5112
 Units : Kip, in, F

Frame : 10 Design Sect: 2L4X4X5/16X3/8
 X Mid : 360.000 Design Type: Beam
 Y Mid : 144.000 Frame Type: Ordinary Moment Frame
 Z Mid : 144.000 Sect Class : Slender
 Length : 576.000 Major Axis : 0.000 degrees counterclockwise from local 3
 Loc : 144.000 RLF : 1.000

Area : 4.800 SHmajor : 2.543 rHmajor : 1.237 AUHmajor : 2.500
 IHmajor : 7.350 SHminor : 0.720 rHminor : 1.802 AUHminor : 2.500
 IHminor : 15.579 2Hmajor : 4.520 E : 29000.000
 Ixy : 0.000 2Hminor : 0.255 Fy : 36.000

STRESS CHECK FORCES & MOMENTS

Location	Pu	Mu23	Mu22	Uu2	Uu3	Tu
144.000	-58.271	-0.059	-0.014	-0.172	-1.410E-04	6.980E-05

PHL DEMAND/CAPACITY RATIO

Governing Equation (H1-1a)	Total Ratio	P Ratio	MHmajor Ratio	MHminor Ratio	Ratio Limit	Status Check
	0.076	0.010	0.066	0.000	0.950	OK

AXIAL FORCE DESIGN

	Pu Force	phi*Pnc Capacity	phi*Pnt Capacity
Axial	-58.271	71.937	155.520

MOMENT DESIGN

	Mu	phi*Mn Capacity	Cn	01 Factor	02 Factor	K Factor	L Factor	Cb
Major Moment	-6.109	82.401	0.850	1.997	1.000	1.000	0.250	1.523
Minor Moment	-0.016	120.542	0.050	1.166	1.000	1.000	0.250	

SHEAR DESIGN

	Uu Force	phi*Vn Capacity	Stress Ratio	Status Check	Tu Torsion
Major Shear	0.172	40.600	0.004	OK	0.000
Minor Shear	1.410E-04	40.600	2.917E-06	OK	0.000

Click the **X** in the upper right-hand corner of the Steel Stress Check Data AISC-LRFD99 form to close it. Click the **Cancel** Button to close the Steel Stress Check Information form.



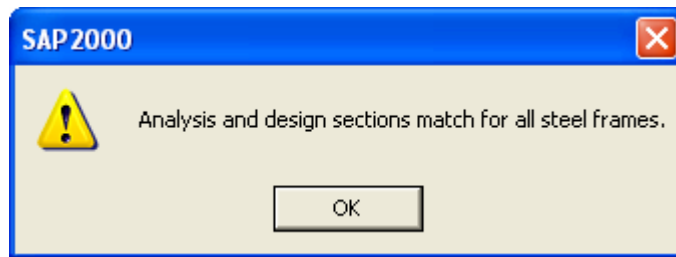
- E. To rerun the analysis with the new analysis sections, click the **Analyze menu > Run Analysis** command or the **Run Analysis**  button, and then click the **Run Now** button on the Set Analysis Cases to Run form.
- F. When the analysis is complete, click the **OK** button to close the analysis 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 45 displays.

Figure 45
*Analysis vs Design
Section message*



The message in Figure 45 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 46 should appear indicating that all members passed.

Figure 46
*Stress/capacity
check message*



Note that members not passing at this stage is an indication of inadequate sections in the auto select section list. The program would have used the largest section in the auto select section 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 **OK** button to close the form.

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

This introductory tutorial for SAP2000 Version 11 is now complete.