



Introductory Tutorial for SAP2000[®]

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

Copyright

Copyright © Computers and Structures, Inc., 1978-2011.
All rights reserved.

The CSI Logo® and SAP2000® are registered trademarks of Computers and Structures, Inc. Watch & Learn™ is a trademark of Computers and Structures, Inc. Windows is a registered trademark of Microsoft Corporation. Adobe and Acrobat are registered trademarks of Adobe Systems Incorporated.

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

No part of this publication may be reproduced or distributed in any form or by any means, or stored in a database or retrieval system, without the prior explicit written permission of the publisher.

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

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

Phone: (510) 649-2200
FAX: (510) 649-2299
e-mail: info@csiberkeley.com (for general questions)
e-mail: support@csiberkeley.com (for technical support questions)
web: www.csiberkeley.com

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND TESTING OF THIS SOFTWARE. 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 THIS PRODUCT.

THIS PRODUCT IS A PRACTICAL AND POWERFUL TOOL FOR STRUCTURAL DESIGN. HOWEVER, THE USER MUST EXPLICITLY UNDERSTAND THE BASIC ASSUMPTIONS OF THE SOFTWARE MODELING, ANALYSIS, AND DESIGN ALGORITHMS AND COMPENSATE FOR THE ASPECTS THAT ARE NOT ADDRESSED.

THE INFORMATION PRODUCED BY THE SOFTWARE MUST BE CHECKED BY A QUALIFIED AND EXPERIENCED ENGINEER. THE ENGINEER MUST INDEPENDENTLY VERIFY THE RESULTS AND TAKE PROFESSIONAL RESPONSIBILITY FOR THE INFORMATION THAT IS USED.

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 Frame Sections	2-8
	Define an Auto Select Section List	2-11
Step 2	Add Frame Objects	2-13
	Draw Frame Objects	2-13
	Assign Member End Releases	2-16
	Divide Frame Objects	2-17
	Save the Model	2-18

Step 3	Add Restraints	2-19
Step 4	Replicate Frame Objects	2-20
	Draw Additional Frame Objects	2-21
Step 5	Add Area Objects	2-23
	Define the Area Sections	2-23
	Draw the Area Object	2-24
	Mesh the Area Object	2-26
	Replicate Area Objects	2-28
Step 6	Move Objects	2-29
Step 7	Assign Area Stiffness Modifiers	2-33
Step 8	Define Load Patterns	2-35
Step 9	Assign Loads	2-37
Step 10	Define Load Combinations	2-41
Step 11	Run the Analysis	2-43
Step 12	Graphically Review the Analysis Results	2-46
Step 13	Design the Steel Frame Objects	2-50

Chapter 1

Introduction

Using this Manual

This manual introduces you to SAP2000 Version 15. 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.

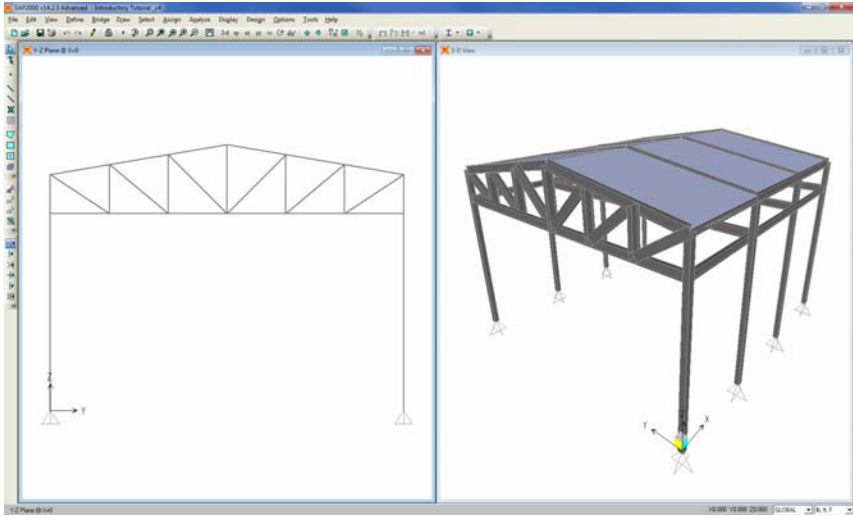
Welcome to SAP2000.

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 1-bay by 3-bay industrial building. The open bay has a dimension of 36 feet, while the three bays in the longitudinal direction are each 12 feet wide. The concrete roof is supported by trusses in the transverse direction, which frame into tube columns pinned at the base.

The trusses are to be constructed of W shapes, while the roof will be a concrete slab 5 inches thick. The columns are HSS 8x8x3/8 sections. The building will be analyzed for static loads only, and the trusses will be loaded at their panel points with a Dead Load = 20 kips, while the roof will be loaded with a Live Load = 30 pounds per square foot (psf) and a Wind Load = 15 psf on the windward side and 6 psf on the leeward side.

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. In addition to these menus, 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.

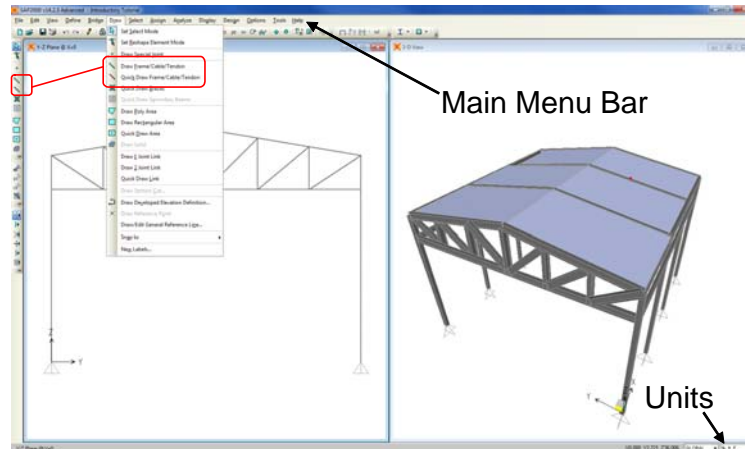

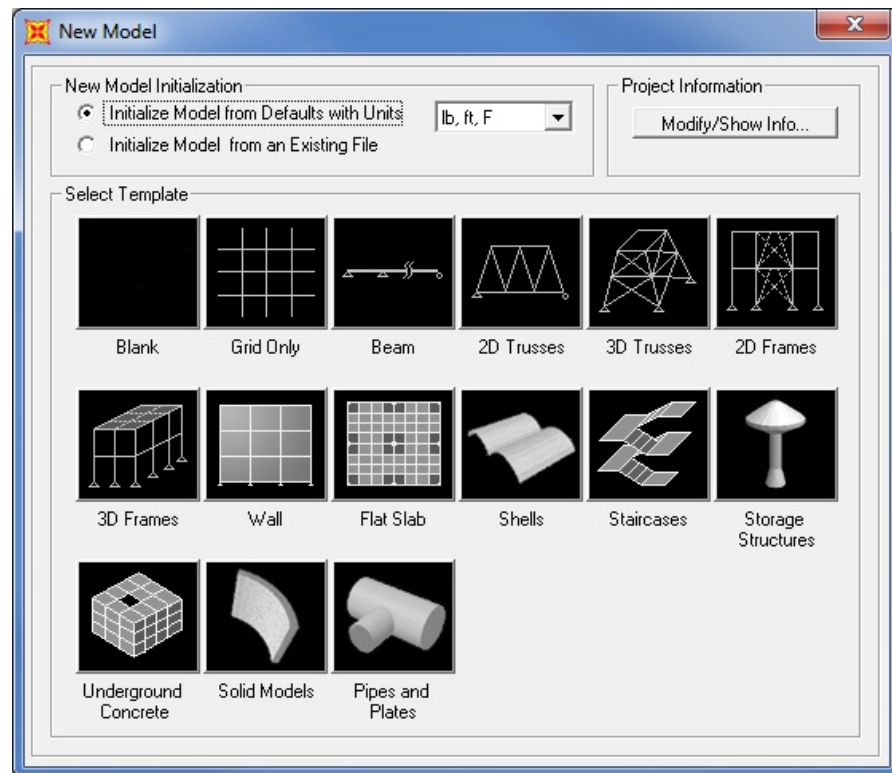


Figure 2
The Interface

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 sections will be selected, including a list of W sections 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 lb-ft.



*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 spacing 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 image shows a dialog box titled "Quick Grid Lines" with two tabs: "Cartesian" and "Cylindrical". The "Cartesian" tab is selected. The dialog contains the following fields and values:

- Coordinate System Name: GLOBAL
- Number of Grid Lines:
 - X direction: 2
 - Y direction: 7
 - Z direction: 3
- Grid Spacing:
 - X direction: 36
 - Y direction: 6
 - Z direction: 20
- First Grid Line Location:
 - X direction: 0.
 - Y direction: 0.
 - Z direction: 0.

At the bottom of the dialog are "OK" and "Cancel" buttons.

- 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 2 for the X direction, to 7 for the Y direction, and to 3 for the Z direction. Type **432 in** (including the in) into the X direction spacing edit box and press the Enter key on your keyboard. Note that the program automatically converts the 432 in to 36 to be consistent with the default units of feet. Enter **6** for the Y direction spacing, and **20** for the 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.

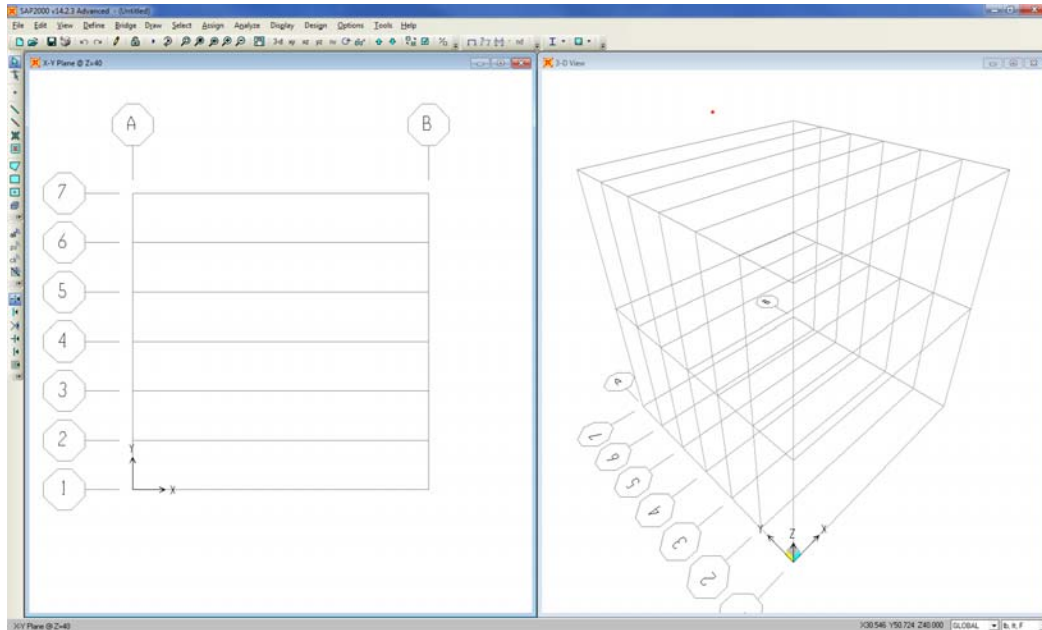


Figure 5
The SAP2000 windows

- 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 drop-down arrow located in the top right corner of each window.

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. If you are unable to see the grid bubbles in this view, click the **View menu > Zoom Out One Step** command until they become visible.

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. Click the **Define menu > Coordinate Systems/Grids** command to display the Coordinate/Grid Systems form. In the Systems area, highlight

GLOBAL and then click the **Modify/Show System** button to display the Define Grid System Data form shown in Figure 6.

Figure 6
Define
Grid
System
Data
form

Define Grid System Data

System Name: GLOBAL Units: lb, ft, F

Grid Lines: Quick Start...

X Grid Data

	Grid ID	Ordinate	Line Type	Visibility	Bubble Loc.	Grid Color
1	A	0.	Primary	Show	End	
2	B	36.	Primary	Show	End	
3						
4						
5						
6						
7						
8						

Y Grid Data

	Grid ID	Ordinate	Line Type	Visibility	Bubble Loc.	Grid Color
1	1	0.	Primary	Show	Start	
2	2	6.	Primary	Show	Start	
3	3	12.	Primary	Show	Start	
4	4	18.	Primary	Show	Start	
5	5	24.	Primary	Show	Start	
6	6	30.	Primary	Show	Start	
7	7	36.	Primary	Show	Start	
8						

Z Grid Data

	Grid ID	Ordinate	Line Type	Visibility	Bubble Loc.	Grid Color
1	Z1	0.	Primary	Show	End	
2	Z2	20.	Primary	Show	End	
3	Z3	24	Primary	Show	End	
4						
5						
6						
7						
8						

Display Grids as: Ordinates Spacing

Hide All Grid Lines
 Glue to Grid Lines

Bubble Size: 5

Reset to Default Color
Reorder Ordinates
OK Cancel

- F. The Define Grid System Data form is used to modify the grid layout. Under the Z Grid Data area, type **24** into the Ordinate edit box for Grid ID Z3.
- G. Click the **OK** button twice and a modified grid will now be displayed.

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 HSS columns. 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 7.

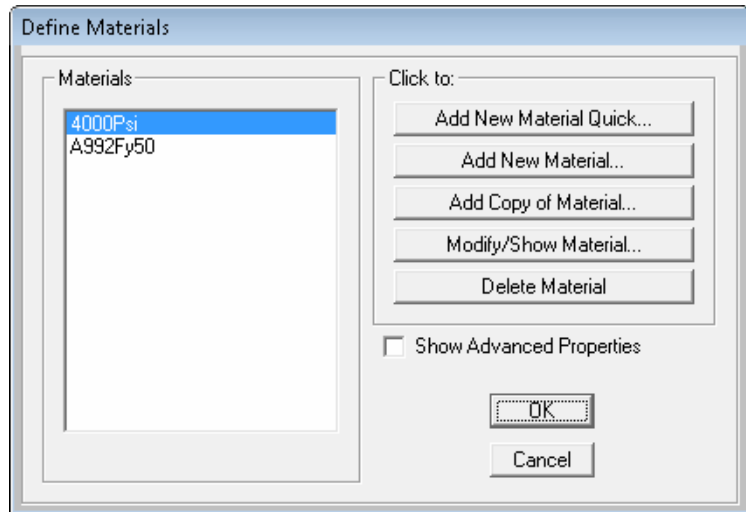


Figure 7
Define
Materials form

- B. Click the **Add New Material Quick** button to display the Quick Material Definition form shown in Figure 8.
- 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.

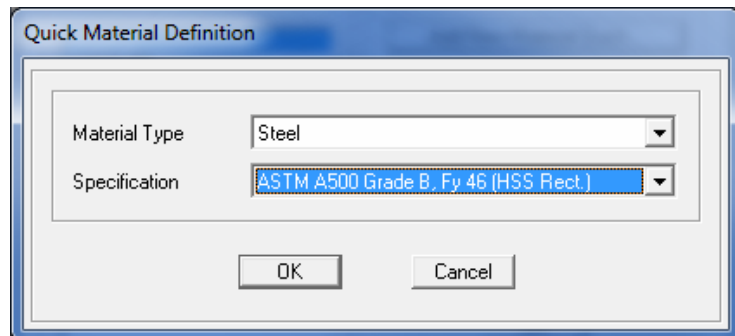


Figure 8
Quick Material
Definition
form

- D. Select *ASTM A500 Grade B, Fy 46 (HSS Rect.)* 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 Frame Sections

A frame section will be defined for the HSS columns. Frame sections must be defined before they can be assigned to frame objects in the model.

- A. Click the **Define menu > Section Properties > Frame Sections** command, which will display the Frame Properties form shown in Figure 9.

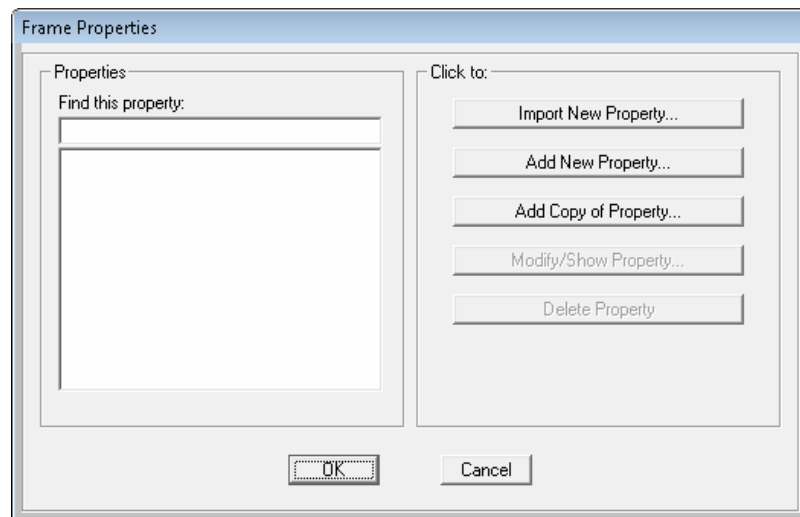
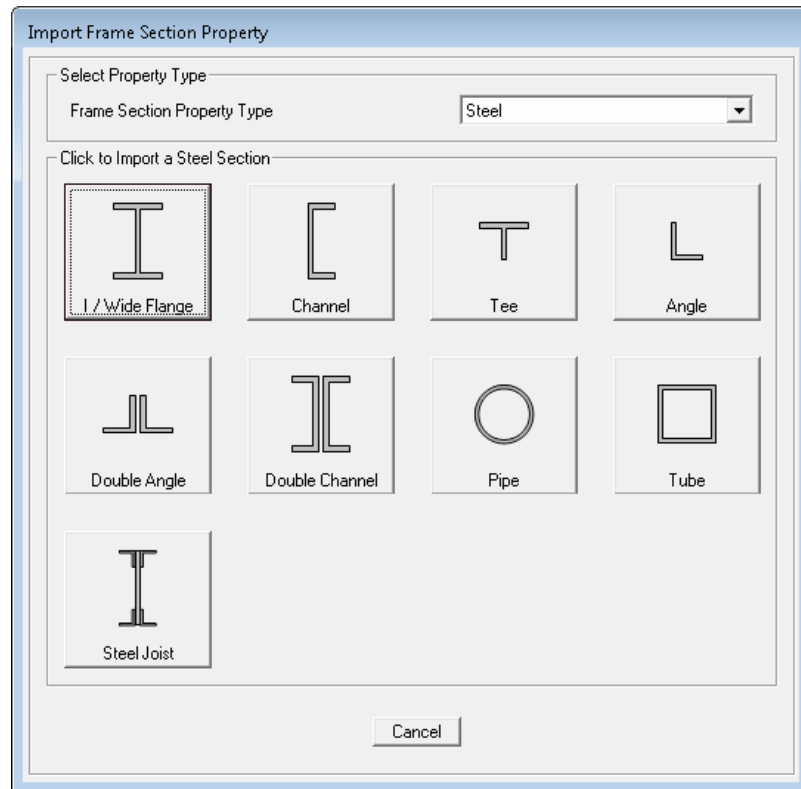


Figure 9
Frame
Properties
form

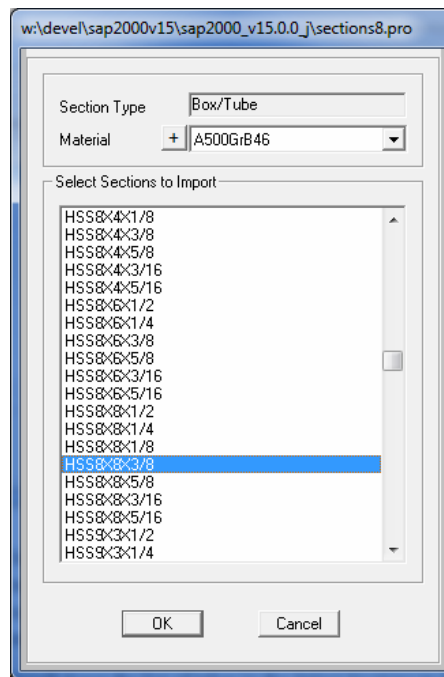
- B. Click the **Import New Property** button, which will display the Import Frame Section Property form shown in Figure 10.

Figure 10
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 **T**ube 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 HSS section 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 **O**pen button. The Sections8.pro sections list form shown in Figure 11 displays.

Figure 11
Sections8.pro
sections
list



- E. Select *A500GrB46* 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 tubes in the Select Sections to Import area until you find *HSS8X8X3/8*. Click once on that member to highlight it.
- G. Click the **OK** button, and then click the **OK** button in the Box/Tube Section form to add the tube selected to the list in the Properties area on the Frame Properties form.
- H. Do not close the Frame Properties form - it will be required in the next segment of the tutorial.

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 the trusses from a set of W shapes (W8X10 thru W8X67), which will be chosen from an auto select sections list created now.

- A. With the Frame Properties form displayed, click the **Import New Property** button to display the Import Frame Section Property form.
- B. Verify that *Steel* is still showing in the Frame Section Property Type drop-down list, and then click the **I/Wide Flange** button, which will display the Sections8.pro sections list shown in Figure 12.

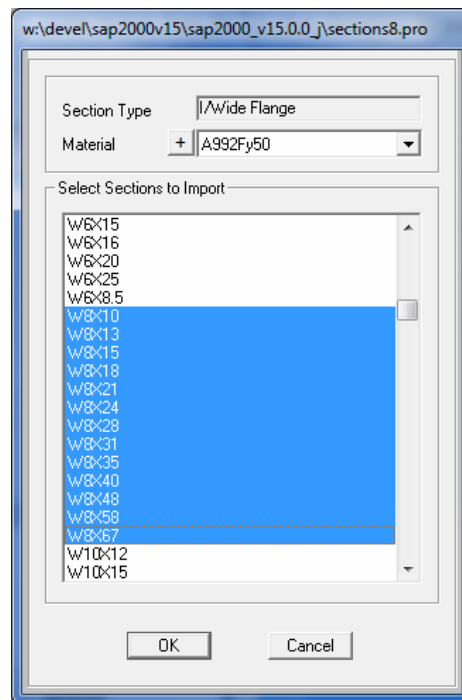
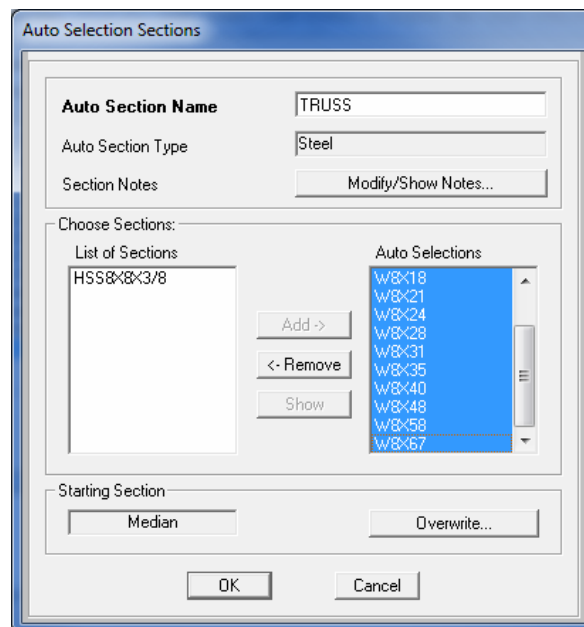


Figure 12
Sections8.pro
sections
list

- C. Select *A992Fy50* from the Material drop-down list – this is the default steel material property.
- D. Scroll down the list of shapes in the Select Sections to Import area until you find *W8X10*. Click once on that member to highlight it.
- E. Scroll further down the list until you find *W8X67*. Hold down the Shift key on your keyboard and click once on *W8X67*; a collection of W8s should now be highlighted.
- F. Click the **OK** button, and then click the **OK** button in the I/Wide Flange Section form to add the shapes selected to the list in the Properties area on the Frame Properties form.
- G. 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.
- H. In the Frame Section Property Type drop-down list, select *Steel*.
- I. Click the **Auto Select List** button to display the Auto Selection Sections form shown in Figure 13.

Figure 13
Auto
Selection
Sections
form




- J. Type **TRUSS** in the Auto Section Name edit box.
- K. Locate the *W8X10* shape under the List of Sections, and click once to highlight it.
- L. Continue down the list until you find the last W shape, *W8X67*, and while holding down the Shift key on the keyboard, click once on this section. All of the W shapes should now be highlighted.
- M. Click the **Add** button to move the selected list to the Auto Selections edit box on the right side of the form.
- N. 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 sections previously defined 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=40 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 *Y-Z plane* option.
 - 2. Type **0** into the X = edit box to display an elevation view, and click **OK**.


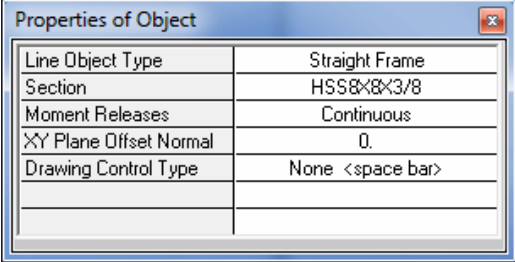
- B. Click the **Draw Frame/Cable**  button or use the **Draw menu > Draw Frame/Cable/Tendon** command. If you accessed the **Draw Frame/Cable/Tendon** command via the **Draw menu**, the **Draw Frame/Cable** button will depress verifying your command selection. The Properties of Object pop-up form for frames will appear as shown in Figure 14.


Figure 14
Properties of Object form



Properties of Object	
Line Object Type	Straight Frame
Section	HSS8X8X3/8
Moment Releases	Continuous
XY 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 *HSS8X8X3/8*. Single click on it to assign this section to the members you will draw.
- D. To draw the first frame object, left click once in the Y-Z Plane view at the Y-Z origin (lower left corner of the grid), and then click again along the vertical grid at the Grid Point that gives $L=24.00$ ($y=0, z=24$). The cursor location is indicated in the lower right-hand corner of the interface. A frame line should appear in both views (elevation and 3D). After clicking to define the end point of the frame object, right click to “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 **Set Select Mode**  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. Draw another column by clicking on the other side of the grid at Grid Point (y=36, z=0) to start, and then click again at Grid Point (y=36, z=24) to draw a vertical member with a L=24.00.
- F. Click the right mouse button to "lift" the pen.
- G. Return to the Properties of Object form and select *TRUSS* from the Section drop-down list. This will assign the auto select list TRUSS to the next members you will draw.
- H. In the Y-Z Plane @ X=0 view, a bottom chord will be drawn. Click at Grid Point (y=0, z=20) to start, and then click again at Grid Point (y=36, z=20) to draw a horizontal member with L=36.00. Click the right mouse button to lift the pen.

Note that the bottom chord was drawn as a single "physical" object, similar to the way that it would typically be constructed. No regard was paid to where the other truss members frame into it. In order to maintain connectivity with the other truss members, i.e., diagonals and verticals, SAP2000 will automatically mesh this chord object into multiple internal elements when performing the analysis. However, when reporting results and doing design, SAP2000 will treat the chord as a single "physical" member.

- I. Next draw a top chord. Click at Grid Point (y=0, z=24) to start, and then click again at grid Point (y=36, z=24).
- J. With the draw mode still active, click on the lower chord one grid line over at Grid Point (y=30, z=20) to draw the first diagonal member. Draw additional verticals and diagonals by clicking at the following Grid Points; (y=30, z=24); (y=24, z=20); (y=24, z=24); (y=18, z=20); and (y=18, z=24).
- K. Click the right mouse button to lift the pen.
- L. Draw the next series of diagonals and verticals by clicking at Grid Point (y=18, z=20) to start, and then clicking at the following Grid Points: (y=12, z=24); (y=12, z=20); (y=6, z=24); (y=6, z=20); and (y=0, z=24).

- M. Click the **Set Select Mode**  button, or press the Esc key on the keyboard to exit the **Draw Frame/Cable/Tendon** command.

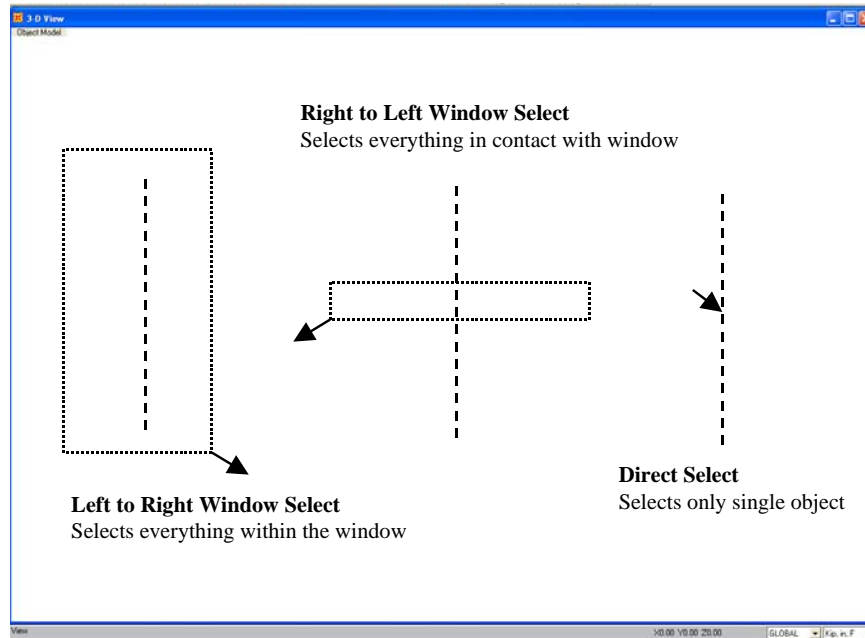
Assign Member End Releases

Make sure that the program is in the Select mode, and that the Y-Z view is active.

- A. Select the internal diagonal and vertical members spanning between the longitudinal chords by left clicking directly on the members, or left clicking to the right of the objects, holding the left mouse button down, and dragging the mouse across the members. See Figure 15 for selection options.

The status bar in the lower left of the program window should show *11 Frames Selected*.

*Figure 15
Graphical
Selection
Options*



- B. Click the **Assign menu > Frame > Releases/Partial Fixity** command to access the form shown in Figure 16. Check the *Moment 33 (Major)* check boxes for both the *Start* and *End* Releases.

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

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

*Figure 16
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 Units: lb, ft, F

OK Cancel

- D. Click the **Assign menu > Clear Display of Assigns** command to remove the Frame Releases identifiers.

Divide Frame Objects

SAP2000 allows members to be sub-divided into multiple objects after they are drawn to accommodate changes in geometry (this differs from the internal meshing done during analysis where the number of objects remains the same). In this tutorial, the top chord of the truss was drawn as a single object, but will be divided in two at midspan to allow for the sloped roof.

Make sure that the program is in the Select mode, and that the Y-Z view is active.


- A. Select the top chord of the truss (the horizontal member at elevation $z=24$) by left clicking directly on the member.
- B. Click the **Edit menu > Edit Lines > Divide Frames** command to access the form shown in Figure 17.

Figure 17
Divide
Selected
Frames
form

- C. Select the *Divide into Specified Number of Frames* option in the Divide Selected Straight Frame Objects area.
- D. Type **2** into the Number of Frames edit box.
- E. Type **1** into the Last/First Length Ratio edit box.
- F. Click the **OK** button to accept the changes and return to the Select mode.

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 *Tutorial Building*.

Step 3 Add Restraints

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


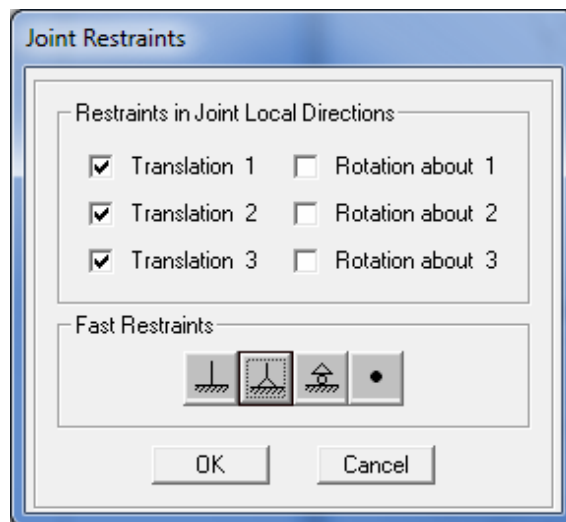

- A. Click on the two joints marking the bottom ends of the two columns ($y=0, z=0$ and $y=36, z=0$). The status bar in the lower left corner should show *2 Points Selected*.
- B. Click on the **Assign menu > Joint > Restraints** command to access the Joint Restraints form shown in Figure 18.
- C. Click 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.

Figure 18
Joint
Restraints
form




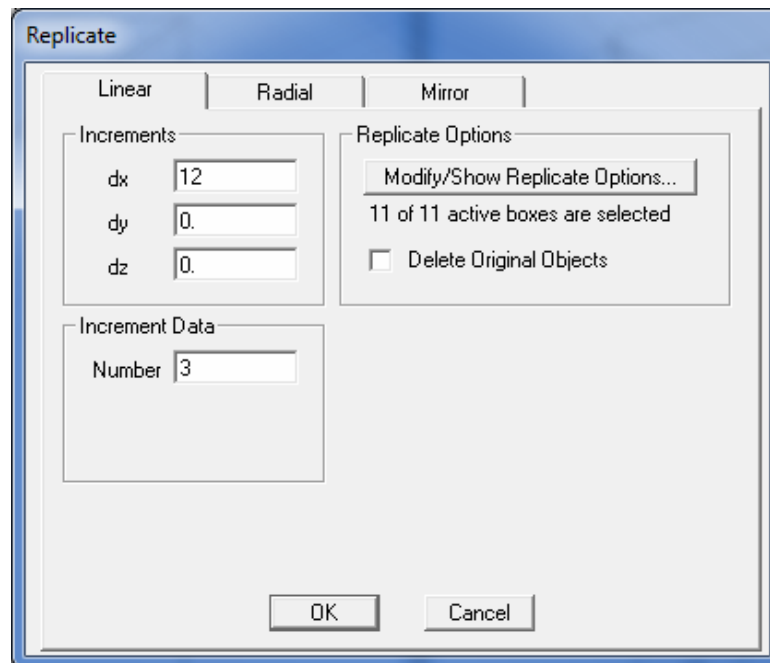
- D. Click the **File menu > Save** command, or the **Save**  button, to save your model.

Step 4 Replicate Frame Objects

In this step, the planar frame just drawn will be replicated at three additional locations.

Make sure that the program is in the Select mode.



- A. Click the **Select > Select > All** command, or the **All**  button, to select the entire model. The status bar should show *16 Points 16 Frames Selected*.
- B. Click the **Edit menu > Replicate** command to access the form shown in Figure 19.
- C. On the Linear tab, type **12** into the dx edit box.
- D. Type **3** into the Number edit box.
- E. Click the **OK** button. Note that three additional frames have been generated.



*Figure 19
Replicate
form*

Draw Additional Frame Objects

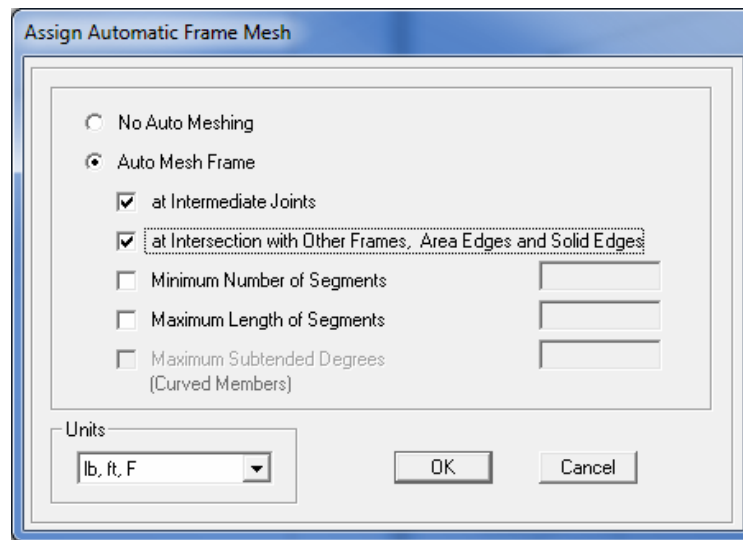
Make sure that the program is in the Select mode, and that the Y-Z view is active.


- A. Click the **View menu > Set 2D View** command.
 1. In the Set 2D View form click on the *X-Z plane* option.
 2. Type **0** into the Y= edit box to display an elevation view, and click **OK**.
- B. Click the **Draw Frame/Cable**  button or use the **Draw menu > Draw Frame/Cable/Tendon** command to display the Properties of Object form.
- C. Select *W8X31* from the Section drop-down list. This will assign this section to the next members you will draw.
- D. In the X-Z Plane @ Y=0 view, click at Grid Point (x=0, z=24) to start, and then click again at Grid Point (x=36, z=24) to draw a horizontal member with L=36.00. Click the right mouse button to lift the pen.
- E. Click at Grid Point (x=0, z=20) to start and then click at Grid Point (x=36, z=20) to draw another member parallel to the one just drawn.
- F. Click the **Set Select Mode**  button, or press the Esc key on the keyboard to exit the **Draw Frame/Cable/Tendon** command.
- G. Select the two horizontal members just drawn by left clicking directly on the objects.
- H. Click on the **Assign menu > Frame > Automatic Frame Mesh** command to access the form in Figure 20. 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 done to ensure connectivity between the beams and the columns because the beams were drawn as single “physical” objects. In reality, it is unlikely that a single beam would be used, and therefore better practice would be to divide these beams into multiple


objects spanning between the columns. But since dividing a frame was illustrated earlier in the tutorial, it was decided that these members would be meshed to demonstrate this feature. With that said, note that this meshing step was not absolutely necessary as the default was set to *at Intermediate Joints*.

Figure 20
Assign
Automatic
Frame Mesh
form



- I. Click the **Assign menu > Clear Display of Assigns** command.
- J. Click the **Get Previous Selection**  button or use the **Select menu > Get Previous Selection** command to again select the two horizontal beam objects.
- K. Click the **Edit menu > Replicate** command to access the Replicate form.
 1. Type **0** into the dx edit box, **36** 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 framing will be replicated at the other side of the building.

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

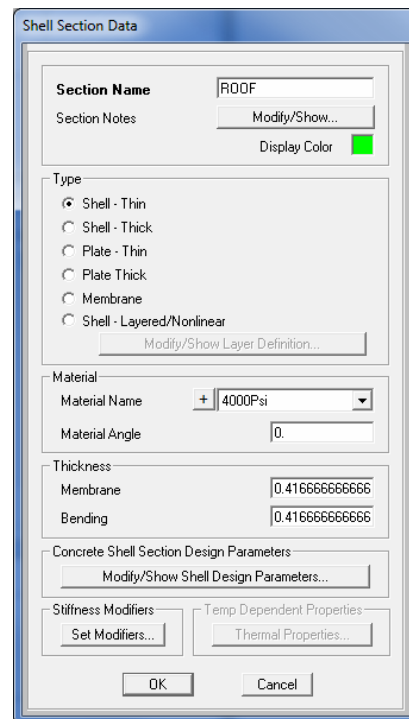
Step 5 Add Area Objects

In this step, a concrete roof 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 the *X-Y plane* option, type **24** into the Z = edit box and click the **OK** button.
- B. Click the **Define menu** > **Section Properties** > **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 21 displays.




*Figure 21
Shell Section
Data
form*

1. Type **ROOF** into the Section Name edit box.
2. Select the *Shell-Thin* option in the Type area.
3. 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.
4. Set the Thickness (both Membrane and Bending) to **0.4167** (or **5 in**) to indicate that the concrete roof is 5 inches thick. By definition, a Shell object has both Membrane and Bending behavior.
5. Click the **OK** button and then click the **OK** button in the Area Sections form to complete the roof definition.

Draw the Area Object

Make sure that the X-Y Plane @ Z=24 view is active. Now draw an area object to represent a roof panel 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 22.



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

Figure 22
Properties of Object box

Properties of Object	
Section	ROOF
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 counter-clockwise around the model, click once at these object points in this order to

draw the outline of the roof: (x=12,y=0), (x=12,y=18) and (x=0,y=18).

- D. Press the Enter key on your keyboard to stop drawing.
- E. Click on the **Set Select Mode**  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 23.

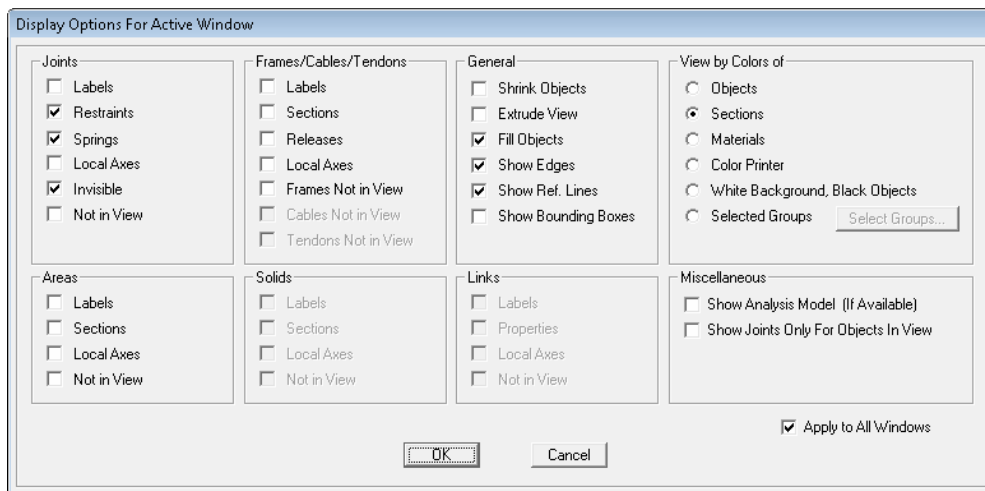


Figure 23
Display Options for Active Window form

- G. Click **OK** to accept the changes.
- H. Click anywhere in the 3-D View (right window) to make it active.
- I. In order to make it easier to differentiate between the frame members and the grid, click on the **View menu > Show Grid** command to toggle off the grid in the 3-D View. The model should now appear as shown in Figure 24.

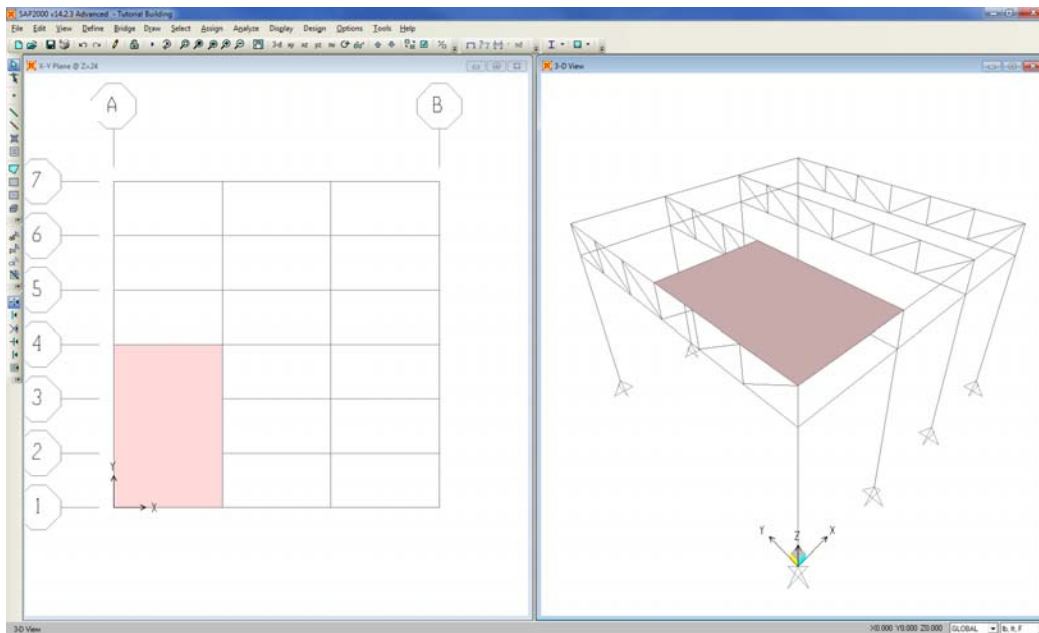


Figure 24
Model after an area object has been drawn

Mesh the Area Object

The area object will now be meshed similar to the previous meshing of frame objects.

- A. Click anywhere in the X-Y Plane @ Z=24 view to make it active.
- B. Right click anywhere on the area object to display the Object Model – Area Information form shown in Figure 25.
- C. Click on the Assignments tab on the Object Model – Area Information form.
- D. Double click in the edit box opposite the Automatic Area Mesh item to display the Assign Automatic Area Mesh form shown in Figure 26.
- E. Select the *Mesh Area Based On Points On Area Edges* option.

- F. Check the *Intersections of Straight Line Objects In Meshing Group With Area Edges* and the *Point Objects In Meshing Group That Are on Area Edges* check boxes.

The area object representing the roof panel was drawn as a single object, but needs to be meshed into multiple analysis elements so that there will be connectivity between the roof 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.

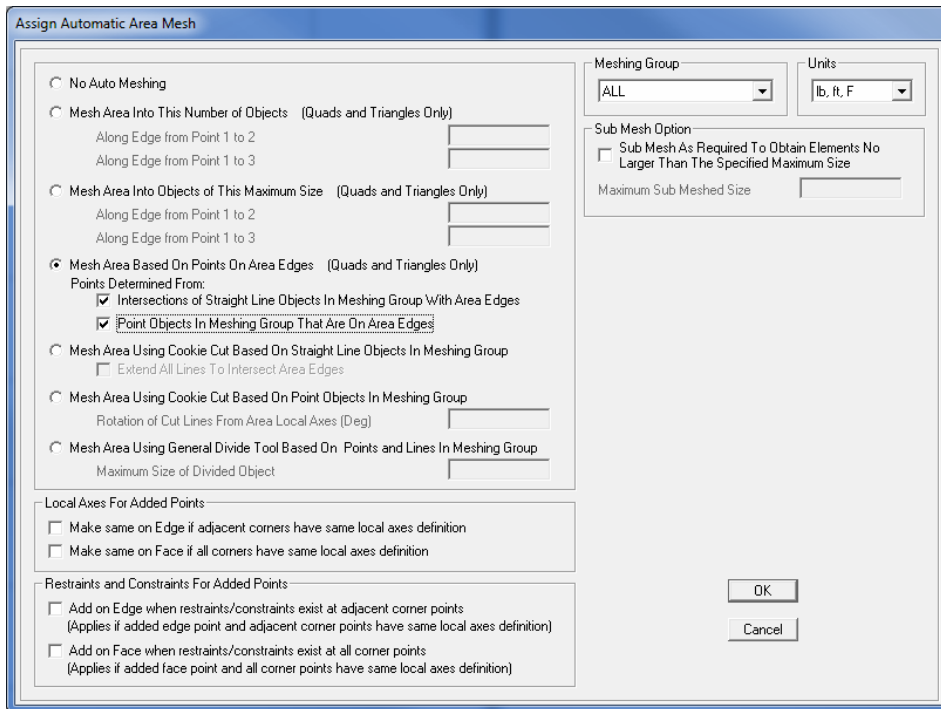
Figure 25
Object Model –
Area Information form

The screenshot shows the 'Object Model - Area Information' dialog box with the 'Location' tab selected. The 'Label' field contains '1'. The 'Section Property' table is as follows:

Section Property	
Section Name	ROOF
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 Mesh	None
Automatic Area Mesh	None
Auto Edge Constraint	No
Material Temp	Default
Group	ALL
Plot Functions	None

Additional controls on the right include a unit dropdown set to 'lb, ft, F', a 'Reset All' button, 'Update Display' and 'Modify Display' buttons, and 'OK' and 'Cancel' buttons. A note at the bottom states: 'Double click white background cell to edit item.'

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



The dialog box titled "Assign Automatic Area Mesh" contains the following options and controls:

- No Auto Meshing
- Mesh Area Into This Number of Objects (Quads and Triangles Only)
 - Along Edge from Point 1 to 2:
 - Along Edge from Point 1 to 3:
- Mesh Area Into Objects of This Maximum Size (Quads and Triangles Only)
 - Along Edge from Point 1 to 2:
 - Along Edge from Point 1 to 3:
- Mesh Area Based On Points On Area Edges (Quads and Triangles Only)
 - Points Determined From:
 - Intersections of Straight Line Objects In Meshing Group With Area Edges
 - Point Objects In Meshing Group That Are On Area Edges
- Mesh Area Using Cookie Cut Based On Straight Line Objects In Meshing Group
 - Extend All Lines To Intersect Area Edges
- Mesh Area Using Cookie Cut Based On Point Objects In Meshing Group
 - Rotation of Cut Lines From Area Local Axes (Deg):
- Mesh Area Using General Divide Tool Based On Points and Lines In Meshing Group
 - Maximum Size of Divided Object:

Additional controls on the right side:

- Meshing Group:
- Units:
- Sub Mesh Option:
 - Sub Mesh As Required To Obtain Elements No Larger Than The Specified Maximum Size
 - Maximum Sub Meshed Size:

Buttons: OK, Cancel

Local Axes For Added Points:

- Make same on Edge if adjacent corners have same local axes definition
- Make same on Face if all corners have same local axes definition

Restrains and Constraints For Added Points:

- Add on Edge when restrains/constraints exist at adjacent corner points (Applies if added edge point and adjacent corner points have same local axes definition)
- Add on Face when restrains/constraints exist at all corner points (Applies if added face point and all corner points have same local axes definition)

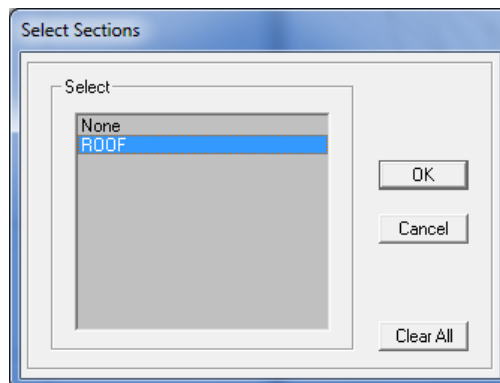
Figure 26
Assign Automatic Area Mesh form

Replicate Area Objects

Make sure that the X-Y Plane @ Z=24 view is active.

- A. Click the **Select > Select > Properties > Area Sections** command to display the Select Sections form shown in Figure 27.

Figure 27
Select Sections form



The dialog box titled "Select Sections" contains the following elements:

- A list box labeled "Select" with the following items:
 - None
 - ROOF (highlighted in blue)
- Buttons: OK, Cancel, Clear All

- B. Click *ROOF* in the Select area to highlight it.
- C. Click on the **OK** button to select the area object.
- D. Click the **Edit menu > Replicate** command to access the Replicate form.
- E. On the Linear tab, type **12** into the dx edit box, **0** into the dy edit box, and **0** into the dz edit box.
- F. Type **2** into the Number edit box.
- G. Click the **OK** button. Note that two additional roof panels have been drawn.
- H. Click the **Select > Select > Properties > Area Sections** command to display the Select Sections form.
- I. Select *ROOF* and click the **OK** button. The status bar should show *3 Areas, 12 Edges Selected*.
- J. Click the **Edit menu > Replicate** command to access the Replicate form.
- K. On the Linear tab, type **0** into the dx edit box, **18** into the dy edit box, and **0** into the dz edit box.
- L. Type **1** into the Number edit box.
- M. Click the **OK** button. Note that roof panels have been drawn on the other side of the building.

Step 6 Move Objects

In this step, objects are moved to create the roof slope. Make sure that the X-Y Plane @ Z=24 view is still active, and that the program is in the Select mode.

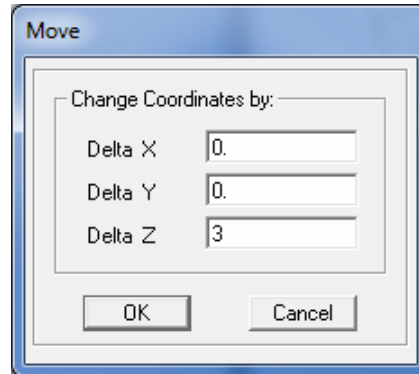
- A. Click on the **Select > Select > Coordinate Specification > Specified Coordinate Range** command to display the Select By Specified Coordinate Range form shown in Figure 28.

- B. Select the *Not Limited* option in the X Coordinate Limits area.
- C. Select the *At A Single Value* option in the Y Coordinate Limits area and select = from the Operator drop-down list, and type **18** into the Value edit box, and **1** into the Tolerance edit box.
- D. Select the *At A Single Value* option in the Z Coordinate Limits area and select = from the Operator drop-down list, and type **24** into the Value edit box, and **1** into the Tolerance edit box.

Figure 28
Select By
Specified
Coordinate
Range form

- E. Click on the **OK** button to accept the changes. The status bar should show *4 Points Selected*.
- F. Click the **Edit menu > Move** command to display the Move form shown in Figure 29.

Figure 29
Move
form



- G. Type **3** into the Delta Z edit box in the Change Coordinates by: area.
- H. Click the **OK** button. Note that the ridge line of the roof has moved up.
- I. Click on the **Select menu > Select > Coordinate Specification > Specified Coordinate Range** command to display the Select By Specified Coordinate Range form.
- J. Select the *Not Limited* option in the X Coordinate Limits area.
- K. Select the *Between Two Values* option in the Y Coordinate Limits area and select **>=** from the first Operator drop-down list, and type **12** into the first Value edit box. Select **<=** from the second Operator drop-down list, and type **24** into the second Value edit box.
- L. Select the *At A Single Value* option in the Z Coordinate Limits area and select **=** from the Operator drop-down list, and type **24** into the Value edit box, and **1** into the Tolerance edit box.
- M. Click on the **OK** button to accept the changes. The status bar should show *8 Points Selected*.
- N. Click the **Edit menu > Move** command to display the Move form.
- O. Type **2** into the Delta Z edit box in the Change Coordinates by: area.

- P. Click the **OK** button. Note that additional truss members have aligned with the roof.
- Q. Click on the **Select menu > Select > Coordinate Specification > Specified Coordinate Range** command to display the Select By Specified Coordinate Range form.
- R. Select the *Not Limited* option in the X Coordinate Limits area.
- S. Select the *Between Two Values* option in the Y Coordinate Limits area and select \geq from the first Operator drop-down list, and type **6** into the first Value edit box. Select \leq from the second Operator drop-down list, and type **30** into the second Value edit box.
- T. Select the *At A Single Value* option in the Z Coordinate Limits area and select $=$ from the Operator drop-down list, and type **24** into the Value edit box, and **1** into the Tolerance edit box
- U. Click on the **OK** button to accept the changes. The status bar should show *8 Points Selected*.
- V. Click the **Edit menu > Move** command to display the Move form.
- W. Type **1** into the Delta Z edit box in the Change Coordinates by: area.
- X. Click the **OK** button. Note that additional truss members have aligned with the roof.
- Y. Click the **View menu > Set 2D View** command. In the Set 2D View form, click the *Y-Z plane* option, type **0** into the X = edit box and click the **OK** button. The model should now appear as shown in Figure 30.

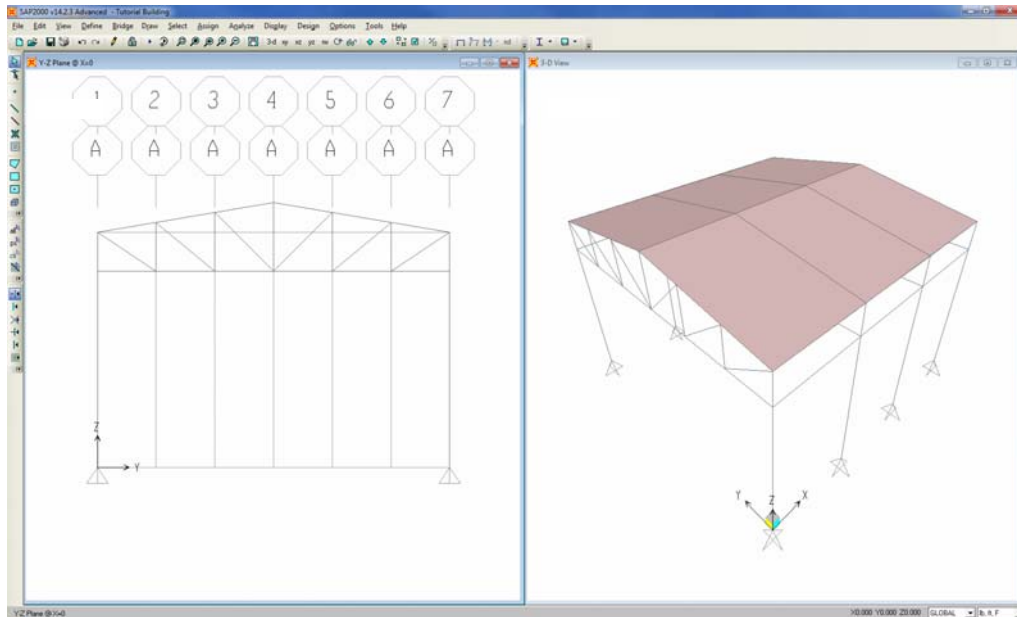


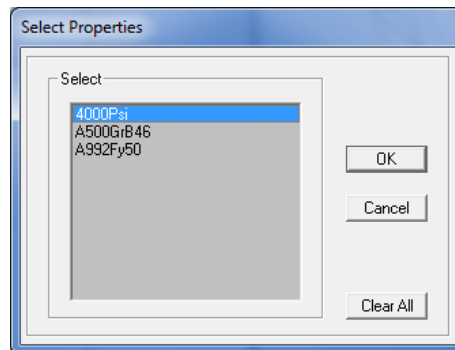
Figure 30
Model after object move

Step 7 Assign Area Stiffness Modifiers

In this step, the membrane properties of the area objects are modified to prohibit the roof panels from acting as flanges for the top chords of the trusses.

- A. Click anywhere in the 3-D View to make it active
- B. Click **Select menu > Select > Properties > Material Properties** to display the Select Properties form shown in Figure 31.
- C. Click *4000Psi* in the Select area to highlight it.
- D. Click on the **OK** button to select all of the concrete area objects.

Figure 31
Select Sections form



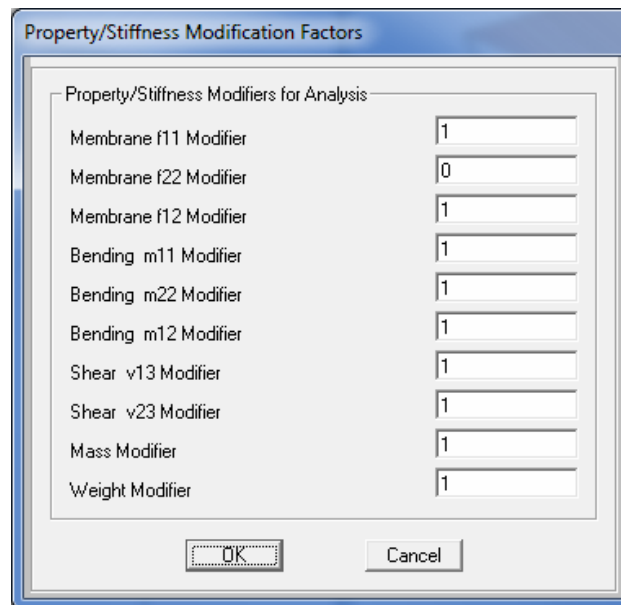
- E. Click the **Assign menu > Area > Area Stiffness Modifiers** command to access the Property/Stiffness Modification Factors form shown in Figure 32.


1. Type **0** in the Membrane f22 Modifier edit box.

This action will prohibit the roof area objects from carrying in-plane axial loads.

2. Click **OK** to accept the assignment.

Figure 32
Property/Stiffness Modification Factors form



- F. Click the **Assign menu > Clear Display of Assigns** command to clear the display of the stiffness modifiers.
- G. Click the **File menu > Save** command, or the **Save**  button, to save your model.

Step 8 Define Load Patterns

The loads used in this tutorial consist of dead, live and wind static load patterns. The dead and live loads act in the gravity direction, while the wind load will be applied as a surface pressure to the roof objects.

For this example, assume that the dead load pattern consists of the self-weight of the building plus an additional 20 kips at each lower chord panel point of each truss. The live load pattern is taken to be 30 pounds per square foot (psf) applied to the projected area of the concrete roof. The wind load pattern is taken to be 15 psf applied to the windward roof, and 6 psf applied to the leeward side of the roof.

- A. Click the **Define menu > Load Patterns** command to access the Define Load Patterns form shown in Figure 33. Note there is only a single default load pattern defined, which is a dead load pattern with self-weight (DEAD).

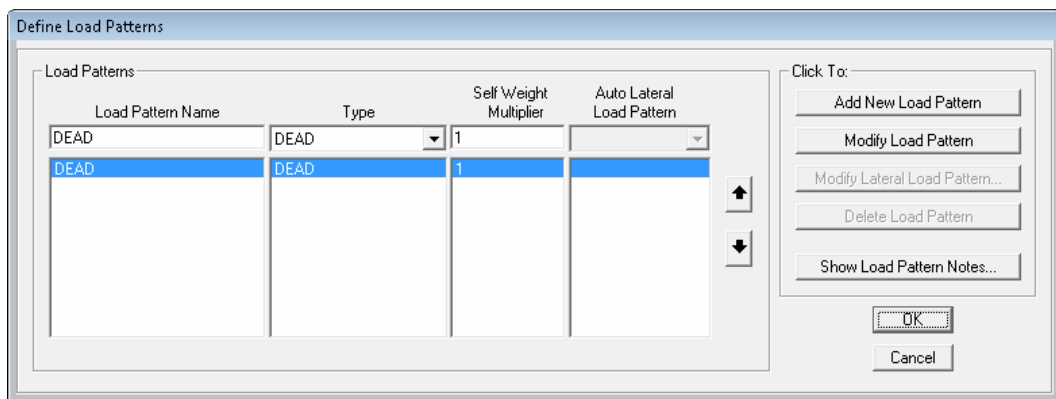


Figure 33
Define Load Patterns form

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

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

- B. Click in the edit box for the Load Pattern Name column. Type the name of the new pattern, **LIVE**. Select a Type of load pattern 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 Pattern** button to add the LIVE load pattern to the load list.
- C. Click in the edit box for the Load Pattern Name column once again. Type the name of the last new pattern, **WIND**. Select *WIND* from the Type drop-down list. Make sure that the Self Weight Multiplier is set to zero, and that *None* is showing in the Auto Lateral Load Pattern drop-down list. Click the **Add New Load Pattern** button to add the WIND load pattern to the load list.

The Define Load Patterns form should now appear as shown in Figure 34. Click the **OK** button in that form to accept the newly defined load patterns.

Load Pattern Name	Type	Self Weight Multiplier	Auto Lateral Load Pattern
WIND	WIND	0	None
DEAD	DEAD	1	
LIVE	LIVE	0	
WIND	WIND	0	None

Figure 34
The Define Load Patterns form after all of the load patterns have been defined

Step 9 Assign Loads

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


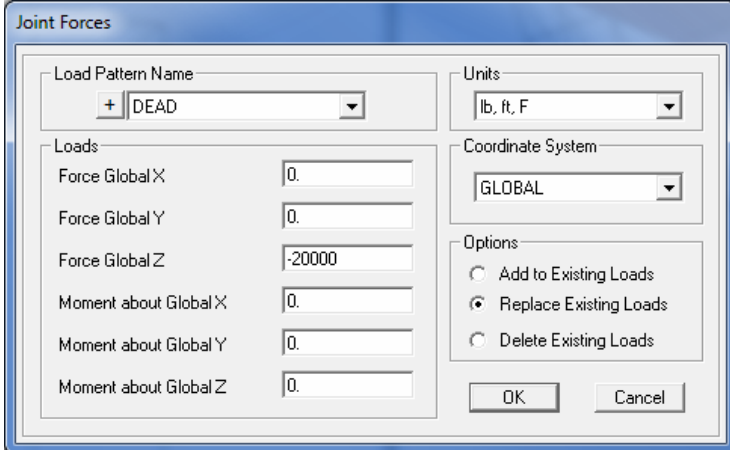

- A. Click on each of the panel points on the bottom chord of the truss. The status bar should show *5 Points Selected*. If you make a mistake in selecting, click the **Clear Selection**  button, and try again.
- B. Select the **Assign menu > Joint Loads > Forces** command to access the Joint Forces form. Select *DEAD* from the Load Pattern Name drop-down list as shown in Figure 35. Clicking on the + button will display the Define Load Patterns form where load patterns may be altered or added.
 1. Type **-20000** in the Force Global Z edit box in the Loads area. The minus sign indicates that the load is applied in the negative Global Z direction.

Figure 35
Joint Forces form



The screenshot shows the 'Joint Forces' dialog box. It has a title bar 'Joint Forces'. Inside, there are several sections:

- Load Pattern Name:** A dropdown menu with a '+' button on the left and 'DEAD' selected.
- Units:** A dropdown menu with 'lb. ft. F' selected.
- Coordinate System:** A dropdown menu with 'GLOBAL' selected.
- Options:** Three radio buttons: 'Add to Existing Loads' (unselected), 'Replace Existing Loads' (selected), and 'Delete Existing Loads' (unselected).
- Loads:** A list of six input fields:
 - Force Global X: 0.
 - Force Global Y: 0.
 - Force Global Z: -20000
 - Moment about Global X: 0.
 - Moment about Global Y: 0.
 - Moment about Global Z: 0.
- Buttons:** 'OK' and 'Cancel' buttons at the bottom right.

2. Click the **OK** button to accept the dead load.
- C. Click the **Get Previous Selection**  button or use the **Select menu > Get Previous Selection** command to again select the bottom chord panel points.

- D. Click the **Edit menu > Replicate** command to display the Replicate form.
1. On the Linear tab, type **12** into the dx edit box, **0** into the dy edit box, and **0** into the dz edit box.
 2. Type **3** into the Number edit box.
 3. Click the **OK** button to accept the changes
- E. Click the **Select menu > Select > Properties > Area Sections** command to display the Select Sections form.
1. Click on *ROOF* in the Select area to highlight it.
 2. Click the **OK** button to select all of the roof area objects.
- F. Select the **Assign menu > Area Loads > Uniform (Shell)** command to access the Area Uniform Loads form. Select *LIVE* from the Load Pattern Name drop-down list as shown in Figure 36.

Figure 36
Area Uniform Loads form

The screenshot shows the 'Area Uniform Loads' dialog box. It has a title bar 'Area Uniform Loads'. Inside, there are several sections: 'Load Pattern Name' with a dropdown menu showing 'LIVE' and a '+' icon; 'Units' with a dropdown menu showing 'lb. ft. F'; 'Uniform Load' section with 'Load' set to '30', 'Coord System' set to 'GLOBAL', and 'Direction' set to 'Gravity Projected'; and 'Options' section with three radio buttons: 'Add to Existing Loads', 'Replace Existing Loads' (which is selected and has a dashed border), and 'Delete Existing Loads'. At the bottom, there are 'OK' and 'Cancel' buttons.


1. Type **30** in the Load edit box in the Uniform Load area.
2. Select *Gravity Projected* from the Direction drop-down list. Remember that the Gravity direction is in the negative Global Z direction.
3. Click the **OK** button to accept the live load.

- G. Click on the **Select menu > Select > Coordinate Specification > Specified Coordinate Range** command to display the Select By Specified Coordinate Range form.
1. Verify that the *Entirely Inside The Specified Selection Volume* option is selected in the Select All Objects That Are: area.
 2. Uncheck all of the check boxes except for *Area* in the Select These Object Types area.
 3. Select the *Not Limited* option in the X Coordinate Limits area.
 4. Select the *Between Two Values* option in the Y Coordinate Limits area and select \geq from the first Operator drop-down list, and type **0** into the first Value edit box. Select \leq from the second Operator drop-down list, and type **18** into the second Value edit box.
 5. Select the *Not Limited* option in the Z Coordinate Limits area.
 6. Click the **OK** button to select the roof area objects on the windward side. The status bar should show *3 Areas, 12 Edges Selected*.
- H. Click the **Assign menu > Area Loads > Surface Pressure (All)** command to display the Area Surface Pressure Load form shown in Figure 37.

Figure 37
Area Surface Pressure Load form

The screenshot shows the 'Area Surface Pressure Load' dialog box. The 'Load Pattern Name' is set to '+ WIND' and the 'Units' are 'lb. ft. F'. Under the 'Pressure' section, the 'By Element' radio button is selected, and the 'Pressure' value is 15. The 'Face' is set to 'Top'. In the 'Options' section, the 'Replace Existing Loads' radio button is selected. The 'By Joint Pattern' section is currently unselected.

1. Select *WIND* from the Load Pattern Name drop-down list.
 2. Select the *By Element* option in the Pressure area.
 3. Type **15** in the Pressure edit box.
 4. Select *Top* from the Face drop-down list.
 5. Click the **OK** button to apply a wind pressure load inward to the top face of the windward roof area objects.
- I. Click on the **Select menu > Select > Coordinate Specification > Specified Coordinate Range** command to display the Select By Specified Coordinate Range form.
1. Verify that the *Entirely Inside The Specified Selection Volume* option is selected in the Select All Objects That Are: area.
 2. Verify that only the check box for *Area* in the Select These Object Types area is checked.
 3. Select the *Not Limited* option in the X Coordinate Limits area.
 4. Select the *Between Two Values* option in the Y Coordinate Limits area and select \geq from the first Operator drop-down list, and type **18** into the first Value edit box. Select \leq from the second Operator drop-down list, and type **36** into the second Value edit box.
 5. Select the *Not Limited* option in the Z Coordinate Limits area.
 6. Click the **OK** button to select the roof area objects on the leeward side. The status bar should show *3 Areas, 12 Edges Selected*.
- J. Click the **Assign menu > Area Loads > Surface Pressure (All)** command to display the Area Surface Pressure Load form.
1. Select *WIND* from the Load Pattern Name drop-down list.
 2. Select the *By Element* option in the Pressure area.

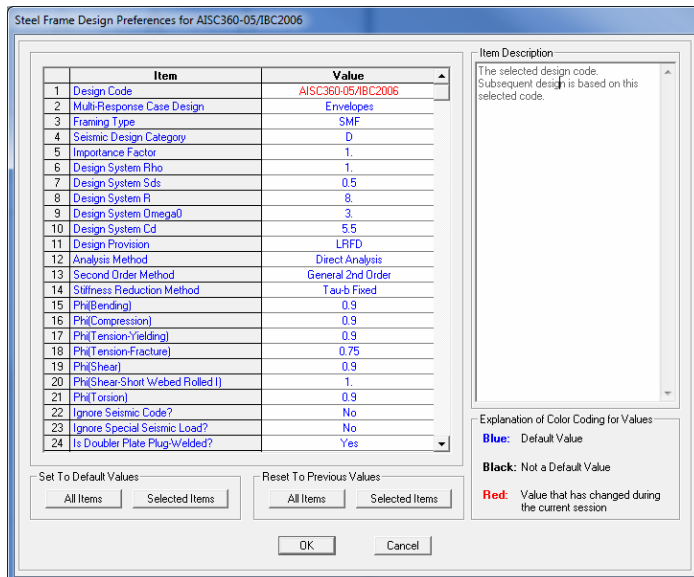
3. Type **-6** in the Pressure edit box.
 4. Select *Top* from the Face drop-down list.
 5. Click the **OK** button to apply a wind pressure load outward to the top face of the leeward roof area objects.
- K. Click the **Assign menu > Clear Display of Assigns** command to clear the display of the assigned loads.
- L. Click the **File menu > Save** command, or the **Save**  button, to save your model.

Step 10 Define Load Combinations

In this step, the steel design code will be selected and the load combinations required for design will be automatically generated.

- A. Click the **Design menu > Steel Frame Design > View/Revise Preferences** command to display the Steel Frame Design Preferences form shown in Figure 38.

Figure 38
Steel Frame
Design Preferences
form



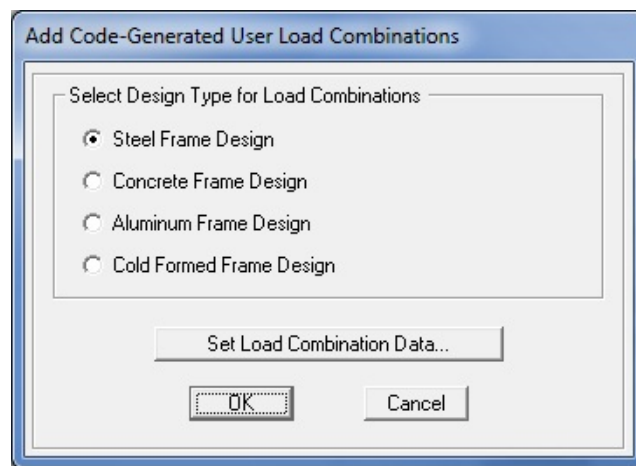
The dialog box displays a table of design preferences with columns for Item, Value, and Item Description. The values are color-coded: blue for default values and black for non-default values. A legend at the bottom right explains the color coding.

Item	Value	Item Description
1 Design Code	AISC360-05/IBC2006	The selected design code. Subsequent design is based on this selected code.
2 Multi-Response Case Design	Envelopes	
3 Framing Type	SMF	
4 Seismic Design Category	D	
5 Importance Factor	1.	
6 Design System Rho	1.	
7 Design System Sds	0.5	
8 Design System R	8.	
9 Design System Omega0	3.	
10 Design System Cd	5.5	
11 Design Provision	LRFD	
12 Analysis Method	Direct Analysis	
13 Second Order Method	General 2nd Order	
14 Stiffness Reduction Method	Tau-b Fixed	
15 Phi(Bending)	0.9	
16 Phi(Compression)	0.9	
17 Phi(Tension-Yielding)	0.9	
18 Phi(Tension-Fracture)	0.75	
19 Phi(Shear)	0.9	
20 Phi(Shear-Short Webbed Rolled I)	1.	
21 Phi(Torsion)	0.9	
22 Ignore Seismic Code?	No	
23 Ignore Special Seismic Load?	No	
24 Is Doubler Plate Plug/Welded?	Yes	

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

- B. Select *AISC360-05/IBC2006* from the Design Code drop-down list.
- C. Click the **OK** button to accept the design code changes.
- D. Select the **Define menu > Load Combinations** command to display the Define Load Combinations form.
- E. Click the **Add Default Design Combos...** button to display the Add Code-Generated User Load Combinations form shown in Figure 39.

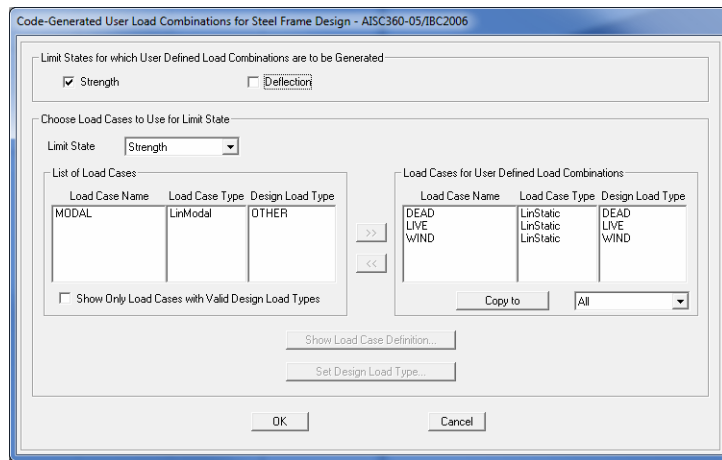
Figure 39
Add Code-Generated
User Load
Combinations form



- F. Select the *Steel Frame Design* option under the Select Design Type for Load Combinations area.
- G. Click the **Set Load Combination Data** button to display the Code-Generated User Load Combinations for Steel Frame Design form shown in Figure 40.
- H. Uncheck the *Deflection* checkbox in the Limit States for which User Defined Load Combinations are to be Generated area - this tutorial is concerned only with strength limit states.
- I. Click the **OK** button twice to return to the Define Load Combinations form. Note that a total of six load combinations have been generated.
 - 1. Click on *UDSTL3* in the Load Combinations area to highlight it.

2. Click the **Modify/Show Combo** button to display the Load Combination Data form. Note that this particular combination consists of 1.2 DEAD + 1.0 LIVE + 1.6 WIND.
 3. Click the **OK** button to return to the Define Load Combinations form.
- J. Repeat Step I for other load combinations if so desired, or click the **OK** button to leave the Define Load Combinations form.

Figure 40
Code-Generated
User Load
Combinations
for Steel Frame
Design form



Step 11 Run the Analysis

In this step, the analysis model will be viewed and the analysis will be run.

- A. Click anywhere in the 3-D View to make it active.
- B. Click the **Set Display Options** button. When the form appears, check the *Show Analysis Model* check box as shown in Figure 41.

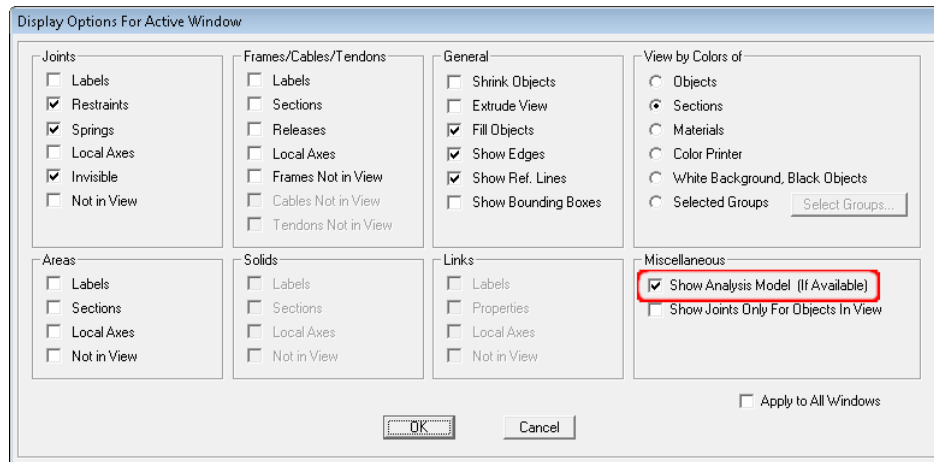


Figure 41
Display Options for Active Window form

- C. Click the **OK** button to accept the display setting.
- D. If the model has not previously been analyzed, a message similar to the one in Figure 42 displays.

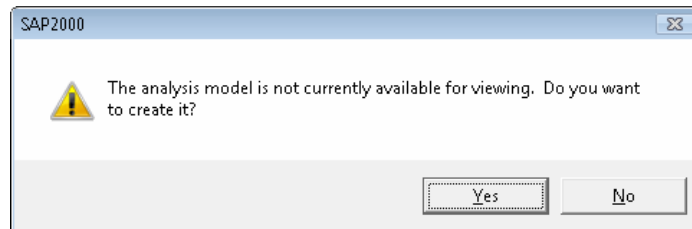
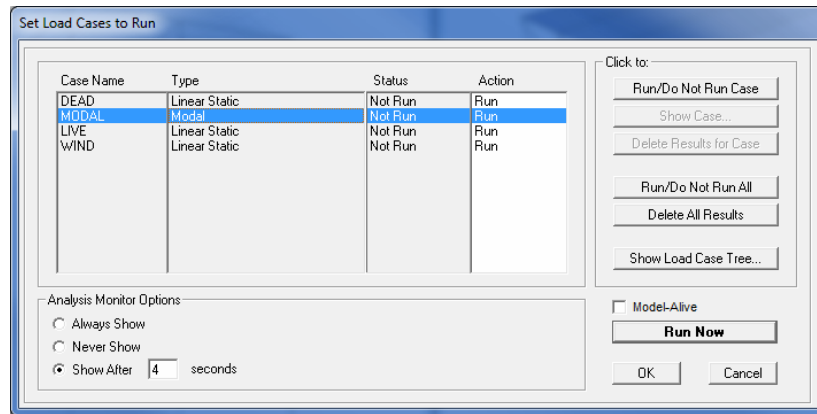


Figure 42
Analysis Model message

- E. Click the **Yes** button to display the analysis model. Take a moment to verify that the element formation is as expected.
- F. Click the **Analyze menu > Run Analysis** command or the **Run Analysis** button, to access the Set Load Cases to Run form as shown in Figure 43.


Figure 43
Set Load
Cases to Run
form



Note that the program has automatically defined four different load cases: DEAD, MODAL, LIVE and WIND based on the load patterns 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 an analysis monitor window may display, depending on the duration of analysis. Analysis information may be accessed at a later time by going to the **File menu > Show Input/Log Files** command and selecting the file with the *.LOG* extension.

- G. When the analysis is finished, 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 12 Graphically Review the Analysis Results

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


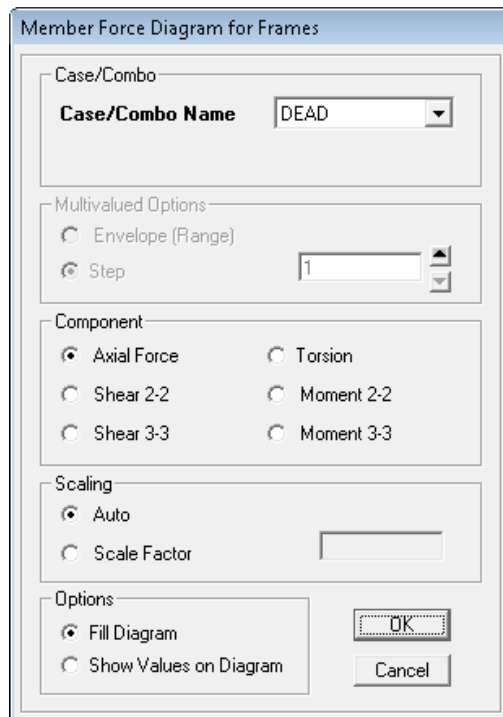
- A. Click anywhere in the Y-Z Plane @ X=0 view to make it active (the elevation view).
- B. Click the **Show Forces/Stresses > Frames/Cables** button, , or the **Display menu > Show Forces/Stresses > Frames/Cables** command to bring up the Member Force Diagram for Frames form shown in Figure 44.
 1. 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.

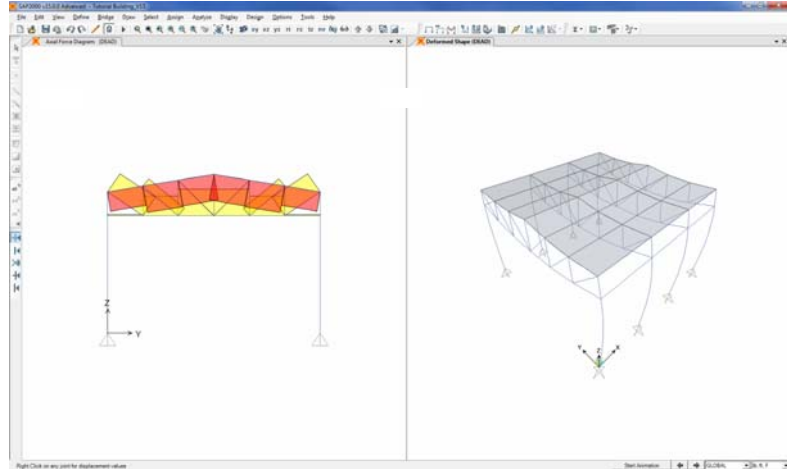
Figure 44
Member Force
Diagram for
Frames form



4. Select the *Fill Diagram* option.

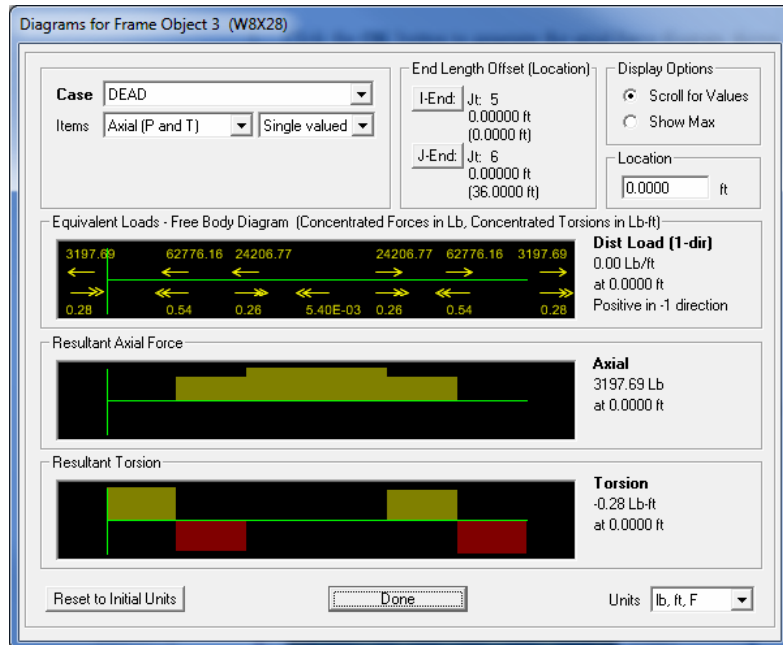
- Click the **OK** button to generate the axial force diagram shown in Figure 45.

*Figure 45
Axial force diagram in
an elevation view*




- Right click on the bottom chord member in the Y-Z view to access the Diagram for Frame Object form shown in Figure 46.

*Figure 46
Force details
obtained by
right-clicking
bottom chord of
truss in the
elevation view in
Figure 45*

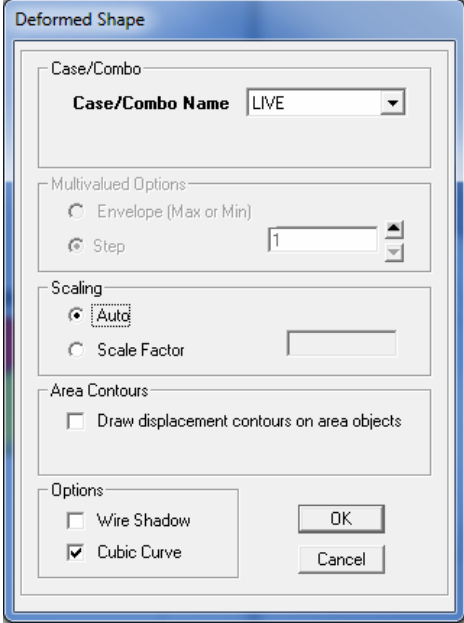


Note that the program displays the force diagrams for the entire bottom chord object just as it was drawn, even though the program has automatically meshed the frame object into smaller elements for analysis.

1. Click the *Scroll for Values* option in the Display Options area, and move the mouse over the diagrams with the left button held down to obtain values at any location.
 2. Click the **Done** button to close the form.
- D. Make sure that the Y-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 47.

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 48.

Figure 47
Deformed Shape form



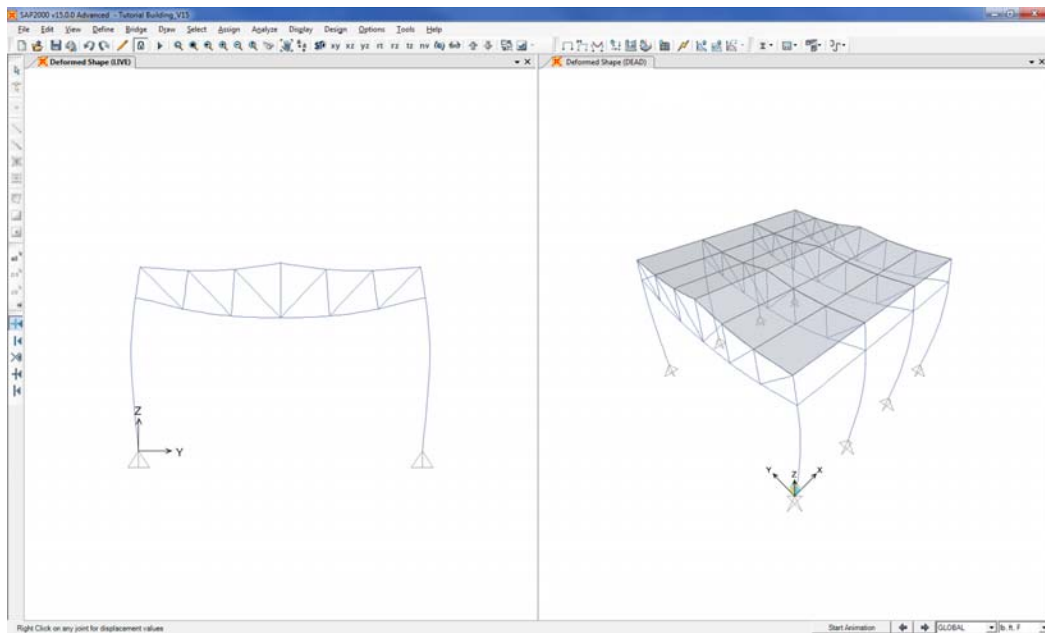


Figure 48
Deformed Shape in an elevation view

- E. Right click on the middle joint on the top chord object in Figure 48 to display the Joint Displacements results form shown in Figure 49.

Figure 49
Joint Displacements obtained by right-clicking a joint shown in the elevation view in Figure 48

Joint Displacements			
Joint Object	Joint Element		
12	1	2	3
Trans	6.439E-05	0.00000	-0.00192
Rotn	0.00000	1.366E-04	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 13 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 **Design menu > Steel Frame Design > View/Revise Preferences** command. The Steel Frame Design Preferences form shown in Figure 50 displays.

Item	Value
1 Design Code	AISC360-05/IBC2006
2 Multi-Response Case Design	Envelopes
3 Framing Type	SMF
4 Seismic Design Category	D
5 Importance Factor	1.
6 Design System Rho	1.
7 Design System Sds	0.5
8 Design System R	8.
9 Design System Omega0	3.
10 Design System Cd	5.5
11 Design Provision	LRFD
12 Analysis Method	Direct Analysis
13 Second Order Method	General 2nd Order
14 Stiffness Reduction Method	Tau-b Fixed
15 Phi(Bending)	0.9
16 Phi(Compression)	0.9
17 Phi(Tension-Yielding)	0.9
18 Phi(Tension-Fracture)	0.75
19 Phi(Shear)	0.9
20 Phi(Shear-Short Webed Rolled I)	1.
21 Phi(Torsion)	0.9
22 Ignore Seismic Code?	No
23 Ignore Special Seismic Load?	No
24 Is Doubler Plate Plug-Welded?	Yes

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


Set To Default Values:

Reset To Previous Values:

Figure 50
Steel Frame Design Preferences form

1. Remember that back in Step 10 the Design Code was set to *AISC360-05/IBC2006*. Click in the Design Code drop-down list to see the many other available design codes. Return to the *AISC360-05/IBC2006* code. The user is encouraged to examine

the design code options available using the **Design menu > Steel Frame Design > View/Revise Preferences** command.

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 for the truss members, and checking the columns and other framing members that are not part of the truss.

When the design is complete, the selected sizes and stress ratios are displayed on the model. The model appears as shown in Figure 51.

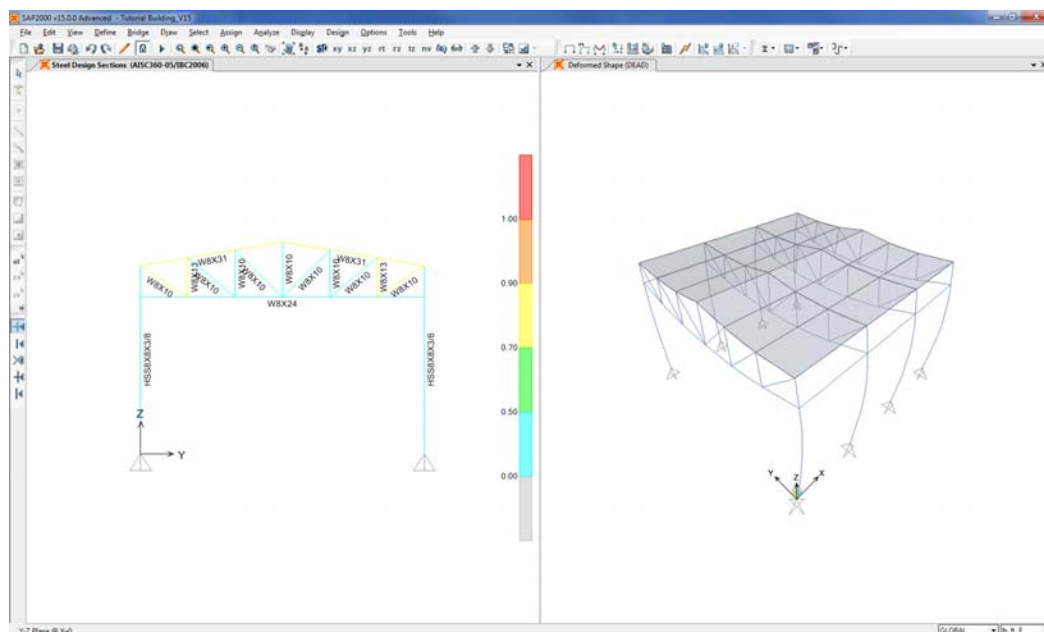


Figure 51
Model after the steel frame design

- C. Click the **Design menu > Steel Frame Design > Verify Analysis vs Design Section** command. A message similar to the one in Figure 52 appears. Click the **No** button to close the form.

*Figure 52
Analysis vs
Design Section*



In the initial analysis (Step 11), the program used the median section by weight from the TRUSS auto select section list for the truss members. During design (this step), the program selected different sections by optimizing from the auto select list of members contained in TRUSS. The number of iteration cycles is only one. Because new member sizes are now present in the model, the original analysis results are no longer valid and the analysis should be rerun.

The goal is to repeat the analysis and design process until the analysis and design sections for the truss are all the same. Note that when the building is reanalyzed, SAP2000 will use the current design sections (i.e., those selected in Step 13) as new analysis sections for the next analysis run.

- D. Right click on one of the truss bottom chord members in the Y-Z view (shown in Figure 51). The Steel Stress Check Information form shown in Figure 53 displays. 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 has automatically created code-specific design load combinations in addition to the user specified combinations previously defined in Step 10. In this instance, the two sets of combinations are identical as they were both generated from the same code specification.

Steel Stress Check Information (AISC360-05/IBC2006)

Frame ID: 3 Analysis Section: W8x28
 Design Code: AISC360-05/IBC2006 Design Section: W8x24

COMBO ID	STATION LOC	MOMENT RATIO	INTERACTION CHECK =	AXL + B-MAJ + B-MIN	-MAJ-SHR	MIN-SHR
DSTL1	16.00	0.442 (T)	=	0.396 + 0.046 + 0.000	0.010	0.000
DSTL1	18.00	0.425 (T)	=	0.396 + 0.029 + 0.000	0.011	0.000
DSTL1	18.00	0.425 (T)	=	0.396 + 0.029 + 0.000	0.011	0.000
DSTL1	20.00	0.442 (T)	=	0.396 + 0.046 + 0.000	0.010	0.000
DSTL1	22.00	0.457 (T)	=	0.396 + 0.060 + 0.000	0.008	0.000
DSTL1	24.00	0.469 (T)	=	0.396 + 0.073 + 0.000	0.007	0.000

Modify/Show Overwrites Display Details for Selected Item Display Complete Details

Strength Deflection Stylesheet: Default

Figure 53
Steel Stress Check Information form

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 AISC360-05/IBC2006 form shown in Figure 54 displays. Use the File menu on the form to print the data.

Click the **X** in the upper right-hand corner of the Steel Stress Check Data AISC360-05/IBC2006 form to close it. Click the **Cancel** button to close the Steel Stress Check Information form.


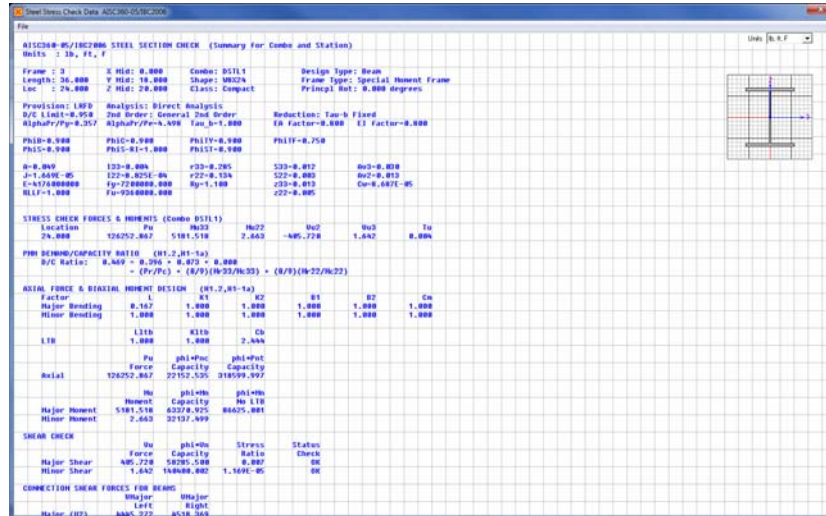
- E. To rerun the analysis with the new analysis sections, click the **Analyze menu > Run Analysis** command or the **Run Analysis**  button, and then click the **Run Now** button on the Set Load Cases to Run form.

Figure 54
Steel Stress Check
Dataform




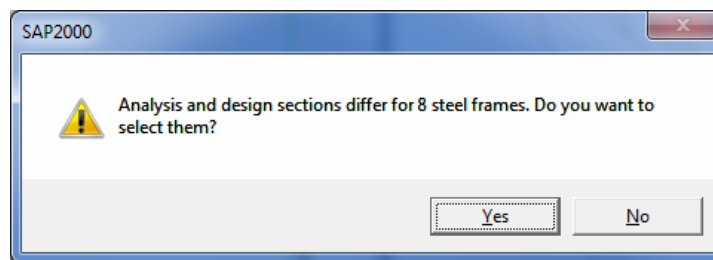
- F. When the analysis is complete, click the **Design menu > Steel Frame Design > Start Design/Check of Structure** command or the **Start Steel Design/Check of Structure**  button, to start the steel frame design process.
- G. When the design is complete, click the **Design menu > Steel Frame Design > Verify Analysis vs Design Section** command. A message similar to the one in Figure 55 displays.

Figure 55
Analysis vs Design
Section message

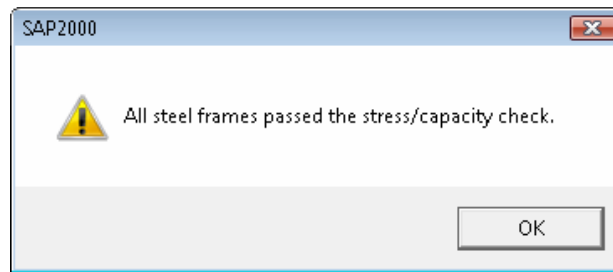


The message in Figure 55 indicates the number of analysis sections that differ from the design sections. Click the **No** button if sections do not match, or the **OK** button if they do match, to close the form.


Repeat Action Items E through G until the message received indicates that all analysis and design sections match. This may take numerous iterations depending on 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 56 should appear indicating that all members passed.

*Figure 56
Stress/capacity
check message*



Note that members not passing at this stage is an indication of inadequate sections, either in the auto select section list or in the other section assignments. With the auto select section list, the program would have used the largest section in the list for both the analysis and design, so a message stating that members do not pass indicates that the auto select section list needs modification. In that case, add larger sections to the auto select sections list. For other members that do not pass, re-assign larger sections 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 is now complete.