SAP2000®

Integrated
Finite Element Analysis
and
Design of Structures

QUICK TUTORIALS



Computers and Structures, Inc.
Berkeley, California, USA

Version 7.0
Revised October 1998

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

tel: (510) 845-2177
fax: (510) 845-4096
e-mail: *info@csiberkeley.com*
web: *www.csiberkeley.com*

© Copyright Computers and Structures, Inc., 1978–1998.
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

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.

ACKNOWLEDGMENT

Thanks are due to all of the numerous structural engineers, who over the years have given valuable feedback that has contributed toward the enhancement of this product to its current state.

Special recognition is due Dr. Edward L. Wilson, Professor Emeritus, University of California at Berkeley, who was responsible for the conception and development of the original SAP series of programs and whose continued originality has produced many unique concepts that have been implemented in this version.

Table of Contents

CHAPTER I Introduction **1**

Overview 1
Organization 2
Recommended Reading 2

CHAPTER II Basic Tutorial **3**

Overview 3
Description of the Model 4
Starting the Tutorial 4
Setting Up the Geometry 6
Defining Structural Sections 7
Assigning Member Sections 9
Defining Load Cases 11
Assigning Joint Loads 12
Analyzing the Model 14
Displaying the Deformed Shape 15
Displaying Member Forces 18
Design Stress Check 20
Modifying the Structure 21
Concluding Remarks 25

CHAPTER III Concrete Design Tutorial **27**

Overview 27
Description of the Model 28
Starting the Tutorial 30
Opening the Model Database File 31

Analyzing the Model 32
Selecting the Design Code 34
Starting Design 34
Changing Member Properties 38
Concluding Remarks 43

CHAPTER IV Steel Design Tutorial 45

Overview 45
Description of the Model 46
Starting the Tutorial 48
Opening the Model Database File 48
Analyzing the Model 50
Selecting the Design Code 51
Starting Design and Stress Check 52
Modifying Member Properties 56
Selecting Sections Automatically 59
Re-analyzing with Updated Elements 62
Concluding Remarks 64

Chapter I

Introduction

Overview

SAP2000 provides the user with options to create, modify, analyze and design structural models, all from within the same user interface. This program features powerful and completely integrated modules for design of both steel and reinforced concrete structures. The program provides an interactive environment in which the user can study the stress conditions, make appropriate changes, such as member size revisions, and update the design without re-analyzing the structure.

This manual is intended to provide three quick tutorials aiming at giving the first time users hands-on experience. In the first tutorial, some of the basic modeling and design features of the SAP2000 are explored. Several features of SAP2000 steel and concrete design are explored in the subsequent two tutorials.

Step-by-step procedures for the modeling and design of the structures are outlined in the tutorials. It is recommended that you actually perform these steps while reading the chapters.

Organization

This manual is organized in the following way:

Chapter II provides a quick tutorial aiming at giving the first time user hands-on experience. Some of the basic features of SAP2000 are explored in this tutorial.

Chapter III provides a quick tutorial for steel design. Several of the basic features of SAP2000 steel design are explored in this tutorial.

Chapter IV provides a quick tutorial for concrete design. Several of the basic features of SAP2000 concrete design are explored in this tutorial.

Recommended Reading

It is recommended that first time users read "The Graphical User Interface" chapter in the *SAP2000 Getting Started* manual before starting the tutorials given in this book. That chapter will provide basic understanding of the menus and buttons of SAP2000 graphical user interface.

After completing the tutorials given in this book, the user might be interested in following through the more detailed tutorials given in the *SAP2000/NL-Push Detailed Tutorial Including Pushover Analysis Manual*.

Chapter II

Basic Tutorial

Overview

This tutorial is aimed at giving the first-time user hands-on experience while describing a few of the basic features and capabilities of SAP2000. It is assumed that you have read the chapter, "The Graphical User Interface" provided in the *SAP2000 Getting Started* manual. It is recommended that you use the comprehensive online Help included in the program. You may also wish to refer to the *SAP2000 Basic Analysis Reference* in the third part of this volume for more information about the structural model.

As you become familiar with the program, you will realize that the order of some of the steps described here is immaterial. In other words, after some practice, you may choose to perform the operations in a different order to set up and run the same model.

We will use the SAP2000 commands either from the Toolbars or from the menus. This is done intentionally to familiarize you with both methods. The Toolbars provide quick access to commonly used features. All of the features available on the Toolbars can also be accessed from the Menu Bar.

Description of the Model

The model chosen for this tutorial is developed, analyzed, design-checked, and then modified. A simple two-dimensional five-bay truss of the "Warren" type is selected from the program template, subjected to both dead and live point loads, and then analyzed for the two load cases. A design stress check is performed, in accordance with AISC/ASD89, to examine the interaction stress ratios. The geometry and loading of the model are then modified, and the process is repeated.

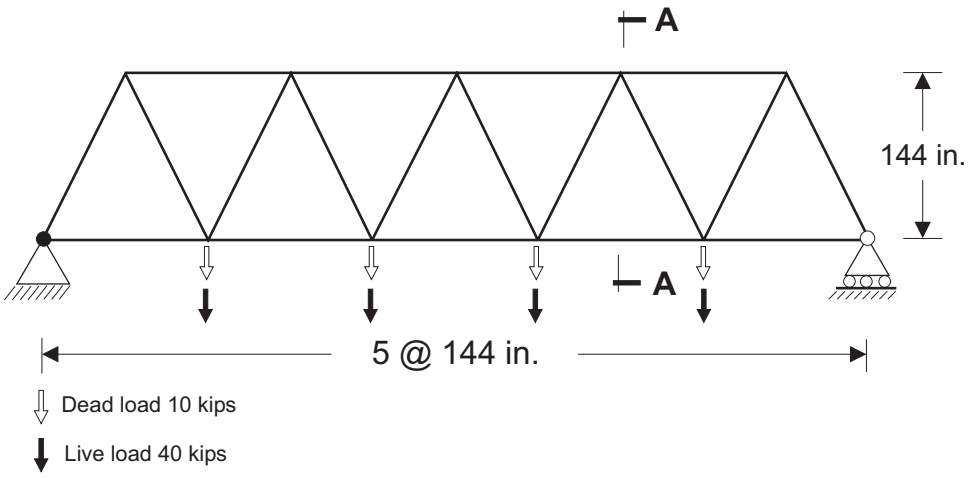
The initial model is shown in Figure II.1. Kip-inch units are used. Structural steel (36 ksi yield strength) Double Angles are used throughout the model.

Starting the Tutorial

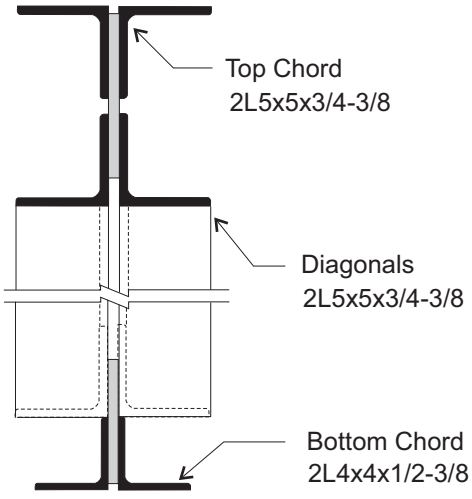
The following topics describe the step-by-step procedure for creating and analyzing the truss model. It is recommended that you actually perform these steps using the program while you are reading this chapter.

The geometry is obtained from an available template. The templates represent a number of common structural configurations. Remember that once the structural geometry has been set up, the order of steps is left to your discretion. Shortcuts will come with experience.

Start the program by running SAP2000 from the Start Menu or from Program Manager, depending on your version of Windows. We will now proceed to develop the model.



Truss Elevation



Section A-A

(Typ.)

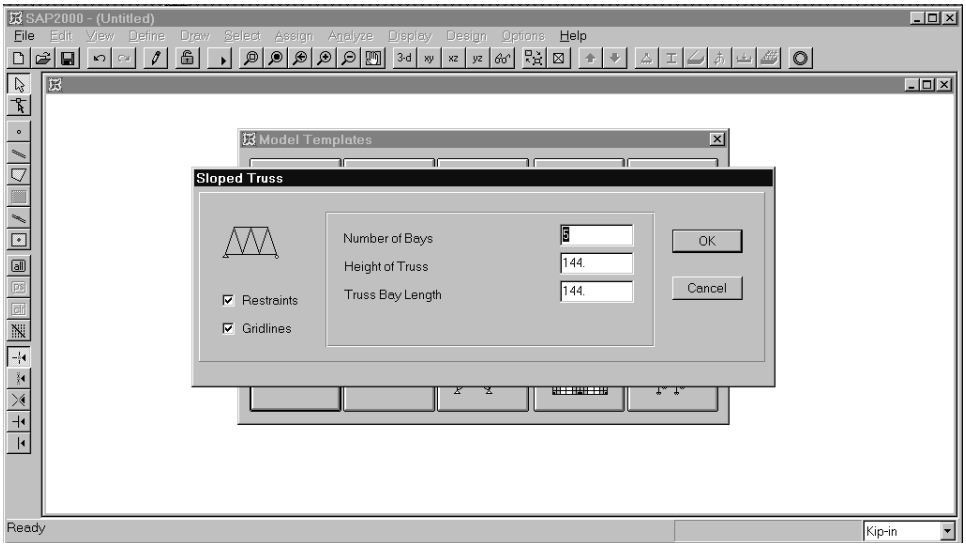
Notes:

- Kip-inch units are used.
- Self weight of truss is included in load case LOAD1.
- Minimum yield stress for steel, Fy = 36 ksi

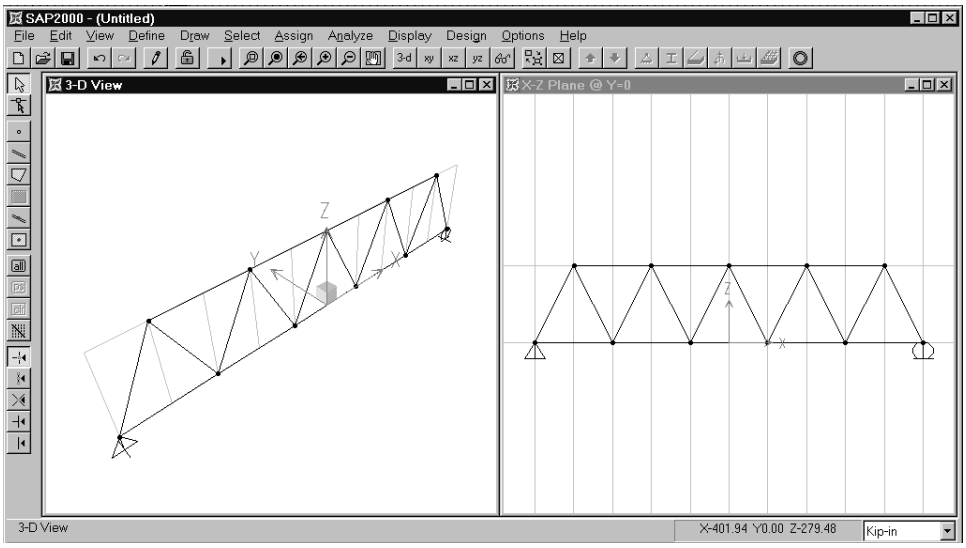
Figure II.1
Geometry and Loading for the Initial Model

Setting Up the Geometry

1. From the **File** menu, choose **New Model from Template...**. This will display the Model Template dialog box.
2. In this dialog box:
   - Click on the Sloped Truss template. This will display the Sloped Truss dialog box.
   - In this dialog box:
     – Change Number of Bays to 5
     – Check the Restraints box
     – Check the Gridlines box
     – Accept the default truss height and bay length
     – Click the **OK** button.



The screen will refresh and display 3-D and 2-D views of the model in vertically-tiled adjoining windows. The right hand window shows the X-Z plane view of the model at Y=0. The left window shows a 3-D perspective view.



This completes the model geometry and includes the support restraints.

Note: Default restraints were added to the model when the Restraints box was checked in the Sloped Truss dialog box.

Defining Structural Sections

We will use two Double Angle sections for the entire truss. We will use the structural sections data file provided with SAP2000. The file is called SECTIONS.PRO and resides in the same directory as the SAP2000 files.

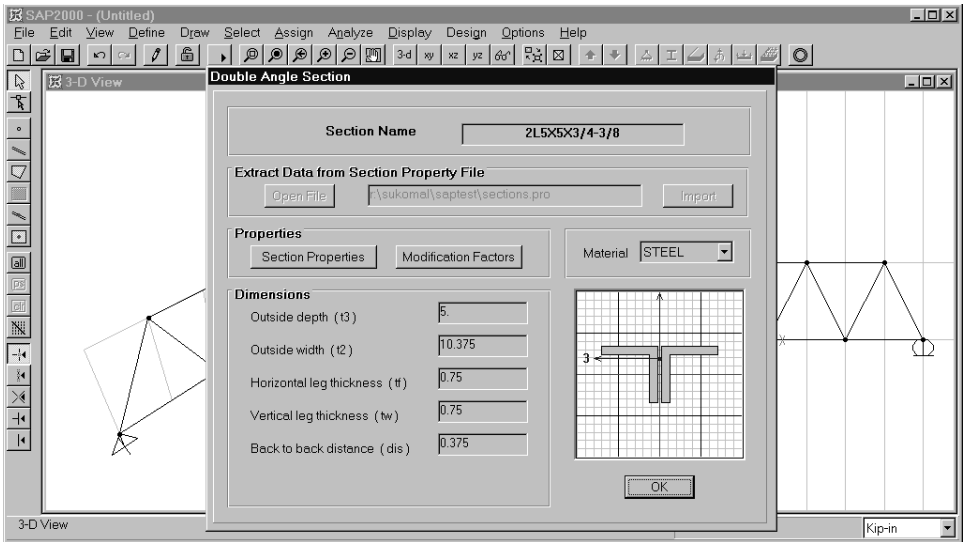
1. From the **Define** menu, choose **Frame Sections...**. This will display the Define Frame Sections dialog box.
2. In this dialog box:
   - Click on the Import drop-down box.
   - Click on Import Double Angle. This will display the Section Property File dialog box.
   - In this dialog box:
     – Locate the SECTIONS.PRO data file
     – Open the SECTIONS.PRO data file by clicking on the **Open** button or double clicking on the file name.

– This will display a drop down list box showing all the Double Angle sections available in the data file.

Note: In a SAP2000 session you have only to locate and open the SECTIONS.PRO data file once. However, you have the option to select another file at any time by choosing **Preferences** *in the* **Options** *menu.*

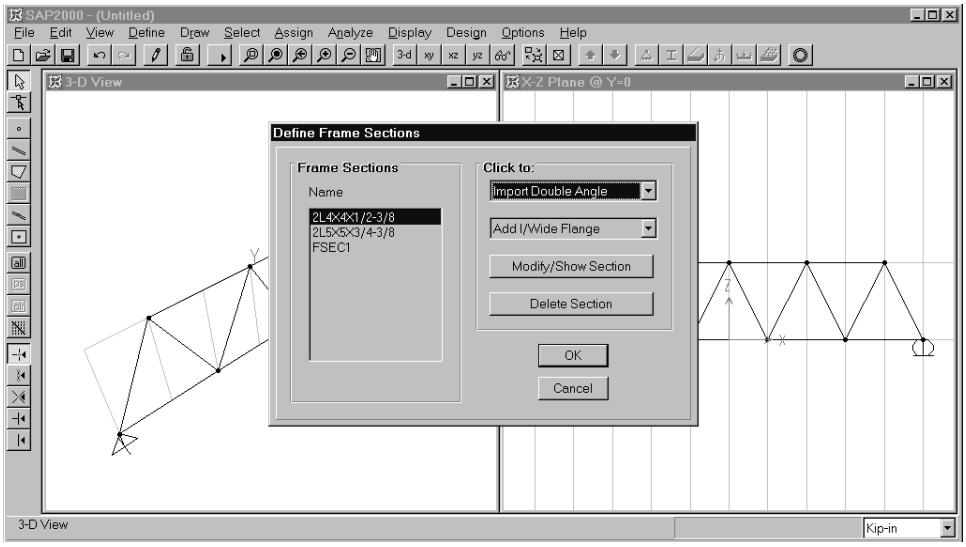
3. In the Double Angle list box:

   - Use Scroll buttons to locate one of the structural sections used in this model, namely 2L5x5x3/4-3/8.
   - Double click on 2L5x5x3/4-3/8. This will display the Double Angle Section dialog box which shows a schematic view of the selected section, section dimensions, and STEEL as the default material type.
   - Click on the **OK** button. This will close the Double Angle Section dialog box. Notice that in the Frame Sections dialog box the label of the selected section (2L5x5x3/4-3/8) is added to the default section name (FSEC1) in the Frame Sections area.



4. Repeat substeps 2 and 3 above to select another double angle, 2L4x4x1/2-3/8.

5. Click on the **OK** button.



Assigning Member Sections

In this step we will assign the two sections previously defined to the various members of the truss. The double angle section 2L5x5x3/4-3/8 is assigned to the top chord and the diagonal members, whereas, the 2L4x4x1/2-3/8 is assigned to the bottom chord members only.

1. We will select the top chord members of the truss in the elevation view by "windowing." To do this:

   - Click on the **Pointer Tool** button on the Side Toolbar.

     Note: We can only select in the SELECTION mode of SAP2000. The alternative mode is the DRAW mode. SAP2000 is usually in the SELECTION mode, which is identified by the pressed **Pointer Tool** *button. To switch from the DRAW mode to the SELECTION mode, you can click on the* **Pointer Tool** *button on the Side Toolbar. If the* **Pointer Tool** *button is already pressed, you do not need to click on it further.*

   - Move the pointer above and to the left of the members you want to select
   - Click and hold the left mouse button
   - While holding, move the pointer below and to the right of the members you want to select. A "rubber-band" window will show the region selected.
   - Release the left mouse button to select all members in this window.

2. We will select the diagonal members by intersection. To do this:
   - Click the **Set Intersecting Line Select Mode** button on the Side Toolbar
   - Move the pointer to the left of the members you want to select
   - Click and hold the left mouse button
   - While holding, move the pointer to the right of the members you want to select. A “rubber-band” will show the intersecting line.
   - Release the left mouse button to select all members that intersect this line.

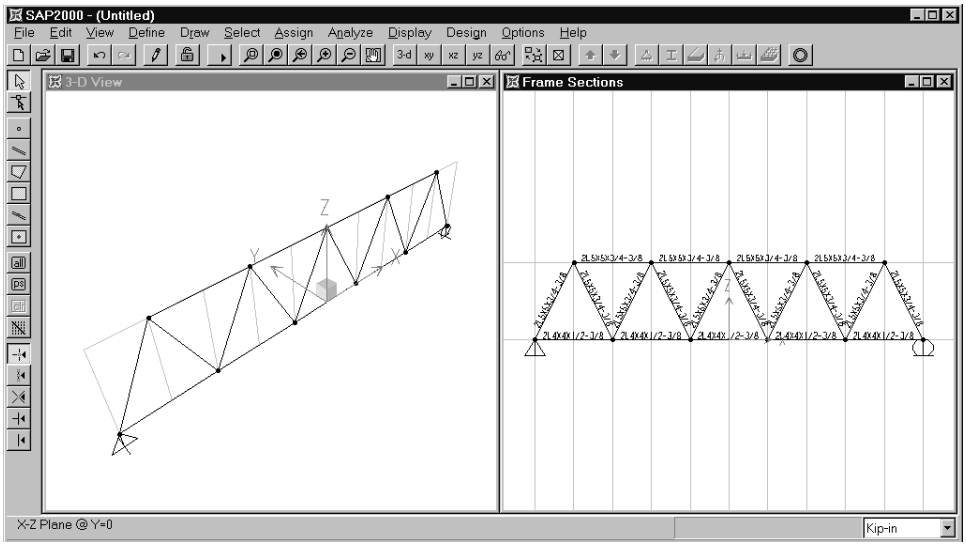
The selection of all the top chord and the diagonal members of the truss is now complete. The selected Frame elements appear as dashed lines.

3. From **Assign** menu, choose **Frame**, then **Sections...** from the submenu. This will display the Define Frame Sections dialog box.
4. In this dialog box:
   - Click on 2L5x5x3/4-3/8 in the Name area.
   - Click on the **OK** button.

The display window in which the selection process was done is refreshed and the section label is displayed on all members.

5. Select the bottom chord members by “windowing” as in substep 1 above.
6. Repeat substeps 3 and 4 to assign 2L4x4x1/2-3/8 to these members.

The screen will now show all the member section assignments.



Note: You can always change the display attributes, e.g. increase the font size, etc. from the **Preferences...** *in the* **Options** *menu.*

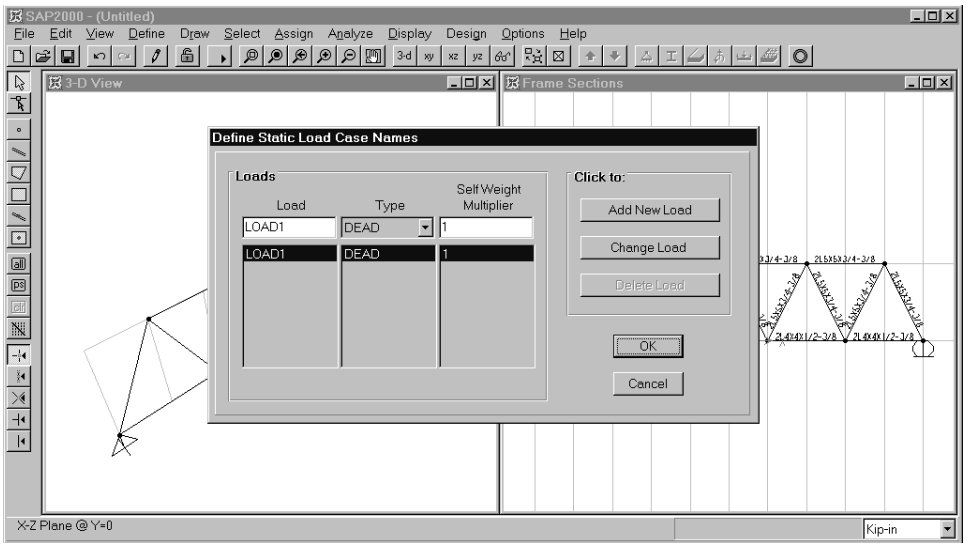
You may turn off the display of the section labels by selecting **Show Undeformed Shape** from the **Display** menu.

Defining Load Cases

Two load cases are considered in the analysis. The first load case is for the dead load which also includes the self-weight of the structure. The program's default name, LOAD1, is accepted for this load case. The second load case is for the live load which we shall name as LOAD2.

1. From the **Define** menu, choose **Static Load Cases...**. This will display the Define Static Load Case Names dialog box.

This will display the default load, LOAD1, with type set to DEAD, and the self-weight multiplier set to unity. We don't have to change anything for this first load case.



Now we define the second load case:

2. In the dialog box:
   - Change LOAD1 to LOAD2.
   - Select LIVE from the Type drop-down list box.
   - Change the self-weight multiplier to zero.
   - Click on the **Add New Load** button. The two load case names and types are now displayed in the Loads list box.
   - Click on the **OK** button.

Assigning Joint Loads

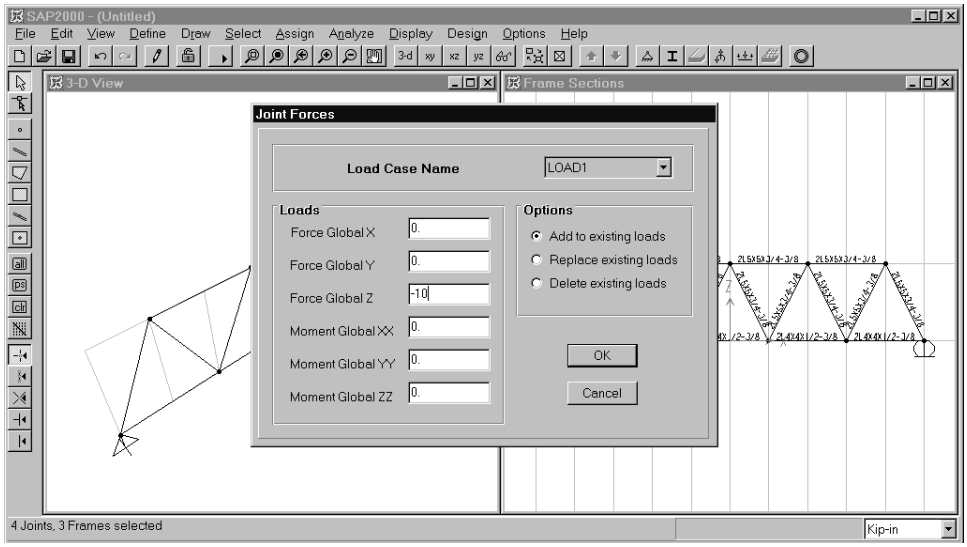
Dead and live loads are applied as joint loads to the bottom chord of the truss. The magnitudes of dead and live joint loads are taken to be –10 kips and –40 kips, respectively.

1. Select the joints on the bottom chord by “windowing” as we did when assigning member sections.

2. From the **Assign** menu, choose **Joint Static Loads...**, then **Forces...** from the submenu. This will display the Joint Forces dialog box.

3. In this dialog box:

- Accept the default load case name as LOAD1.
- Enter –10 in the Force Global Z box in the Loads area.



- Click on the **OK** button.

We will now proceed to enter the live load.

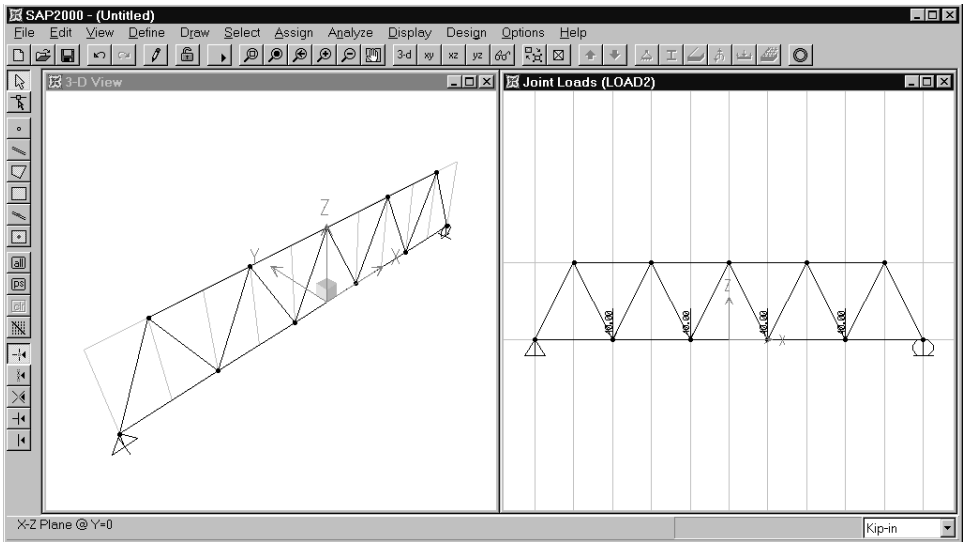
4. Click the **Restore Previous Selection** button on the Side Toolbar. This will re-select the bottom chord joints again.

5. From the **Assign** menu, choose **Joint Static Loads...**, then **Forces...** from the submenu. This will again display the Joint Forces dialog box.

6. In this dialog box:

   - Change the load case name to LOAD2.
   - Enter –40 in the Force Global Z box in the Loads area.
   - Click the **OK** button.

All the loads are now applied to the model, and the model is complete.



Analyzing the Model

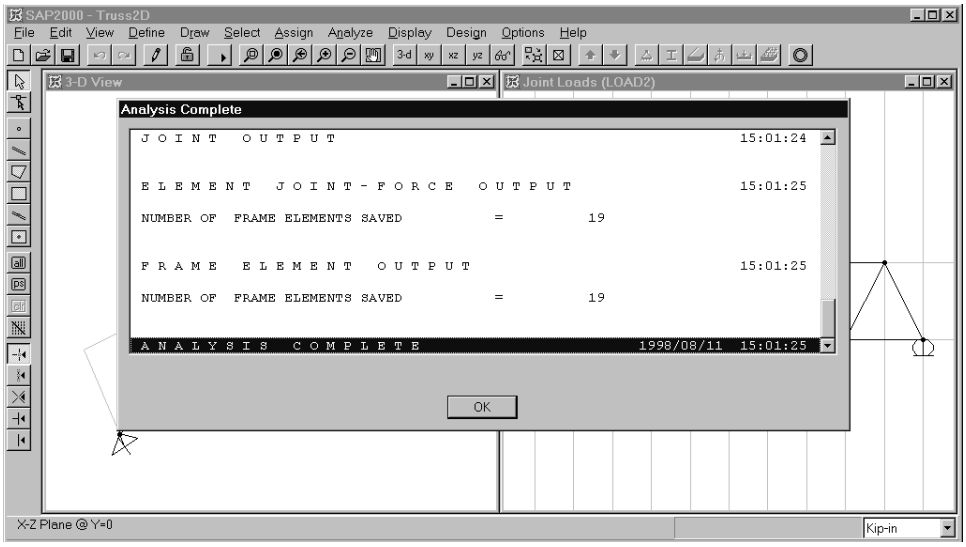
We will now analyze the model.

1. From the **Analyze** menu, select **Run**. This will display the Save Model File As dialog box.
2. In this dialog box:
   - Save the model under a filename. In our case we will call it TRUSS2D.SDB.

     Note: Even if you do not type in the extension .SDB, *the program automatically appends this extension to the filename.*

   - Click on the **Save** button.

A top window is opened in which various phases of analysis are progressively reported. When the analysis is complete the screen will display the following:

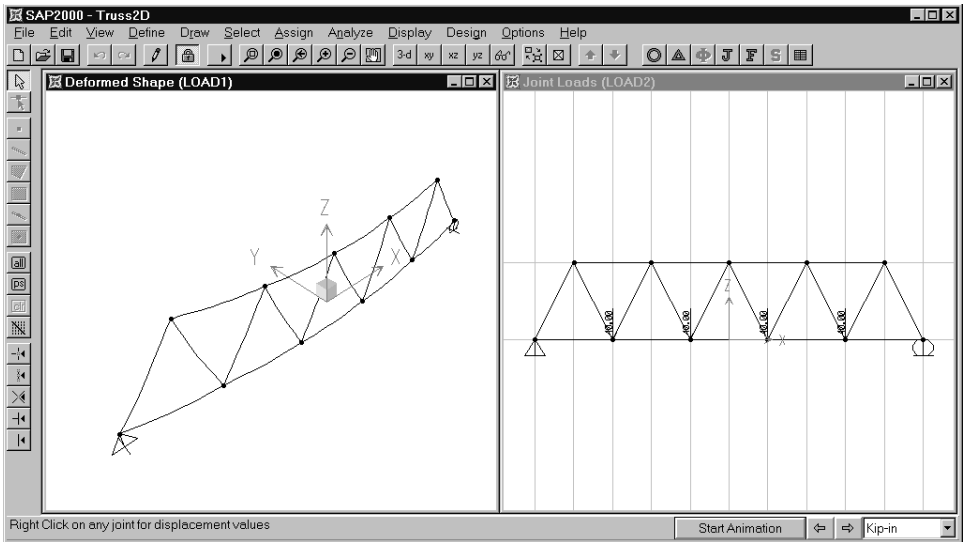


3. Use the scroll bar on the top window to review the analysis messages and to check for any error or warning messages (there should be none).

4. Click on the **OK** button in the top window to close it.

Displaying the Deformed Shape

After the analysis is complete, SAP2000 automatically displays the deformed shape of the model for the default load case, LOAD1, in the window which displays the 3D view, the left window in our case.

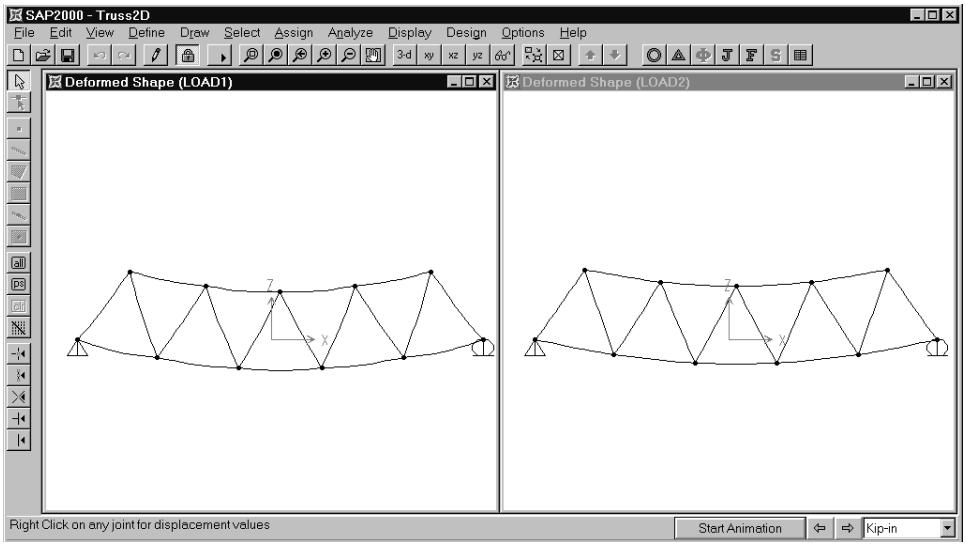


We will now display the deformed shape for the load case LOAD2 in the right window.

1. Click anywhere in the right window to activate this window.
2. Click the **Display Static Deformed Shape** button on the Main Toolbar. This will display the Deformed Shape dialog box.
3. In this dialog box:
   - Select LOAD2 Load Case from the drop down list in the Load area.
   - Click on the **OK** button.

To compare the deformed shape for the two load cases, we will display the deformed shape for load case LOAD1 in 2-D (x-z) view. To do this:

1. Click anywhere in the left window to activate this window.
2. Click on the **2D X-Z View** button on the Main Toolbar to obtain an elevation view of the model.



Note: Results can be printed or saved in a tabulated form by choosing Print **Output Tables... from** *the File menu. Results can also be displayed in a tabulated form by choosing* **Set Output Table Mode...** *from the* **Display** *menu and then by right-clicking on the joint.*

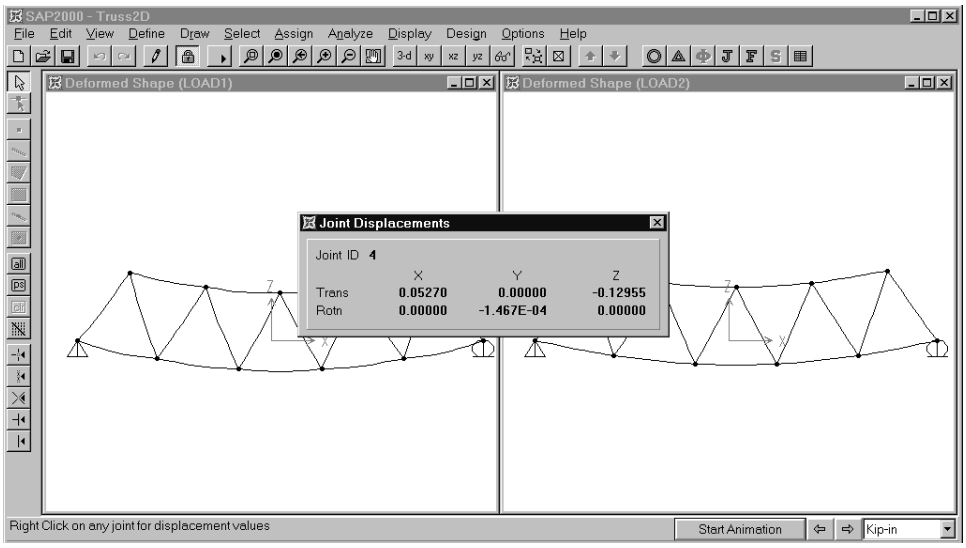
You will observe that the two deformed shapes look similar, even though the loads are different. This is because SAP2000 automatically scales the deflections for display purposes. You can change the scale factors in the dialog box you just used.

You can animate the deformed shape by using the **Start Animation** button at the bottom of the screen. Animation speed is controlled by a horizontal scroll bar that will appear next to this button. Also, you can stop animation by using the **Stop Animation** button.

The left and right arrows at the lower right corner of the screen are used to change the load case that is displayed in the active display window.

We will now select a joint and examine its displacement values for load case LOAD1.

5. Right click on a joint. This will open a floating window in which the values of translations and rotations of the selected joint are displayed. Also, the selected joint will flash on the screen.



Displaying Member Forces

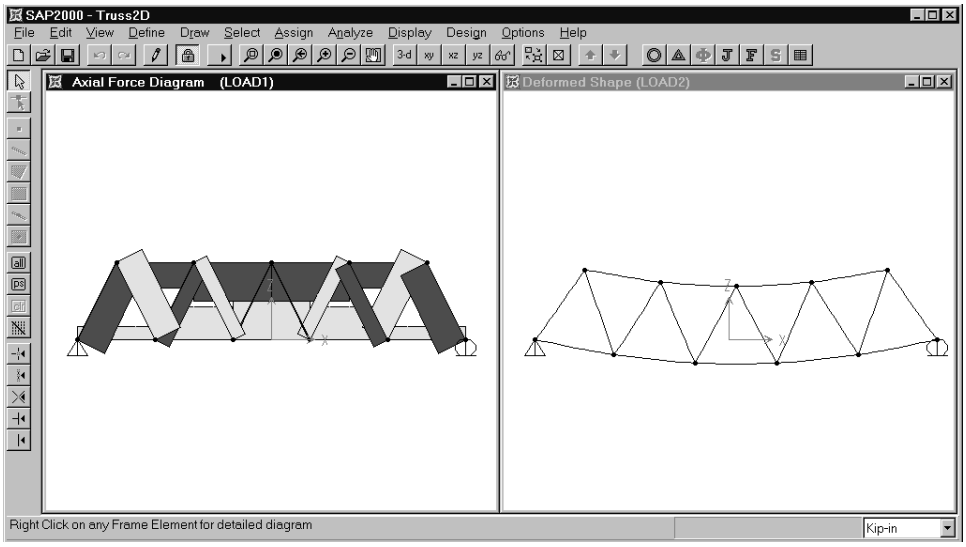
As an example, we will plot the axial force diagram for load case LOAD1 on the left window.

1. Click on the **Member Force Diagram for Frames** button on the Main Toolbar. This will display the Member Force Diagram for Frames dialog box.

3. In this dialog box:

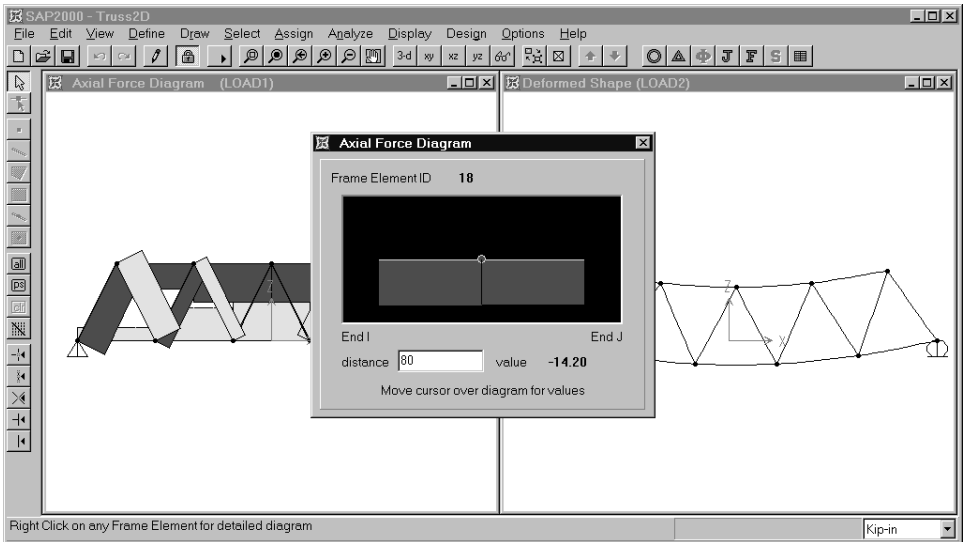
   - Select Axial Force in the Component area.
   - Click the OK button.

The axial force diagram for the entire truss is displayed.



We can now review this force component on a member-by-member basis.

4. Right click on any desired member on the left window. This will open a floating window with title Axial Force Diagram showing the variation of the axial force over the member length. Move the cursor within the floating window to see the numeric values of axial force and the corresponding distances. Click outside this window to close it.



Note: Other element force or stress components can be selected for display in a similar manner.

Note: Results can be printed or saved in a tabulated form by choosing Print **Output Tables... from** *the File menu. Results can also be displayed in a tabulated form by choosing* **Set Output Table Mode...** *from the* **Display** *menu and then by right-clicking on the element.*

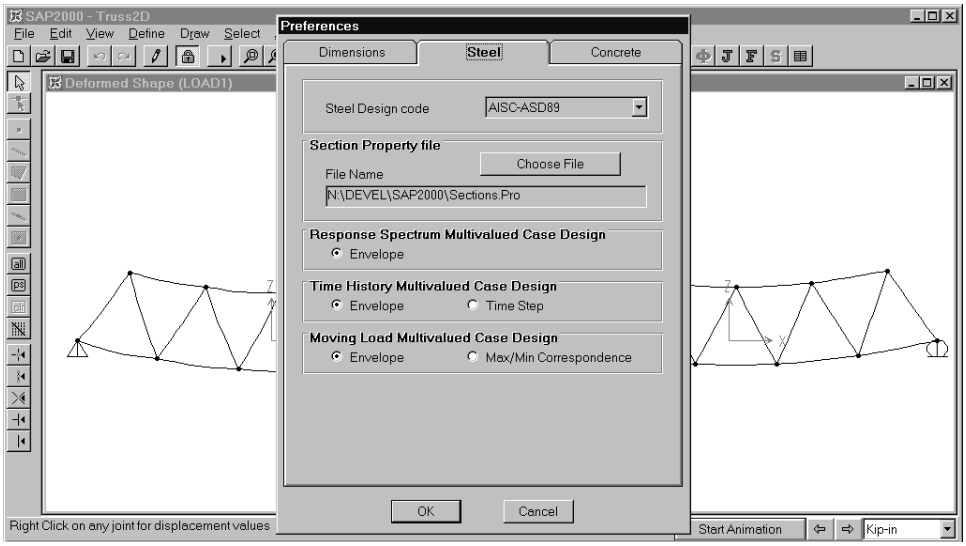
Design Stress Check

The default design code is AISC-ASD89 for design of steel structures. To verify the selected design code, do the following:

1. Click on **Preferences...** from the **Options** menu. This will launch the **Preferences** dialog box.

2. In the dialog box click on the **Steel** tab. You can see the currently selected steel design code, default section property file, and some other options as shown below.

3. You do not need to change anything.

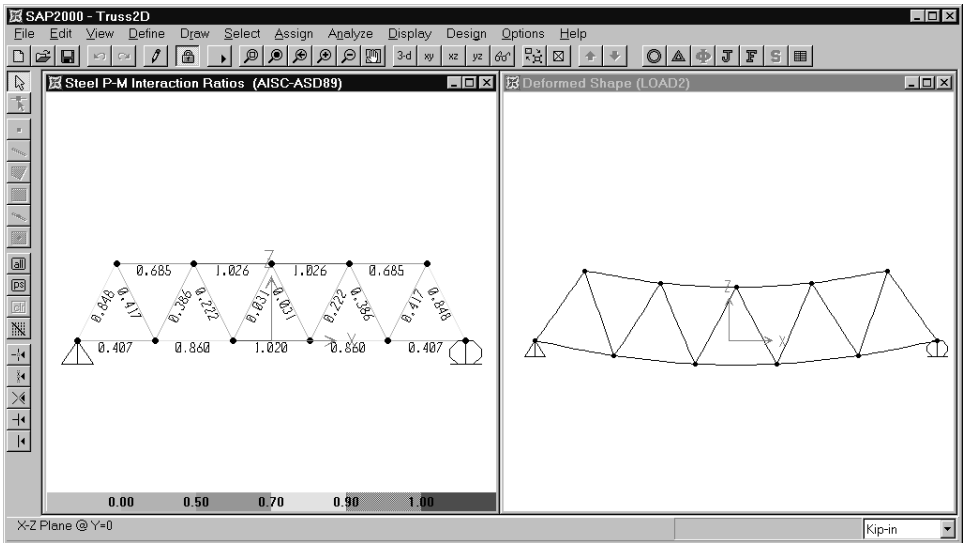


4. Click on the **Cancel** button to close the dialog box.

5. To make sure that SAP2000 will design the steel members, select **Steel Design** menu item from the **Design** menu.

We will now examine member stress ratios using the requirements of AISC/ASD89.

1. From the **Design** menu, choose **Start Design/Check of Structure**. Color-coded stress ratios are then displayed for each member.



Note: The stress ratios, by default, refer to the default design combinations DSTL1 which represent the dead load only and DSTL2 in which the unfactored dead and live loads are added together.

2. Right click on any desired member. This will open a floating window showing the detailed results of the design check, Steel Stress Check Information, at various stations along the element length. You can see the detailed steel stress check information for a station and a specific load combination by clicking on the **Details** button. Click on **OK** to close the window.

SAP2000 allows you to interactively change the design code, member properties, etc. and re-run the design check.

Modifying the Structure

Suppose we needed to modify the truss to support an additional vertical load of 100 kips at the center of the bottom chord. We can do this by adding a vertical member at the center and dividing the central horizontal member into two.

Currently, the model is locked to prevent any changes that would invalidate the analysis and design results we have just obtained. We must first unlock the model, make the desired changes, re-analyze, and finally re-check the design.

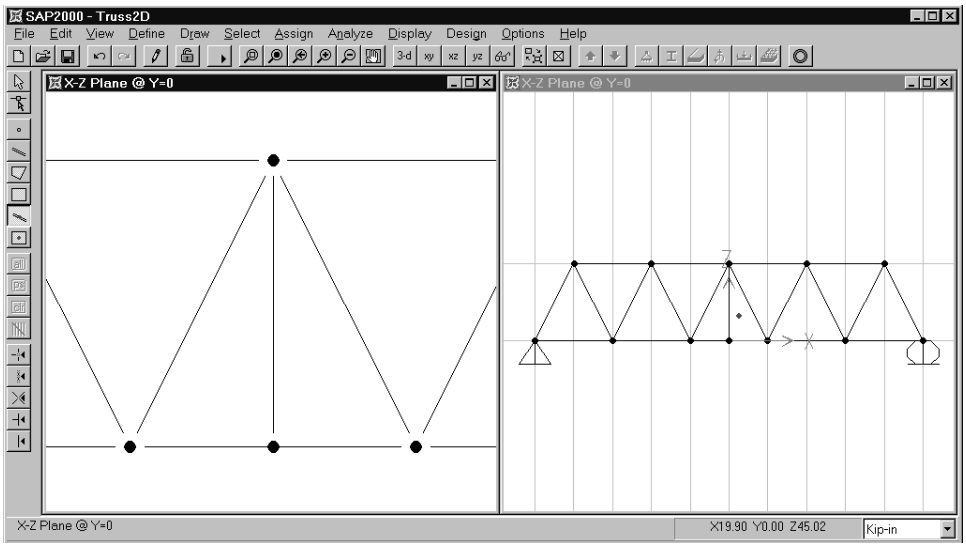
1. Click on the **Lock/Unlock Model** button on the Main Toolbar to unlock the model.
2. You will be warned that unlocking the model will delete all analysis results. Click on OK to acknowledge this.

For illustration purposes, we will use two different methods to draw the new Frame elements. Either method could be used for all of the added elements in this example.

2. Click on the **Quick Draw Frame Element** button on the Side Toolbar, or select it from the **Draw** menu. An element will be drawn each time you click on a grid line.
3. Click the vertical gridline at the center of the model between the top and bottom chords to get the vertical member.

It may now appear that the truss is complete. However, the new vertical member is not actually connected to the horizontal member on the bottom chord. This can easily be seen in a shrunken-element view.

4. Click on the **Element Shrink Toggle** on the Main Toolbar.
5. In order to see more clearly:
   - Select **Show Grid** from the **View** menu to turn off the grid
   - Select **Show Axes** from the **View** menu to turn off the global axes
   - Click on the **Rubber Band Zoom** button on the Main Toolbar
   - In the active Display Window, click and drag a window to enclose the middle part of the structure. The contents of this window will be magnified to fill the Display Window.



We can now see that the middle member of the bottom chord must be broken into two. One way to do this is to use the **Divide Frames** option in the **Edit** menu. We will instead delete the old element and draw two new elements.

6. To delete the old element:

   - Click on the **Pointer Tool** on the Side Toolbar to begin switch to SELECTION mode.
   - Click on the element to be deleted
   - Press the **Delete** key or select **Delete** from the **Edit** menu

7. To draw the two new elements:

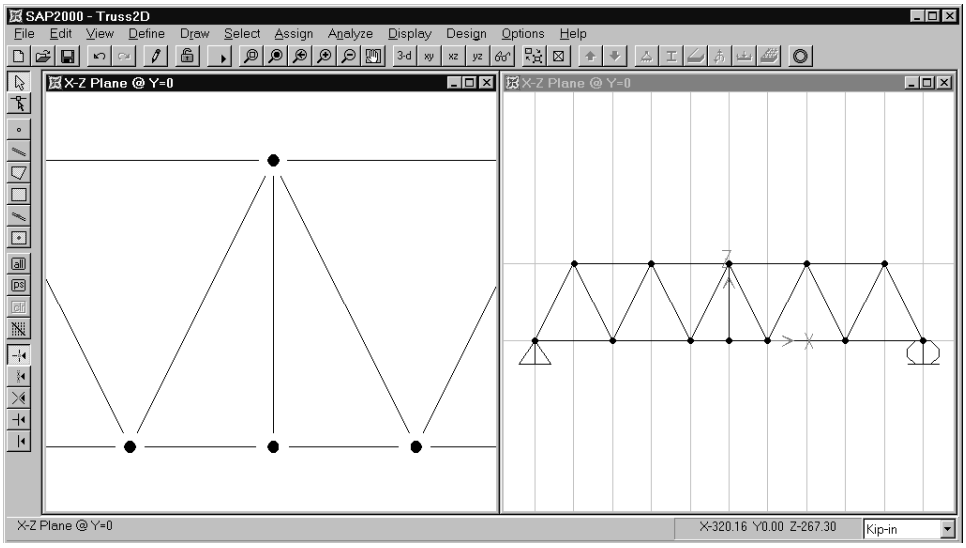
   - Click the **Draw Frame Element** button on the Side Toolbar, or select it from the **Draw** menu.

     Note: You can now draw in either or both windows.

   - Click on the **Snap to Joints and Grid Points** button on the Side toolbar
   - Click on the leftmost of the three middle joints. The start of first element will snap to the exact joint location.
   - As you move the mouse, a “rubber band” will show where the first element will be drawn. Click on the middle joint to draw it.
   - As you move the mouse, you can see that the next element will start where the last element ended.

- Double click on the rightmost of the three middle joints to end the second element. Double-clicking ends “rubber-banding” until the next single click.

8. Click on the **Pointer Tool** to end draw mode.



The remaining steps are a repetition of what we did for the original model

- Select the new Frame elements and assign Frame Sections as desired.
- Select the new joint at the bottom by clicking on it and assign a joint load of –100 kips in load case LOAD2.
- Click on the **Restore Full View** button from the Main Toolbar.
- Analyze the structure
- Display the member forces
- Check the design

At this point you may want to consider changing the section properties for those Frame elements that are under-designed. This would require unlocking the model again, selecting Frame elements, assigning Frame Sections, then performing another analysis and design check.

Concluding Remarks

This marks the end of the quick tour of SAP2000. The intent has been to highlight and demonstrate a few of the basic features. Feel free to experiment and explore other options. Additional information is available within the Help menu.

Chapter III

Concrete Design Tutorial

Overview

Several of the basic features of SAP2000 concrete design are explored in this tutorial. This introduction is aimed at giving the first time user hands-on experience designing concrete frames with SAP2000. The program allows you to select from several U.S. and international codes to design and review concrete structures. A comprehensive on-line Help is included in the program for your quick reference. It is assumed that you have a working knowledge of concrete design procedures and are reasonably familiar with the current codes of practice and their underlying design concepts.

We will access the SAP2000 commands from both the Main Toolbar and the Side Toolbar and from the menus. The Toolbars, however, provide quick access to most commonly used features available from the menus.

In the assignment sequence, there are two important points you must remember. First, you have to define an entity before you can assign an attribute to it, and second, you have to select member(s) before you can assign new attributes or modify old ones.

Description of the Model

The structure is a two-story, two-by-two-bay office building located in Seismic Zone No. 4 (high seismic area). It is designed as a special moment resisting concrete frame using the ACI 318-95 code.

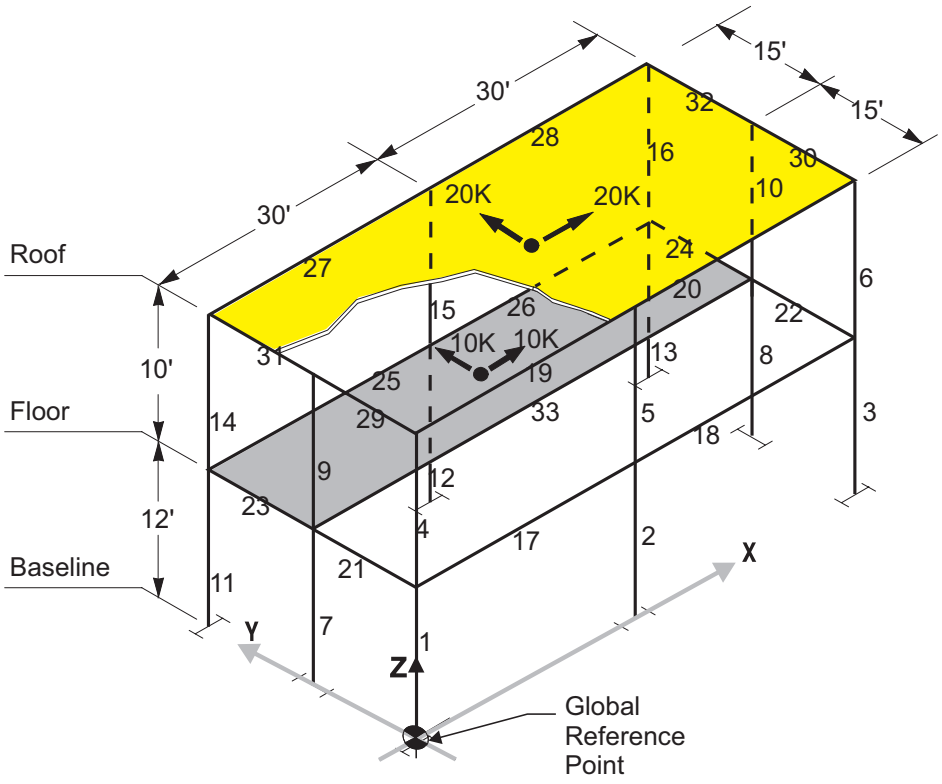


Figure III-1
Ductile Moment Resisting Concrete Frame (Tutorial Example)

Geometry

The two-story structure has a partial floor diaphragm and a full roof diaphragm. See Figure III-1. The story height of the top and bottom floor is taken as $10'0''$ and $12'0''$ respectively. The initial member sizes and reinforcement are given in Table III-1.

ID	Structural Component	Description
1	Typical columns at the top story	$12'' \times 12''$, Rebar not specified, $2''$ cover to center of steel
2	Typical columns at the bottom story	$18'' \times 18''$, Rebar not specified, $2''$ cover to center of steel
3	All beams other than Beam 33	$12'' \times 24''$, Rebar not specified, $2''$ cover to center of steel
4	The longest beam (Beam 33)	$12'' \times 36''$, Rebar not specified, $2''$ cover to center of steel

Table III-1
Structural Property Data (Tutorial Example)

Material Properties

The properties of the materials used in the model are given in Table III-2. It is assumed that the materials used for the beams and columns are the same. However, the shear reinforcement is different from the longitudinal reinforcement.

Material Property	Magnitude
f_c'	4 ksi
E_c	3600 ksi
f_y	60 ksi
f_{ys}	40 ksi

Table III-2
Material Specifications (Tutorial Example)

For analysis in SAP2000, the value of E_c is modified to account for cracking. A multiplier of 0.4 is used for columns assumed to have about 2% steel, and a multiplier of 0.5 is used for the beams. These multiplication factors are slightly different in ACI 318-95. See Section R10.11.1 of ACI 318-95.

Load Cases

Four load cases are considered in the analysis. The dead and live loads are defined as load cases DL and LL respectively. The lateral seismic loads, in turn, are designated as QX and QY respectively.

The dead and live loads are simplified as line loads on the beams. The equivalent static seismic forces are applied as lateral loads at the centroids of the diaphragms:

Load case 1 : DL — 1.0 kip/ft on all beams which are connected to the diaphragm along the X-direction (Self-weight included)

Load case 2 : LL — 0.5 kip/ft on all beams which are connected to the diaphragm along the X-direction

Load case 3 : QX — Static equivalent earthquake force in the X-direction

Load case 4 : QY — Static equivalent earthquake force in the Y-direction

Analysis

Two diaphragm constraints are applied for the two diaphragms at the two floors. These constraints prevent in-plane relative displacements of the nodes at each floor. The lateral earthquake loads are assumed to be applied at the centroid of the diaphragm. A P-Δ analysis is carried out with a load level of 0.75 (1.4 DL + 1.7 LL)/φ as recommended in the chapter "Design for ACI 318-95" of the *SAP2000 Concrete Design Manual*, where φ is taken as 0.75.

Design

The design is performed in accordance with ACI 318-95. Kip-in units are used for the model. The input database file for this model is **EXCONC.SDB**. This is supplied as part of the SAP2000 package.

Starting the Tutorial

A step-by-step procedure for the design of the model is outlined below. It is recommended that you actually perform these steps while reading this chapter. We assume that you have successfully started the program. You can do this by running SAP2000 from the Start Menu.

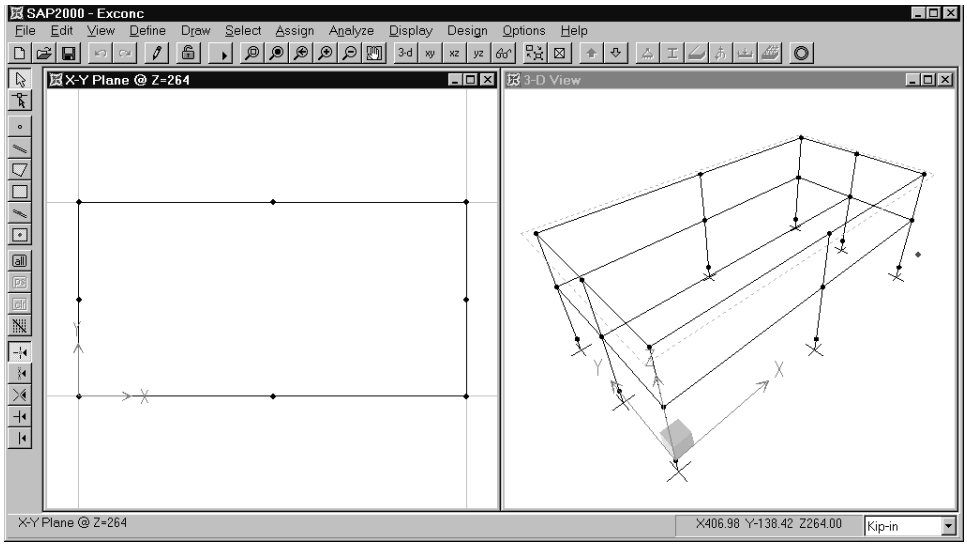
In this tutorial, whenever possible, we will use the Toolbars to access various options quickly. Most of the features available on the Toolbars can also be accessed from the menus. Use the on-line Help or refer to the *SAP2000 Getting Started* manual for a detailed description of the SAP2000 screen.

The input database file for the model (**EXCONC.SDB**) is in the EXAMPLES subdirectory under the main directory where the program has been installed. In this example, the analysis model is already created. This tutorial gives the highlights of the design phase. You are assumed to be familiar with creating and editing structural models using SAP2000.

Opening the Model Database File

1. Click on the **Open** button from the **File** menu. This will display the **Open Model File** dialog box.
2. In this dialog box:
   - Select the **EXCONC.SDB** file.
   - Click on the **Open** button.

The screen will now show two vertically-tiled windows. The left window displays a plan view of the model at level +264 inch. Section labels are not displayed in this view. A three-dimensional view of the model is shown in the right window. The units and the cursor location are displayed at the bottom of the window.



Note: when working with multiple windows, clicking anywhere in a particular window will activate that window.

Before we proceed further, we will make a copy of the data file by saving the model under a new name, say, TUTOR1.SDB. We will use the copy during the tutorial and leave the original file unaltered.

3. From the **File** menu, choose **Save As...**. This will display the **Save Model File As** dialog box.
4. In this dialog box:
   - Enter new filename, Tutor1.SDB.

     Note: Even if you do not type in the extension .SDB, the program automatically appends this extension to the filename.
   - Click on the **Save** button.

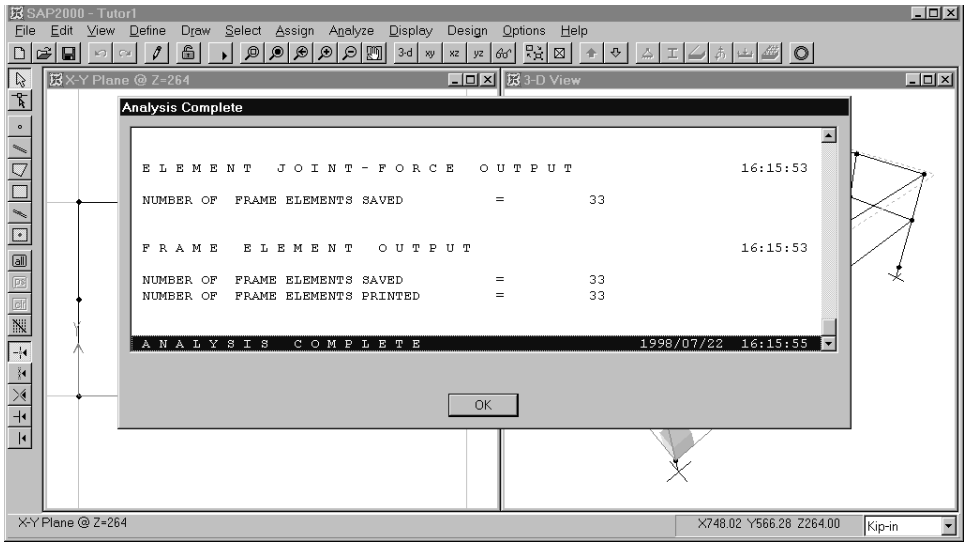
The new name is displayed in the Main Title Bar.

Analyzing the Model

We will now analyze the model. Before analyzing the model we need to set the P-Δ force and other parameters for P-Δ analysis. To do this:

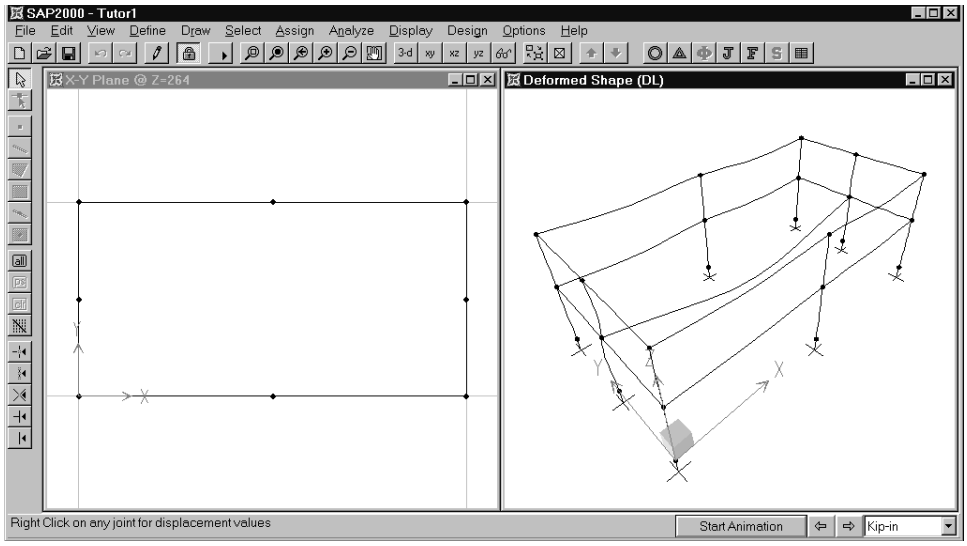
1. Select the **Set Options...** button in the **Analyze** menu. This will immediately bring up the **Analysis Options** dialog box. In this dialog box:
   - Check the **Include P-Delta** box.
   - Click on the **Set P-Delta Parameters** button. This will bring up the **P-Delta Parameters** dialog box. In this dialog box:
     – Set maximum iterations to 5.
     – Change the DL scale factor to 1.4 and click **Modify**.
     – Click on the **Load Case** drop down arrow.
     – Select LL.
     – Change the LL scale factor to 1.7 and click **Add**.
     – Click **OK** to close the **P-Delta Parameters** dialog box
   - Click **OK** to close **Analysis Options** dialog box.
2. Click on the **Run Analysis** button on the Main Toolbar.

A top window is opened in which various phases of analysis are progressively reported. When the analysis is complete the screen will display the following:



3. Use the scroll bar on the top window to review the analysis messages and to check for any error or warning messages. In our case there should be none.

4. Click on the **OK** button in the top window to close it. This will display a deformed shape for the first load case (DL) in the active window (right window in this example) as follows:



Selecting the Design Code

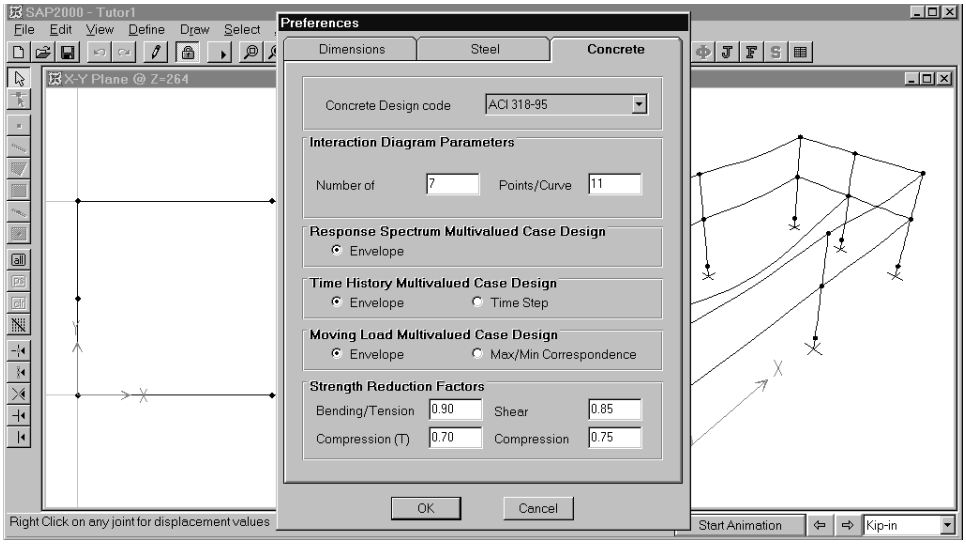
Selection of a design code is activated from **Preferences...** in the **Options** menu. The default design code is ACI 318-95 for reinforced concrete design. Since the default code is used in this tutorial, we can by-pass this step. To confirm, however, you can follow this:

1. Click on the **Preferences...** button from the **Options** menu. This will launch the **Preferences** dialog box.

2. Click on the **Concrete** tab.

3. You can see the currently selected concrete design code, strength reduction factors, interaction diagram parameters, and other parameters as shown in the following screen. You do not need to change anything.

4. Click on the **Cancel** button to close the dialog box.

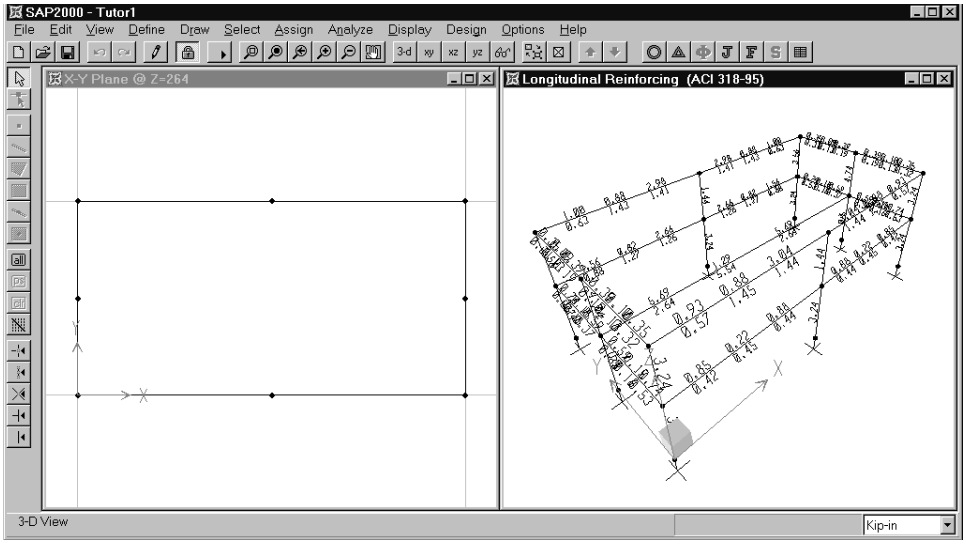


5. To make sure that SAP2000 will design the steel members, select **Concrete Design** menu item from the **Design** menu.

Starting Design

With the analysis phase and selection of the design code completed, we will now design the structure using the requirements of ACI 318-95.

1. From the **Design** menu, choose **Start Design/Check of Structure**. The program now designs each of the concrete frame members. (If we had selected some frame members, then only the selected frames would be designed). In a few moments the longitudinal reinforcement requirements are displayed in the active window. For beams the compression and the tensile reinforcement are displayed separately. For columns the total overall reinforcement area is displayed. In the display, the reinforcement areas are reported for the governing design combination.

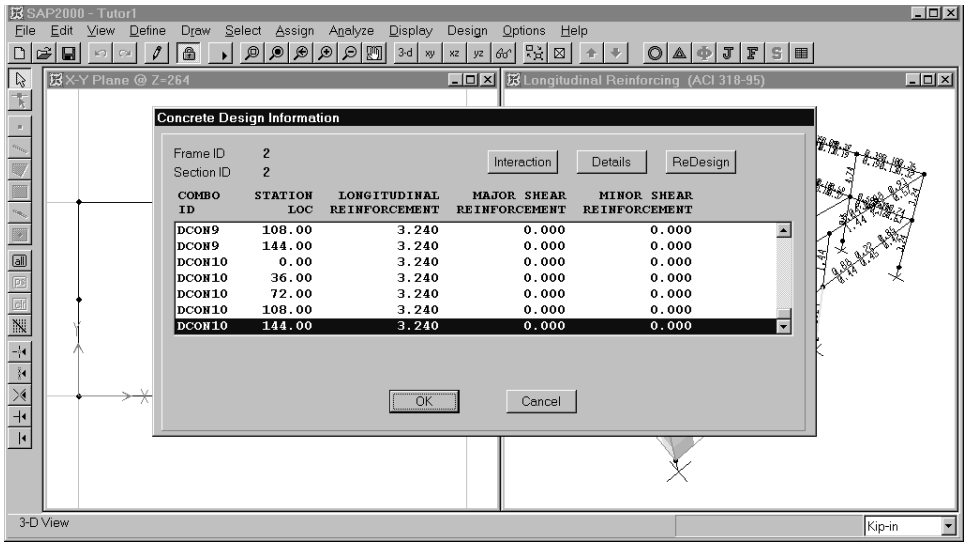


Note: *Since no load combinations were defined in the model, SAP2000 design automatically provided a set of design load combinations for the selected design code (**ACI** for **Concrete**). The default load combinations can be added by any of the three following ways.*

- *Clicking **Add Default Design Combo** once in the **Define Load Combinations** dialog box launched from the **Define** menu by clicking **Load Combinations....***
- *Or clicking **Select Design Combos...** from the **Design** menu when there are no design load combinations defined in the model.*
- *Or clicking **Start Design/Check of Structure** from the **Design** menu when there are no design load combinations defined in the model.*

2. Right click on a column member, for example element 2 (see Figure III-1). This will open the **Concrete Design Information** dialog box showing longitudinal and shear reinforcement requirements at various stations along the element length for the various load combinations (see screen that follows). The dialog

box also can show information regarding the **Details** of calculation for design, the element overwrite assignments for **ReDesign** for the selected member, and column **Interaction** properties. However, if the member is a beam, rather than a column, the **Interaction** properties are not relevant and are not available from the **Concrete Design Information** dialog box.

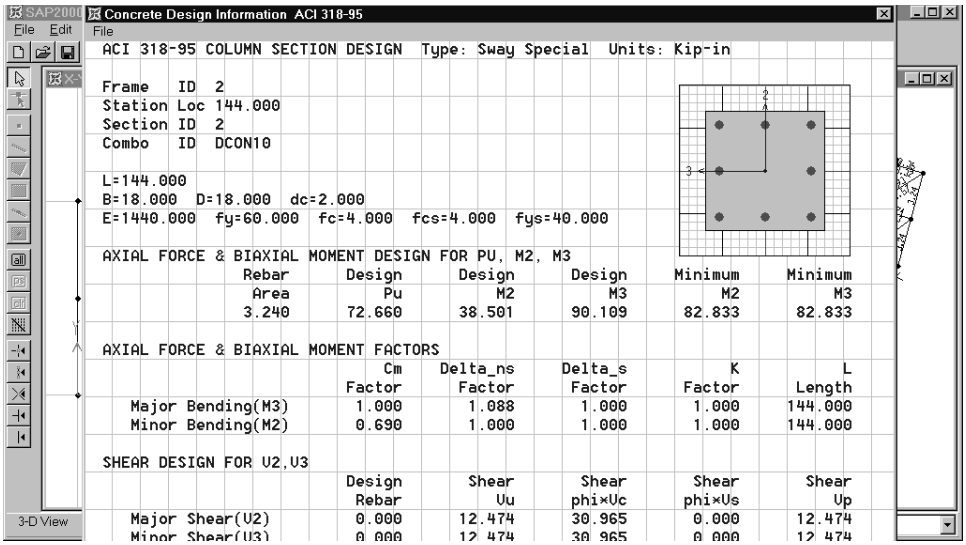


3. In this dialog box:

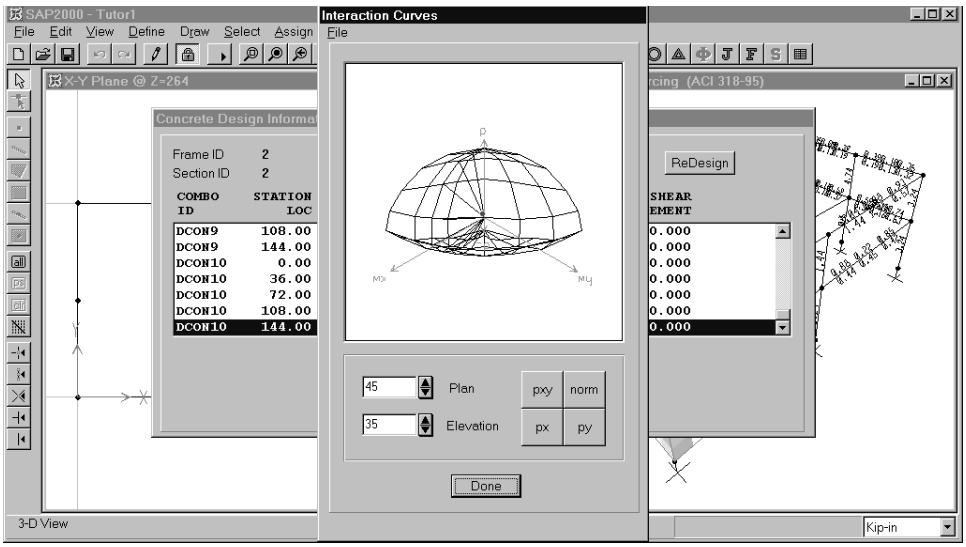
   - Select a design check station in the **Concrete Design Information** dialog box.

     Note: The number of stations (number of segments + 1) used in the design is set by the user through **Frame** *and* **Output Segment** *buttons from the* **Assign** *menu prior to the analysis phase. The default number of segments is 4 for beams and 2 for columns.*

   - Click on the **Details** button. This will open the **Concrete Design Information ACI 318-95** screen showing the design parameters including the reinforcement areas and the factored member forces for the selected load combination at that particular station. See the following screen.

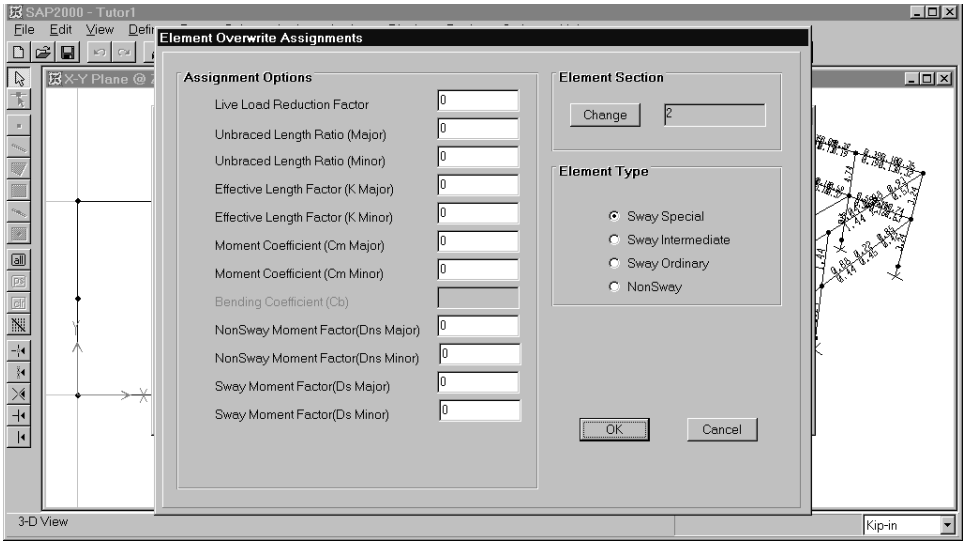


- Close the **Concrete Design Information ACI 318-95** window.
- Click on the **Interaction** button. This will open the dialog box showing the column interaction diagram and the current state of the design forces in the diagram for the selected load combination at that particular station. The interaction diagram can be rotated about any axis to view the diagram from different directions. See the follwing screen.



- Click on the **Done** button to close the **Interaction** information dialog box.

- Click on the **ReDesign** button. This will open the **Element Overwrite Assignments** information dialog box showing the input design factors including the K factors, C_m factors, etc. These factors can be edited for redesigning. See the following screen. There is also an alternative way of editing the properties of a set of members which will be demonstrated in the next section "Changing Member Properties".



- Click on the **Cancel** button to close the **Element Overwrite Assignments** dialog box.
- Click on the **Cancel** button to close the **Concrete Design Information** dialog box.

Till now we have analyzed and designed the concrete frame and reviewed some of the design information. SAP2000 allows you to interactively change the design code, member properties, remove or add new load combinations, etc. and re-run the analysis and design phases. As a demonstration in this tutorial, we will edit/change a member property for a set of frame members in the next section.

Changing Member Properties

With the analysis and preliminary design successfully completed, we will now modify the section properties of all the columns of the bottom story before performing re-analysis. Initially, in the analysis, the section type of each column member of the bottom story was taken to be 2. Referring to the screen that follows, we will

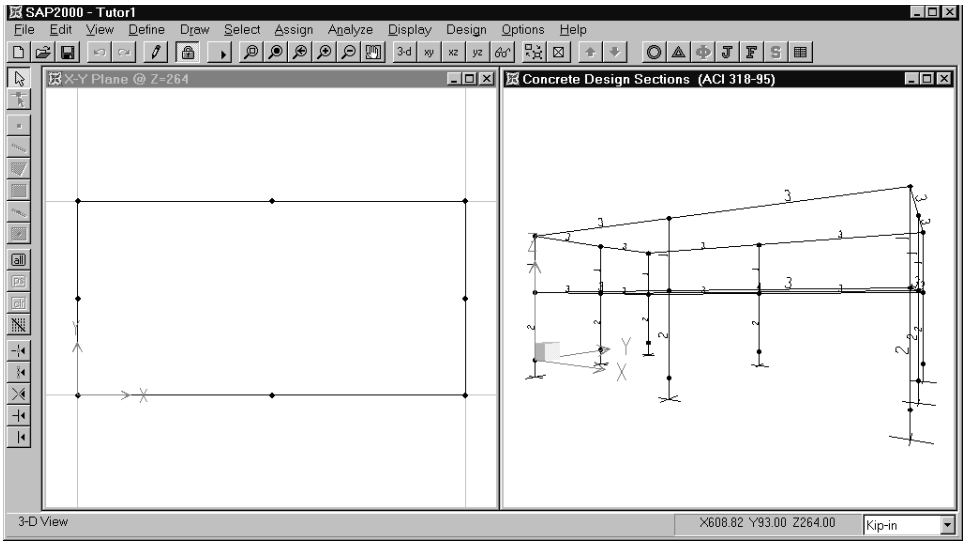
change the section type of every column at the bottom story to be 1. Note that there are already four previously defined section types in the model which were named numerically as 1, 2, 3, and 4. In order to make these changes, we will change the view in the right window to make all the columns visible for selection. Notice that this window is currently showing the longitudinal reinforcing from the previous design.

1. Click on the **Show Undeformed Shape** button from the Main Toolbar.
2. Click on the **2D View** (**xz**) button from the Main Toolbar for an elevation view.
3. Click on the **Perspective Toggle** button from the Main Toolbar. This will display a 3D view. All columns except the middle two will be visible. These two columns will be overlapping each other. To look at them better, we need to rotate the model about a vertical axis.
4. Click on the **Set 3D View ...** button on the **View** menu. Increase the **plan View Direction Angle** from 270 to 300 on the **Set 3D View** popup window and then click on the **OK** button.

Now, with all the columns visible, we can select and modify their design section information. Remember, SAP2000 maintains two sets of information for sections. One is for analysis and the other is for design. Changing section type here will affect the design section only. To update the analysis section, you need to explicitly request an update of the analysis information from the current design state using the menu item **Update Analysis Sections** in the **Design** menu.

5. To see the current setting of Design Sections do the following:
   - Click on the **Display Design Info ...** menu item from the **Design** menu. Select the **Design Input** option button.
   - Select **Design Sections** from the drop-down list.
   - Click **OK**.

This will display the design sections on the screen as shown below. Now we can select and modify the sections for the columns at the first story.

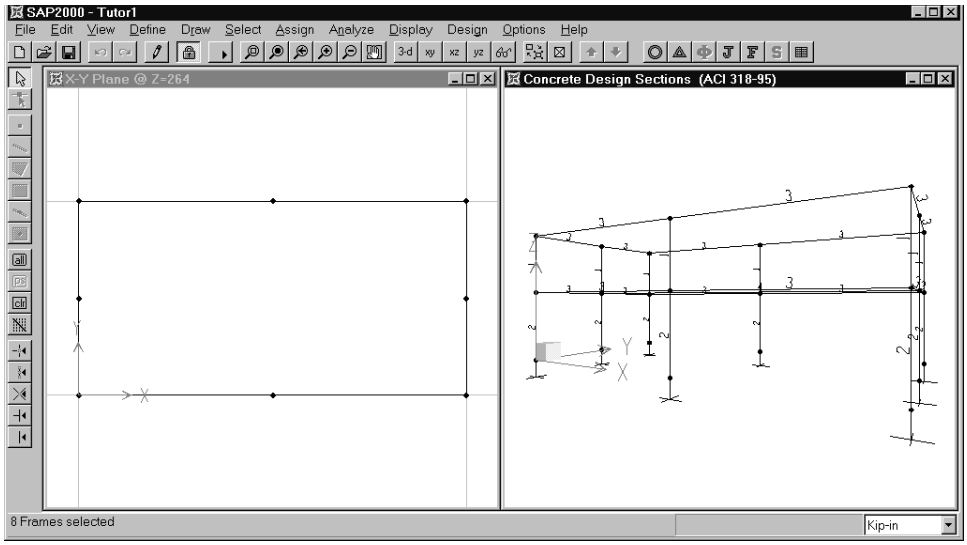


6. To select all the columns at the bottom story do the following:

   - Click the **Set Intersecting Line Select Mode** button on the Main Toolbar.
   - Move the pointer to the left and middle of the leftmost corner column at the bottom story.
   - Click and hold down the left mouse button.
   - While holding, move the pointer horizontally to the right of the members intersecting all the columns at the bottom story. A “rubber-band line” will show the intersecting line.
   - Release the left mouse button to select all members that intersect the rubber-band line.

     Note: To select all the columns in the bottom story, we have to do this operation only once. Any member can also be selected just by clicking the member itself.

The selection of all the bottom story columns is now complete. The selected members appear as dashed lines.

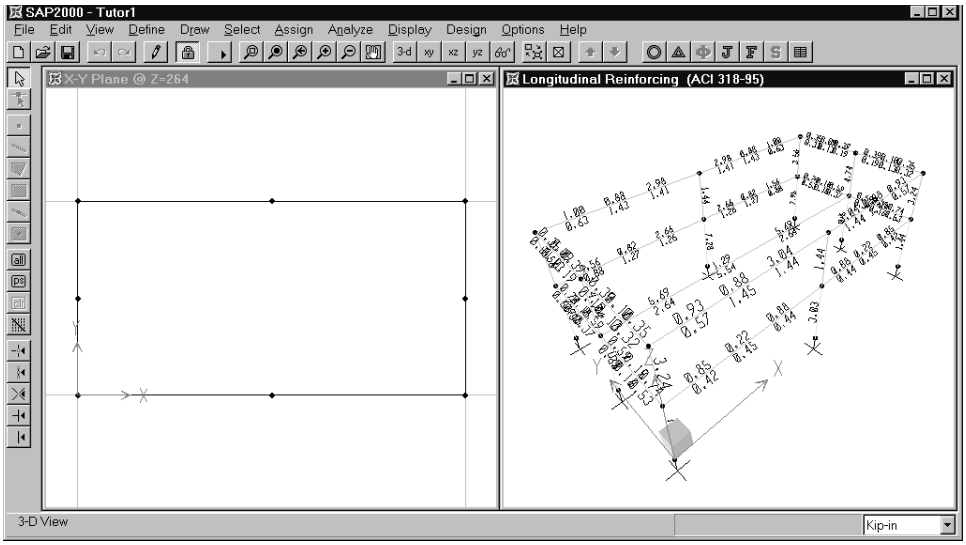


7. From the **Design** menu, choose **ReDefine Element Design Data....** This will display the **Element Overwrite Assignments** dialog box to edit the sections and the design factors. The design factors are code dependent. To change the sections from this dialog box:

   - Click the **Change** button on the **Element Section** area. This will display **Select Sections** dialog box. In this dialog box:
     - Select 1 by clicking once.
     - Click on the **OK** button to accept the change.
   - Click on the **OK** button on **Element Overwrite Assignment** dialog box. This will recompute the longitudinal reinforcement based on the new section properties and the previous analysis results.
   - Click on the **Refresh Window** button on the Main Toolbar.

8. To see the recomputed longitudinal reinforcement, do the following:

   - Click on the **Display Design Info ...** menu item from the **Design** menu. Select the **Design Output** option button.
   - Select **Longitudinal Reinforcing** from the drop-down list.
   - Click **OK**. This will display the longitudinal reinforcement as recomputed based on the new section properties and the previous analysis results.
   - Click on the **3D View (3-d)** button from the Main Toolbar to display the results in an orientation used earlier.



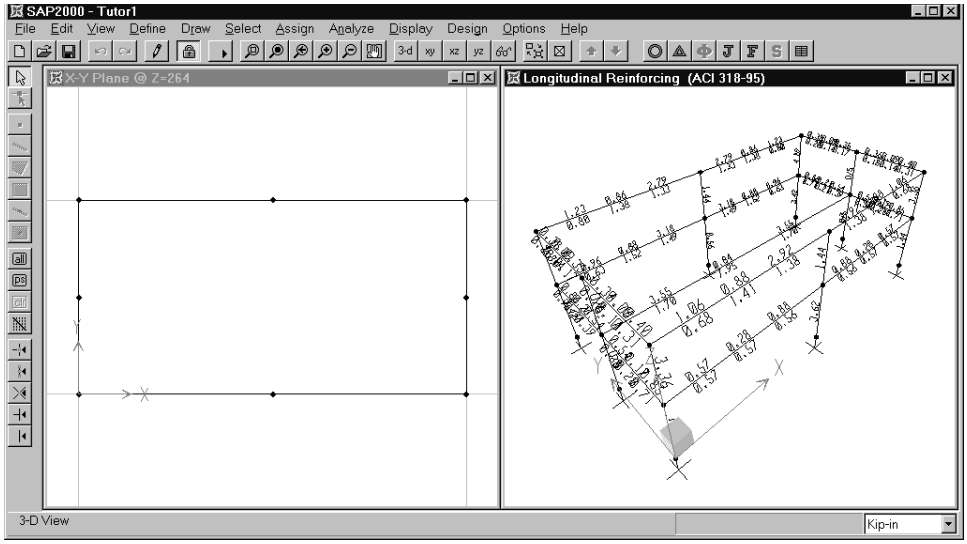
Notice that as a result of changing the section, the reinforcement areas in those particular columns are changed. To see the difference, compare this display with the one on page 35.

It is important to realize that changes made to member section properties in the design phase are **not** automatically reflected in the analysis results. These changes are only local to the post-processing phase unless a re-run of the analysis, with updated elements, is requested by the user. In other words, overwriting the section properties only affects the stress values and not the factored element forces obtained in the analysis preceding such changes. The redistribution of member forces due to change of stiffness (revision of section properties) is effected in a re-run of analysis. The section properties can only be updated on a set of selected elements. To re-analyze and re-design, do the following:

- Click on the **Restore Previous Selection** from the Side Toolbar to re-select the previously selected frame elements.
- In the **Design** menu click on **Update Analysis Sections**. This will prompt a dialog box asking **"Updating Analysis Section will unlock model! OK to update?"**. Click **OK**.
- From the **Analyze** menu, choose **Run.** This will immediately start the analysis procedure. A top window is opened in which various phases of analysis are progressively displayed. The results will obviously differ from those produced in the initial analysis because of the change of section properties we made in the design stage. Click **OK** to close the top window.

- Click on the **Start Design/Check of Structures** from the **Design** menu. This will redesign the structure and display the new required longitudinal reinforcement.

You can see the difference after re-running the design based on the latest analysis results.



Concluding Remarks

We have come to the end of this tutorial on the SAP2000 concrete design options. The intent has been to highlight and demonstrate a few of the basic features in order to open up the path for you to explore and use the more advanced options. For more information on various topics consult the on-line **Help** provided with the program.

Chapter IV

Steel Design Tutorial

Overview

Several of the basic features of SAP2000 steel design are explored in this tutorial. This tutorial is aimed at giving the first time user hands-on experience. The program allows you to select from several U.S. and international codes to stress check and design a steel structure. It is assumed that you have a working knowledge of steel design procedures and are reasonably familiar with the current codes of practice and their underlying design concepts. A comprehensive on-line Help is included in the program for your quick reference.

We will access the SAP2000 commands from both the Main Toolbar and the Side Toolbar and from the menus. The Toolbars, however, provide quick access to most commonly used features available from the menus.

In the assignment sequence, there are two important points you must remember. First, you have to define an entity before you can assign an attribute to it, and second, you have to select member(s) before you can assign new attributes or modify old ones.

Description of the Model

The structure is a two-story, two-by-three-bay office building. The frame will be designed in accordance with the AISC-LRFD93 code. Earthquake induced force is considered in the analysis and design of this frame. However, special requirements for the design of moment resisting ductile steel frames are not currently implemented in SAP2000.

Geometry

The moment frame is of structural steel as shown in Figure IV-1. The second floor has metal deck and lightweight concrete fill, whereas the roof has metal deck only. Typical story height is taken as $13'0''$. The initial member sizes are given in Figure IV-1.

Material Properties

Material specifications are:

Beams and girders:	ASTM A36	($F_y = 36$ ksi)
Columns:	ASTM A572	($F_y = 50$ ksi)

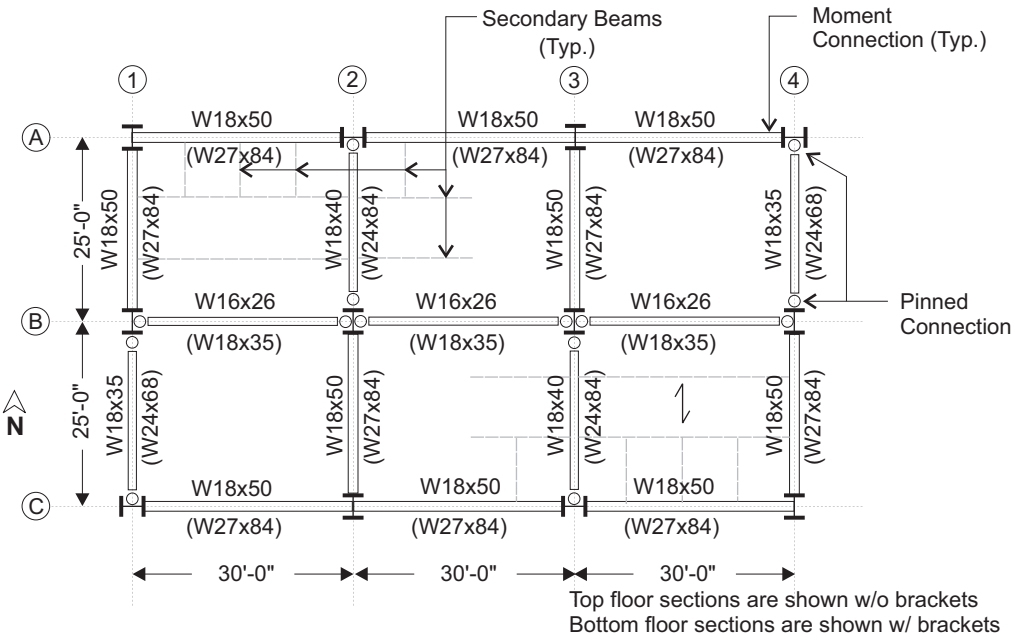
Load Cases

Four load cases are considered in the analysis — DL, LL, QX, and QY. The dead and live loads are defined as the load cases DL and LL respectively. The lateral seismic loads, in turn, are designated as the load cases QX and QY.

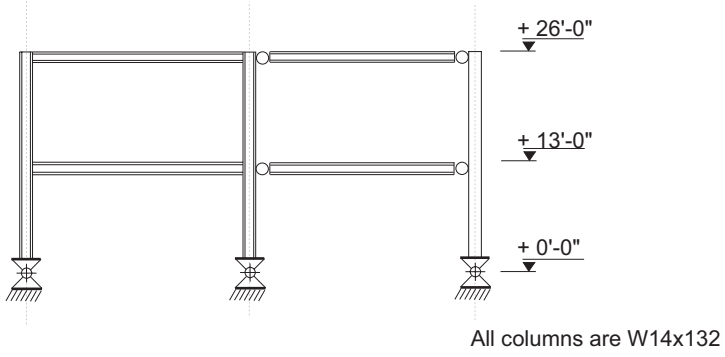
The dead and live loads are applied as beam span loads based on the loading intensities given below:

Roof	Dead load	= 30 psf
	Live load	= 20 psf
Floor	Dead load	= 75 psf
	Live load	= 80 psf
Ext. Wall	Dead load	= 15 psf

Equivalent static seismic forces are applied as lateral joint loads in the global X and global Y directions, separately. The total base shear in each direction is computed as 51 kips.



Framing Plan



Transverse Elevation

Loading:

Roof	Dead load	= 30 psf
	Live load	= 20 psf
Floor	Dead load	= 75 psf
	Live load	= 80 psf
Exterior	Wall	= 15 psf

Steel Grades:

For beams F_y = 36 ksi
For columns F_y = 50 ksi

Figure IV-1
Description of the Model (Tutorial Example)

Analysis

Two diaphragm constraints are applied for the two diaphragms at the two floor levels. These constraints prevent in-plane relative displacements of the nodes at each floor. The lateral earthquake loads are applied at nodal points on the floor levels. A P-Δ analysis is carried out with a load combination of 1.2 DL + 0.5 LL as described in Chapter "Check/Design for AISC-LRFD93". The initial unbraced length is taken as the full member length.

Design

The stress check and design are performed in accordance with AISC-LRFD93. Kip-in units are used. The input data file for this model is **EXSTL.SDB**. This is supplied as part of the SAP2000 package.

Starting the Tutorial

A step-by-step procedure for the stress check and design of the model is outlined below. It is recommended that you actually perform these steps while reading this chapter. We assume that you have successfully started the program. You can do this by running SAP2000 from the Start Menu of the Windows operating system.

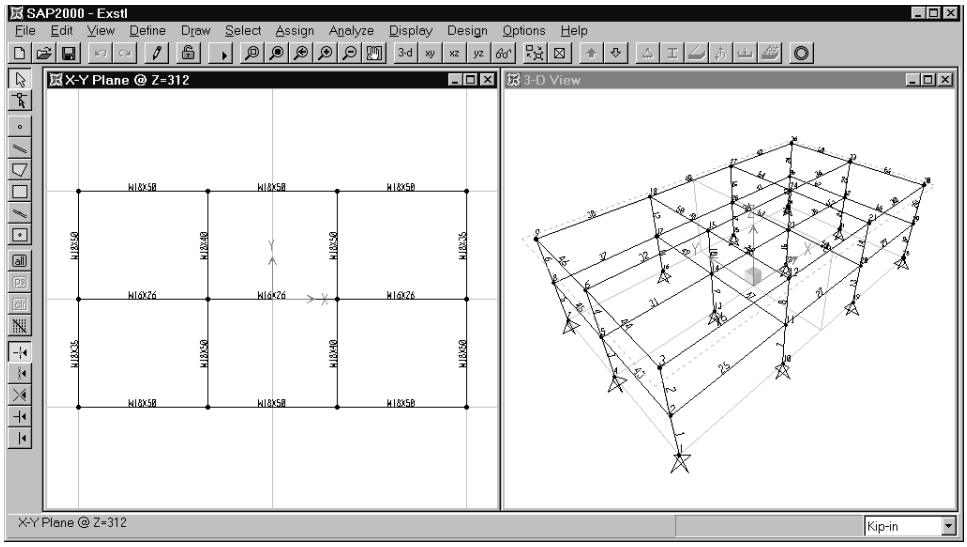
In this tutorial we will typically use the Toolbars to access various options quickly. Most of the features available on the Toolbars can also be accessed from the Menu Bar. Use the on-line Help or refer to the chapter "The Graphical User Interface" in *SAP2000 Getting Started* manual for a detailed description of the SAP2000 screen.

The input data file for the model (**EXSTL.SDB**) is in the EXAMPLES subdirectory under the directory where the program has been installed. A section property file of name **SECTIONS.PRO** is also required for the tutorial. The SECTIONS.PRO file is also available in the directory where the program has been installed.

Opening the Model Database File

1. Click on the **Open** button in the **File** menu. This will display the **Open Model File** dialog box.
2. In this dialog box:
   - Select the **EXSTL.SDB** file.
   - Click on the **Open** button.

The screen will now show two vertically-tiled windows. The left window displays a plan view of the model at level + 312 in. Section labels are displayed in this view. A three-dimensional view of the model is shown in the right window. Element and node IDs are shown in this view. In addition, the Side Toolbar is displayed at the left-side of the main window along with the Main Toolbar at the top of the window.



Note: When working with multiple windows, clicking anywhere in a particular window will activate that window.

Before we proceed further, we will make a copy of the data file by saving the model under a new name, say, **TUTOR2.SDB**. We will use this copy during the tutorial and leave the original file unaltered.

3. From the **File** menu, choose **Save As...**. This will display the **Save Model File As** dialog box.

4. In this dialog box:

   - Enter new filename, Tutor2.SDB.

     Note: Even if you do not type in the extension .SDB, the program automatically appends this extension to the filename.

   - Click on the **Save** button.

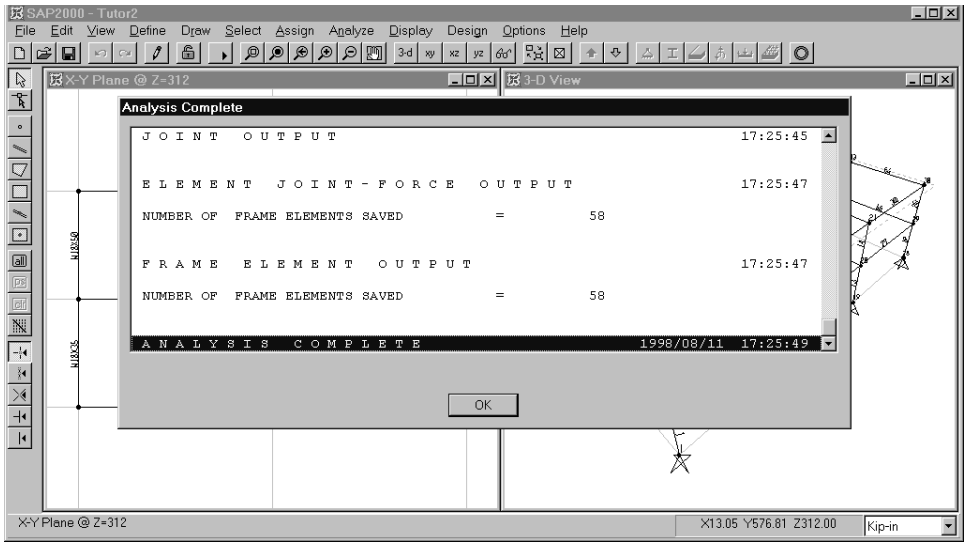
The new name is displayed in the Main Title Bar.

Analyzing the Model

We will now analyze the model. Before analyzing the model we need to set the P-Δ force (1.2 DL + 0.5 LL) and other parameters for P-Δ analysis. To do this:

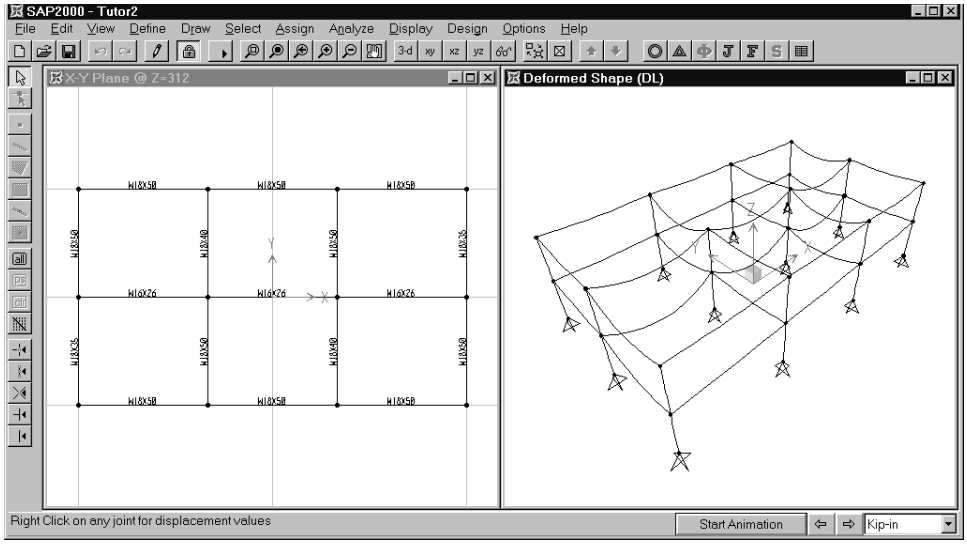
1. Select the **Set Options...** menu item in the **Analyze** menu. This will immediately bring up the **Analysis Options** dialog box. In this dialog box:
   - Click the **Include P-Delta** check box.
   - Click on the **Set P-Delta Parameters** button. This will bring up the **P-Delta Parameters** dialog box. In this dialog box:
     – Set maximum iteration to 5.
     – Change the DL scale factor to 1.2 and click **Modify**.
     – Click on the **Load Case** drop down arrow.
     – Select LL.
     – Change the LL scale factor to 0.5 and click **Add**.
     – Click **OK** to close the **P-Delta Parameters** dialog box
   - Click **OK** to close the **Analysis Options** dialog box.
2. Click on the **Run Analysis** button from the Main Toolbar.

A top window is opened in which various phases of analysis are progressively reported. When the analysis is complete the screen will display the following:



3. Use the scroll bar in the top window to review the analysis messages and to check for any error or warning messages. In our case there should be none.

4. Click on the **OK** button in the top window to close it. This will respond by displaying the deformed shape in the right window, which is in the 3D display mode, for the first load case (DL).



Selecting the Design Code

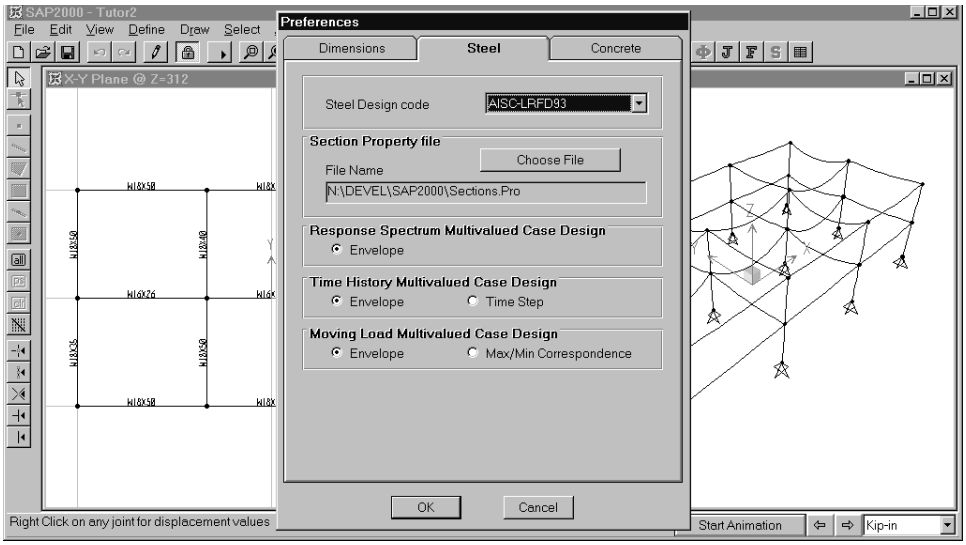
The default design code is AISC-ASD89 for design of steel structures. To choose the AISC-LRFD93 design code, do the following:

1. Click on **Preferences...** from the **Options** menu. This will launch the **Preferences** dialog box.

2. In the dialog box click on the **Steel** tab. You can see the currently selected steel design code, default section property file, and some other options as shown below.

3. You need to change the default design code (AISC-ASD89) to AISC-LRFD93. to do this:

   - Click on the drop down arrow in the design code box.
   - Select the **AISC-LRFD93** design code.



4. Click on the **OK** button to close the dialog box.

5. To make sure that SAP2000 will design the steel members, select **Steel Design** menu item from the **Design** menu.

Starting Design and Stress Check

With the analysis phase and selection of design code completed, we will now examine member stress ratios using the requirements of AISC-LRFD93.

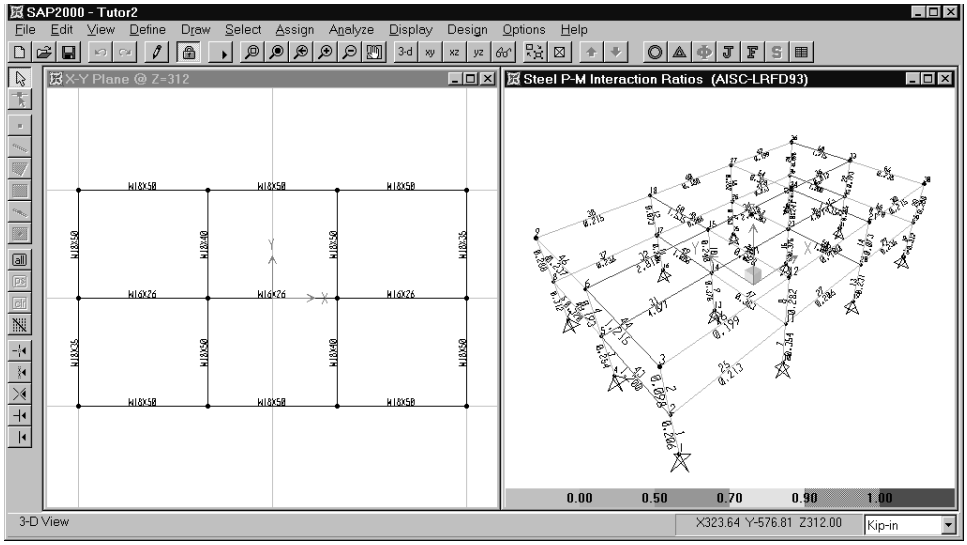
1. From the **Design** menu, choose **Start Design/Check of Structure**. The program now calculates the interaction ratios for each of the steel frame members. (If we had selected some of the frame members, then stress ratios would have been calculated for only the selected frame members). In a few moments the color-coded stress ratios are displayed for each member. By default, these are the axial force-moment interaction ratios which, according to the code, should not exceed 1.0. In the display, the stress ratios are reported for the governing design load combination.

*Note: Since no load combinations were defined in the model, SAP2000 design automatically provided a set of design load combinations for the selected design code (**AISC-LRFD93** for **Steel**). The default load combinations can be added by any of the three following ways.*

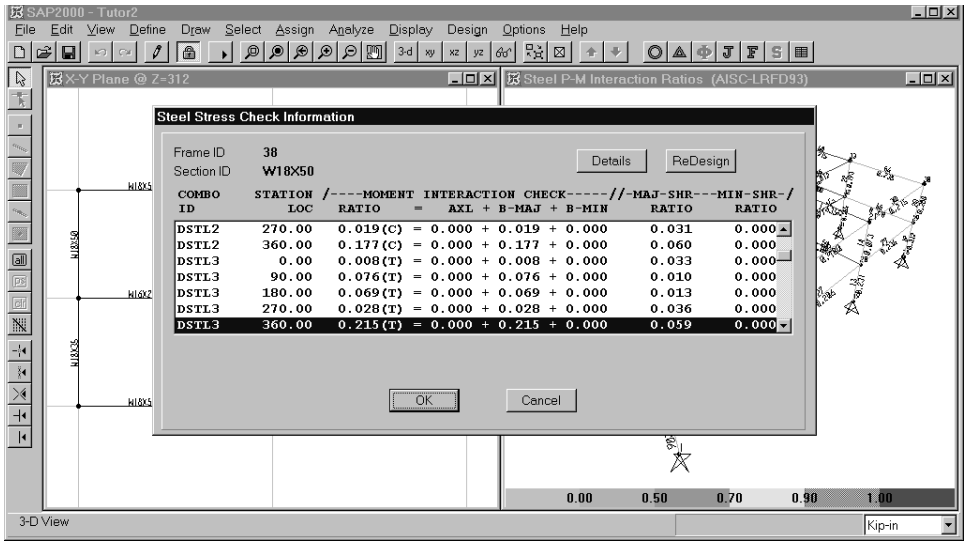
Clicking **Add Default Design Combo** *once in the* **Define Load Combinations** *dialog box launched from the* **Define** *menu by clicking* **Load Combinations....**

Or clicking **Select Design Combos...** *from the* **Design** *menu when there are no design load combinations defined in the model.*

Or clicking **Start Design/Check of Structure** *from the* **Design** *menu when there are no design load combinations defined in the model.*

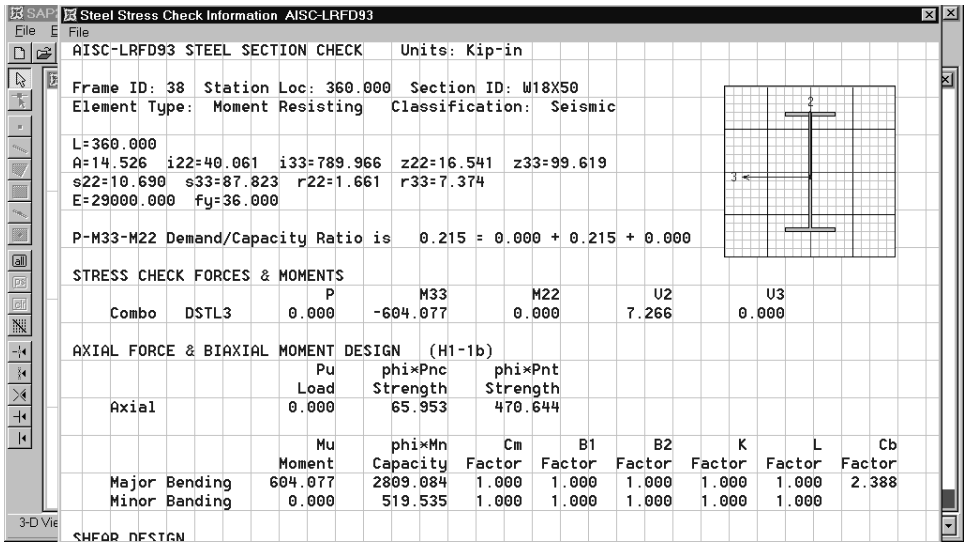


2. **Right click** on a member, say the beam on gridline A, lines 1-2 (element ID 38). See Figure IV-1 on page 47 and the screen above. This will open the **Steel Stress Check Information** dialog box showing stress ratios at various stations along the element length for different load combinations. The dialog box also shows information regarding the structural section (W18x50) assigned to the member. The largest stress ratio is highlighted in this box. The dialog box also can show information regarding the **Details** of calculation for design and the element overwrite assignments for **ReDesign** for the selected member.



3. In this dialog box:

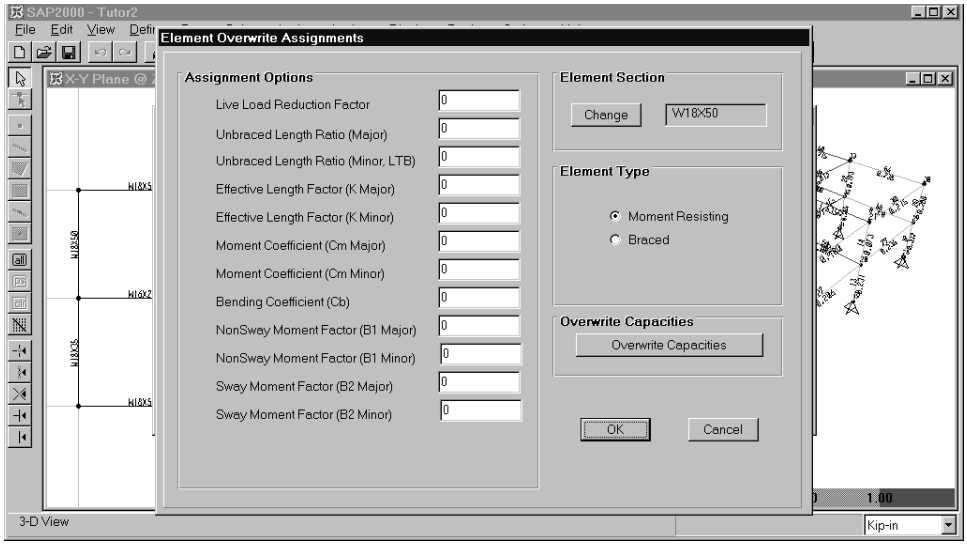
   - Select a design check station by clicking once (select the default highlighted one by doing nothing).
   - Click on the **Details** button. This will open the **Steel Section Design** detailed information window showing the steel section design parameters including the member identification, geometric parameters, material properties, nominal strength values, the factored member forces for the selected load combination, and other design parameters at that particular station.



- Close the **Steel Section Design** information window.

4. Click on the **ReDesign** button on the **Steel Stress Check Information** dialog box. This will open the **Element Overwrite Assignments** information dialog box showing the input design factors including the *K* factors, C_m factors, etc. These factors can be edited for redesigning. See screen below. Here "0" means the default values. There is also an alternate way of editing the properties of a set of members which will be demonstrated in the next section "Modifying Member Properties of a Group".

   - Click on the **Cancel** button to close the **Element Overwrite Assignments** dialog box.
   - Click on the **Cancel** button to close the **Steel Stress Check Information** dialog box.



Note: The number of stations (number of segments + 1) used in the design and stress check is set by the user through **Frame** *and* **Output Segment** *menu items from the* **Assign** *menu prior to the analysis phase. The default number of segments is 4 for beams and 2 for columns.*

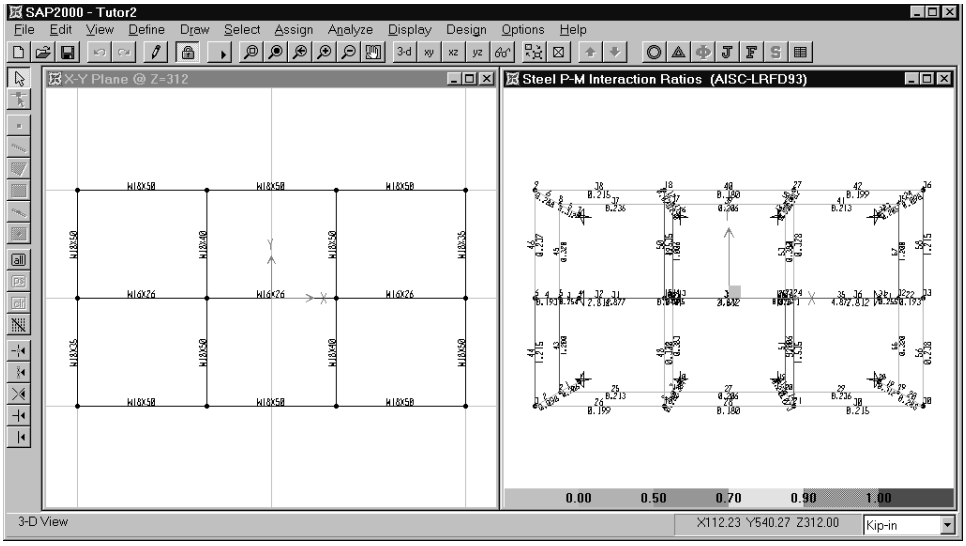
At this point we have analyzed and designed the steel frame and reviewed some of the design information. SAP2000 allows you to interactively change the design code, member properties, remove or add new load combinations, etc. and re-run the analysis and design phases. As a demonstration in the tutorial, we will edit/change a member property for a set of frame members.

Modifying Member Properties

With the analysis and stress check successfully completed, we will now modify the lateral unsupported length of all the beams and girders because the stress ratios are greater than 1 for some of the beams and girders and $l/r > 200$ for many of the beams. Initially, in the design check, the unbraced length of each member was taken to be the member full length in both the major and minor directions, i.e. $l_{22} = l_{33} = L$. However, the secondary beams of the floor structure provide restraints against lateral displacement of the compression flange. Referring to Figure IV-1 (page 47) we will take the minor direction unbraced length, l_{22}, of every beam spanning N-S (Y-Y) as 33% of the actual length. Likewise, for beams spanning E-W (X-X), l_{22} is assumed to be 25% of the actual length. In order to make these modifications we will change the view in the right window to make all the beams visible for selection. Notice that this window is currently showing the stress ratios from the previous check in a 3D view.

1. Click on the **2D View** (**xy**) button on the Main Toolbar.

2. Click on the **Perspective Toggle** button on the Main Toolbar.



Now, with all the beams visible, we can select beams and modify their lateral unbraced lengths.

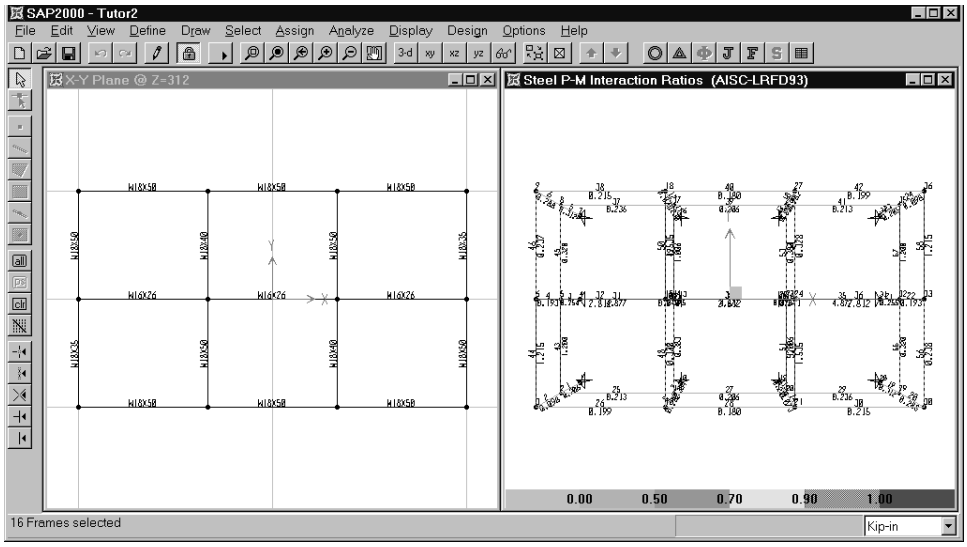
3. To select the N-S (Y-Y) beams do the following:

   - Click the **Set Intersecting Line Select Mode** button on the Side Toolbar.

- Move the pointer to the left of the beam on line 1, bay A-B.
- Click and hold down the left mouse button.
- While holding, move the pointer to the right intersecting all the beams in the bay between lines A and B. A "rubber-band line" in the Y-Y direction will show the intersecting line.
- Release the left mouse button to select all members that intersect this line. The message area at the bottom-left corner of SAP2000 responds by showing that **8 Frames Selected**.

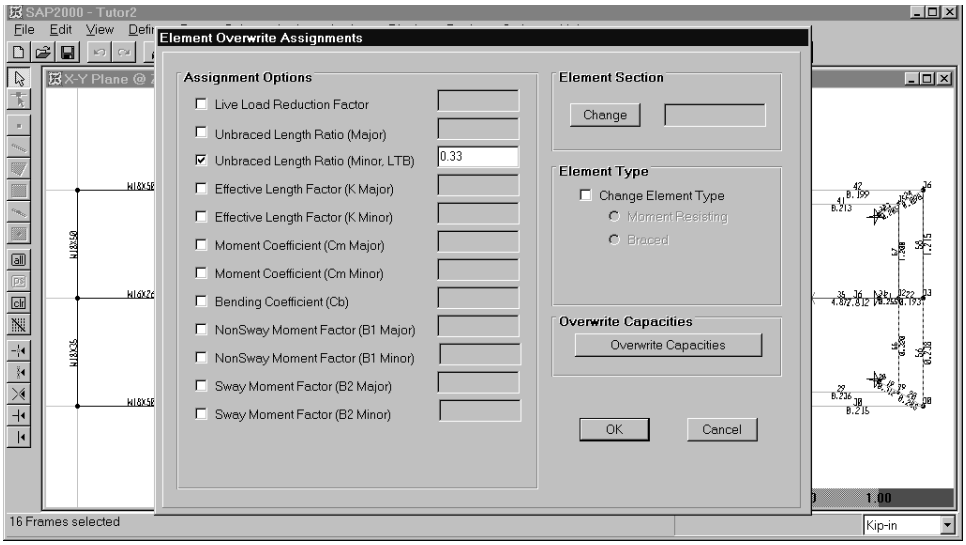
  Note: To select all the N-S beams we have to do this operation (Step 3) twice; once, for all the beams between lines A and B, and once for beams spanning between lines B and C.

The selection of all N-S beams is now complete. The selected members appear as dashed lines.



4. From the **Design** menu, choose the menu item **ReDefine Element Design Data...**. This will display the **Element Overwrite Assignments** dialog box. In this dialog box:

   - Check the **Unbraced Length Ratio (Minor, LTB)** box in the **Assignment Options** area.
   - Replace 0 by 0.33.

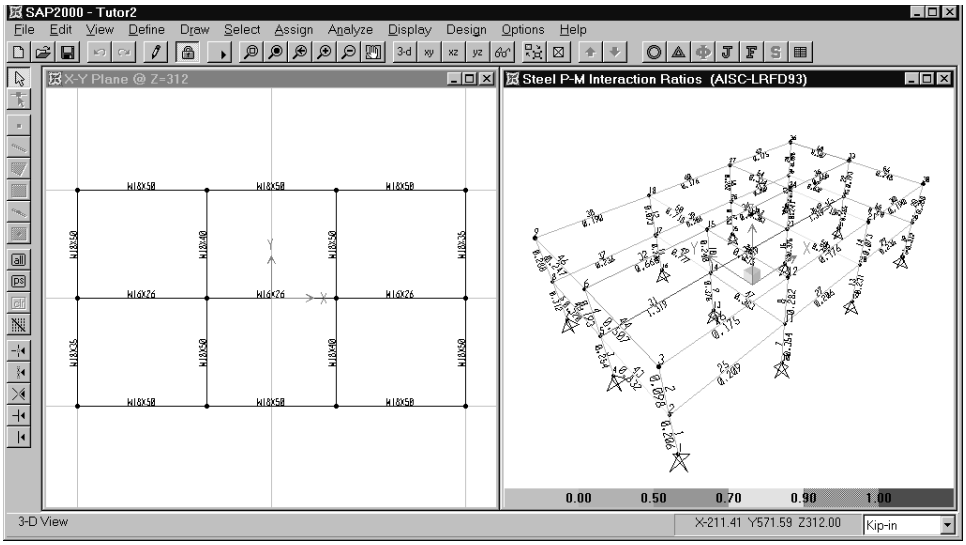


Note: The **Assignment Options** *shown in this dialog box are code-dependent.*

- Click on the **OK** button to accept the unbraced length ratio.
- Click on the **Refresh Window** button from the Main Toolbar. Notice that as a result of using shorter unsupported lengths, the stress ratios in these particular members have decreased significantly.

5. Repeat Steps 3 and 4 to modify all the E-W (X-X) beams, except enter the unbraced length ratio as 0.25.

6. The stress ratios after redefining the unbraced member length can be made more visible in the 3D display. Click **3D View** button on the Main Toolbar. Compare the following display with the one on page 53.



Selecting Sections Automatically

After changing the unbraced length, most of the beam members are found to be understressed except only 3 members which are overstressed. These three overstressed members are color coded with red. As an exercise, we will select sections for these overstressed members automatically.

These three overstressed beams are identified by element numbers 31, 33, and 35. Currently all of these members have a W18x35 section. If we gradually replace these sections with larger ones, the stress ratio can be made close to 1 but less than 1. W18x50 section will satisfy the requirements. Instead we will do an exercise showing automatic selection of members.

In the right side window, where the steel stress ratios are displayed in a 3D view:

- Select these three overstressed members, which are color coded with red, by clicking on them one by one.
- Unlock the model by clicking on the **Lock/Unlock Model** from the Main Toolbar.
  - This will prompt a dialog box asking **"Unlocking model will delete analysis results! OK to delete?".** Click **OK**.
- Click on the **Frame Sections...** in the **Define** menu. This will bring up a **Define Frame Sections** dialog box. This dialog box shows the available sections in the model. We will make some more sections available for the model so that the

program can choose the automatically selected section from a wider group. Since the W18x35 is too big for some members, we will add some smaller sections, especially wide flange sections with 12 inch depth, for the domain of AUTO section. To achieve this, in this dialog box:

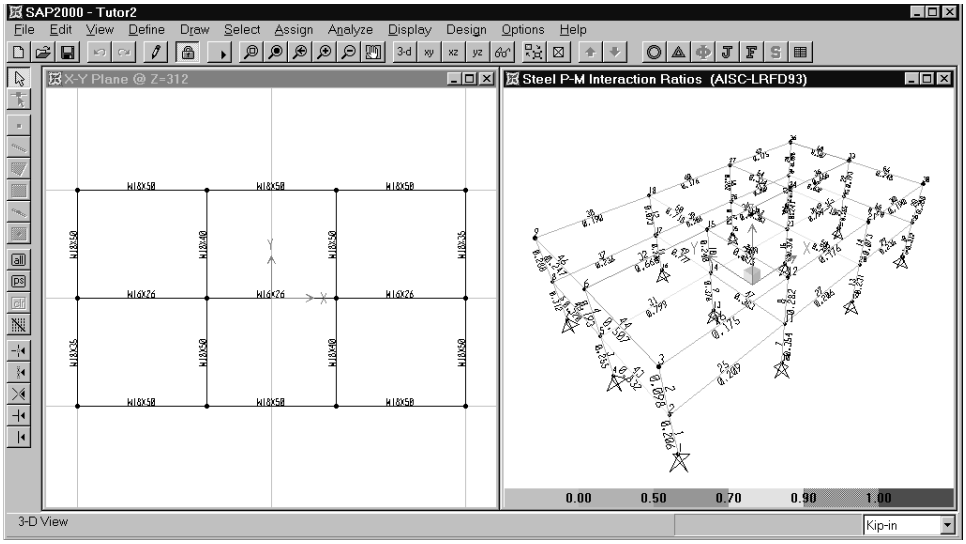
- Click on the **Import** pull down arrow.
- Scroll through the sections and choose **Import I/Wide Flange**. This will bring an **Import I/Wide Flange** section property list.

  *Note: In this dialog box, the default section property file **Section.pro** is used. This file can be re-chosen from the **Preferences** form from the **Options** menu.*

- Scroll down through the sections and select **W12x96** by clicking. Scroll down through the sections again and select **W12x14** by holding down the Shift Key and clicking. This will select all the sections ranging from W12x96 to W12x14.
- Click **OK** to choose the sections just selected. The response will be a display of information in the **I/Wide Flange Sections** dialog box about the last selected section, i.e., **W12x14**.
- Click **OK** in the **I/Wide Flange Sections** dialog box. This will complete importing the newly selected sections into the model from the database. The imported sections are added to the **Frame Sections** list in the **Define Frame Sections** dialog box.
- Click on the **Add** pull down arrow in this dialog box.
- Scroll through the sections and choose **Add Auto Select**. This will bring an **Auto Selection Sections** dialog box. In this dialog box, the default domain of the Auto Sections is listed. You can edit the list by adding and deleting new sections. Scroll through the sections down to W14x132 and select it by clicking. The Remove button is highlighted. Click on the **Remove** button to remove this specific section from the domain of the auto section because the W14x132 is specifically used in this model for columns. The default name of the auto section is given as AUTO1. Click **OK** to accept the default name and the list of sections.
- Click the **OK** button to close the **Define Frame Sections** dialog box.

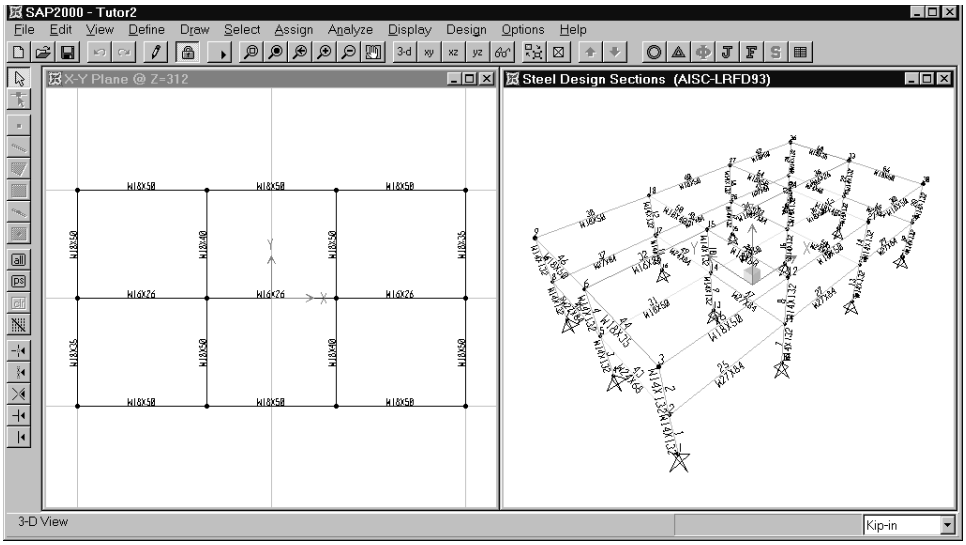
• Click on the **Assign** menu, select the **Frame** menu item, and then select **Sections...** . This will open the **Define Frame Sections** dialog box. In this dialog box you select **AUTO1** and then click **OK**. This will change the display recognizing that the selected members have **AUTO1** section.

- Reanalyze the model by clicking on the **Run Analysis** button on the Main Toolbar. Click **OK** to close the top window.
- Recalculate the stress ratio by clicking on the **Start Design/Check of Structure** menu item in the **Design** menu. This will respond by displaying new stress ratios on a 3D View in the right window.



Notice that as a result of changing the section, the stress ratios for those particular beams are changed. To see the difference, compare this display with the one on page 59.

- To see the newly selected sections, click on the **Display Design Info...** menu item in the **Design** menu. This will bring up the **Display Design Results** dialog box. In this dialog box, select the **Design Input** option button, accept the **Design Sections** from the **Design Input** drop down list item, and click **OK**. This will show the newly selected sections on the 3D view in the right window.



Note: The displayed sections are the design sections. Analysis sections can also be displayed by clicking on the **Set Elements** *button on the Main Toolbar and clicking on the* **Sections** *check box for* **Frames** *on the* **Set Elements** *dialog box.*

Re-analyzing with Updated Elements

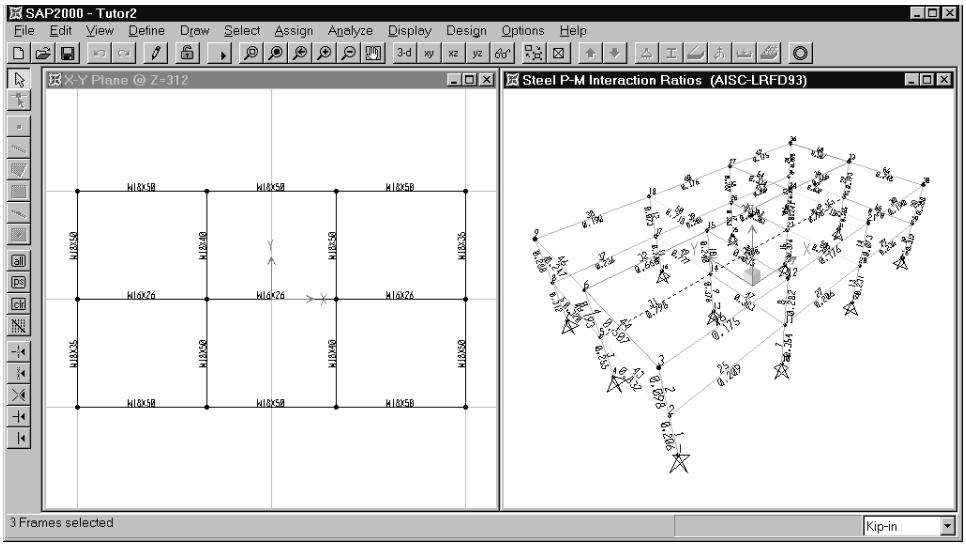
It is important to realize that changes made to member section properties in the stress/design phase are **not** automatically reflected in the analysis results. These changes are only local to the post-processing phase unless a re-run of the analysis, with updated elements, is requested by the user. In other words, overwriting the section properties only affects the stress values and not the factored element forces obtained in the analysis preceding such changes. The redistribution of member forces due to change of stiffness (revision of section properties) is effected in a re-run of analysis. The section properties can only be updated on a set of selected elements. We need to refresh the analysis model, reanalyze the model, and redesign the structure. To do this:

- Click on the **Restore Previous Selection** from the Side Toolbar to re-select the previously selected frame elements.
- In the **Design** menu click on the menu item **Update Analysis Sections**. This will prompt a dialog box asking **"Updating Analysis Section will unlock model! OK to update?".** Click **OK**.

- From the **Analyze** menu, choose **Run**. This will immediately start the analysis procedure. A top window is opened in which various phases of analysis are progressively displayed. The results will obviously differ from those produced in the initial analysis because of the change of section properties we made in the design stage. Click **OK** to close the top window.
- Click on the **Start Design/Check of Structures** from the **Design** menu. This will redesign the structure and display the new stress ratios.
- Now replace auto sections with optimal sections. To do this:
  - Click on the **Restore Previous Selection** from the Side Toolbar to re-select the previously selected frame elements.
  - Click on the **Replace Auto w/Optimal Sections** menu item from the **Design** menu.
  - This will ask "**Replacing Analysis Sections with Auto-Selected Sections will unlock model! OK to replace?**". Click **OK** to replace.

Note: Once you are satisfied with the selected sections, selecting the **Replace Auto w/Optimal Sections** *menu item will permanently replace the auto sections with the current design sections. This effectively replaces the analysis sections with the optimal design sections and removes the auto tag. So selecting this menu item is one of the last things you should do.*

You can see the difference after re-running the design based on the latest analysis results.



Concluding Remarks

We have come to the end of this tutorial on the SAP2000 steel design options. The intent has been to highlight and demonstrate a few of the basic features in order to open up the path for you to explore and use the more advanced options. For more information on various topics consult the on-line Help provided.