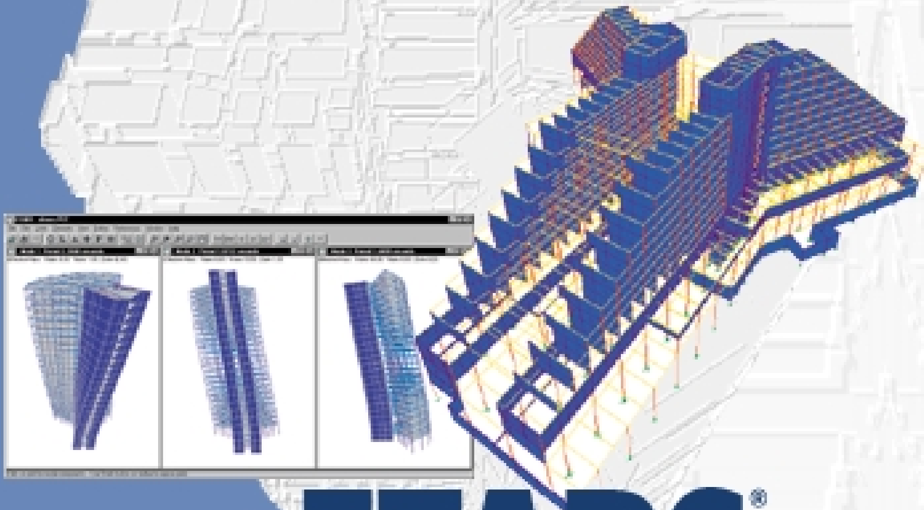


ETABS®

ETABS Tutorial

COMPUTERS & STRUCTURES INC.®

STRUCTURAL AND EARTHQUAKE ENGINEERING SOFTWARE

ETABS®

Three Dimensional Analysis and Design of Building Systems

Tutorial

COMPUTERS & STRUCTURES INC.

Computers and Structures, Inc.
Berkeley, California, USA

First Edition
July 2000

Copyright

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

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

Phone: (510) 845-2177
FAX: (510) 845-4096
e-mail: info@csiberkeley.com (for general questions)
e-mail: support@csiberkeley.com (for technical support questions)
web: www.csiberkeley.com

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF ETABS. 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.

Chapter 1

Introduction

ETABS

ETABS is a special-purpose computer program developed specifically for building structures. It provides the Structural Engineer with all the tools necessary to create, modify, analyze, design, and optimize building models. These features are fully integrated in a single, Windows-based, graphical user interface that is unmatched in terms of ease-of-use, productivity, and capability.

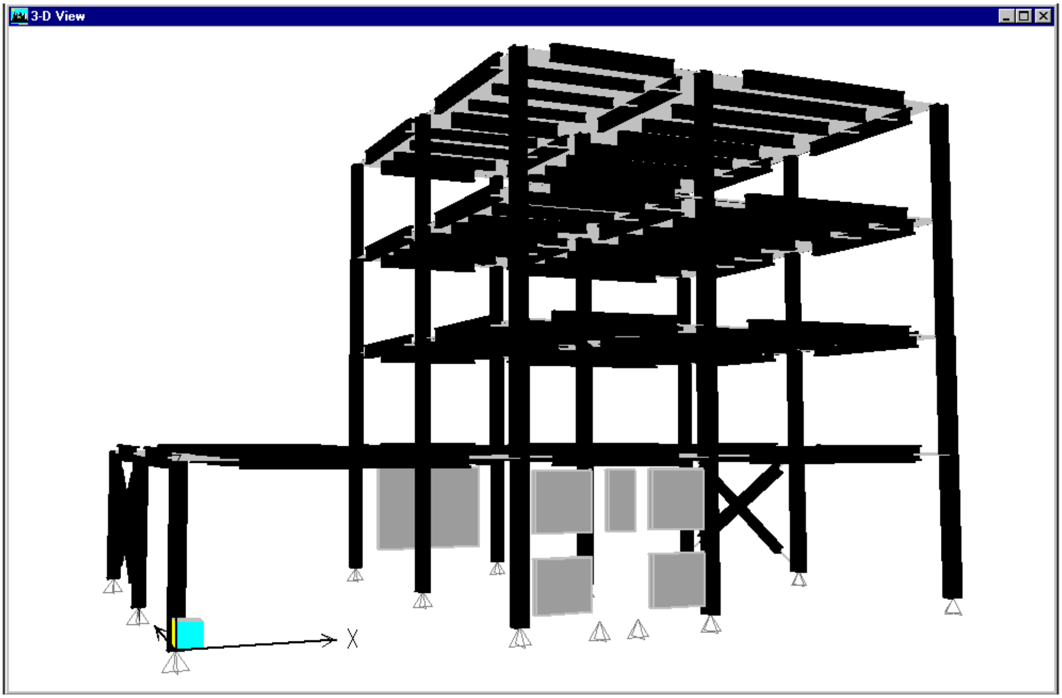
Tutorial

Tip:

Self-running tutorials are also available for installation from the ETABS CD.

The tutorial in the manual is intended to give you hands-on experience using ETABS. For most people, this is the quickest way to become familiar with the program.

Our example is a four-story, three-by-two-bay, steel moment-frame building with a significant setback above the first story.



The bottom story has bracing in two bays and shear walls in two bays. The finished product is shown above.

We will develop the model, perform the analysis, check the design against code requirements, and iterate until we find an "optimum" design.

Procedure

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

The ETABS program must be installed on your computer before you can begin the tutorial. Installation instructions are given in Chapter 2, "Installation," of the *ETABS User's Manual*.

It would also be a good idea to read the Chapters 1, 3, 4, 5, and 6 of the *ETABS User's Manual* before beginning the tutorial, or at

Note:

You may wish to review Chapter 4 of the User's Manual, which provides you with an overview of the ETABS graphical user interface, before starting this tutorial.

least have them readily available as you work through this example.

Print this *ETABS Tutorial* manual before starting the tutorial. It will not be practical to use the ETABS program while trying to read this manual on your computer screen.

The tutorial is divided into four parts, presented in Chapters 2 through 5. They constitute a single example and should be performed in sequence.

During the course of this tutorial, we will explore many of the basic features of ETABS. Prepare to spend at least an hour going through this example. It will probably save you a lot more time in the future than you will spend now.

If at any time you need to stop, save your model, and continue later from where you left off.

With your printed copy of this tutorial and the *ETABS User's Manual* close at hand, sit down and get comfortable…

Chapter 2

Create the Model

We begin the tutorial example with this chapter. Here we will create the initial model and define its basic properties. If you have not done so already, please read Chapter 1 before proceeding.

Start ETABS

If ETABS is not already open, start the program by clicking on the appropriate desktop shortcut or by selecting ETABS from your Windows Start menu. This will open the ETABS main window.

Create a New Model

We will start a new model using the following steps:

Note:

See the section titled "Starting a New Model" in Chapter 8 of the User's Manual for additional information.

1. Set the units to kips and inches, "Kip-in", using the drop-down box in the lower right corner of the ETABS screen.
2. Select the **File menu > New Model** command.
3. Click the **No** button in the **New Model Initialization** form. This indicates that we do *not* wish to use a previous model as the starting point for this model.
4. This now opens the **Building Plan Grid System and Story Data Definition** form, where much of the definition of the structure takes place.

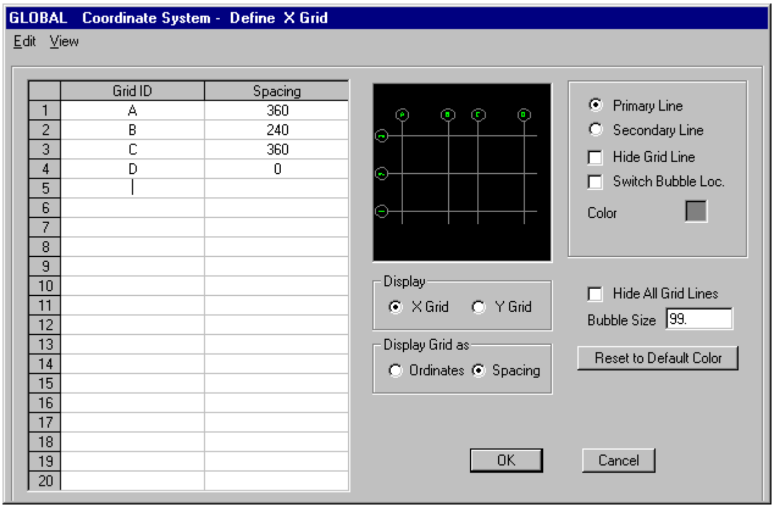
Set Grid Dimensions (Plan)

First we define the plan grid for the structure. The structure has three bays in the X direction with non-uniform spacing, and two equal bays in the Y direction. Working in the **Building Plan Grid System and Story Data Definition** form:

1. We start by selecting **Uniform Grid Spacing**, then entering:
   - "4" for the **Number Lines in X Direction**
   - "3" for the **Number Lines in Y Direction**
   - "360" (inches) for the **Spacing in X Direction**
   - "300" (inches) for the **Spacing in Y Direction**
2. Next we modify the grid spacing by selecting **Custom Grid Spacing** and clicking the **Edit Grid** button. This opens the **Coordinate System** form.
3. Select **Display: X Grid** and **Display Grid as: Spacing**.
4. Click the spacing value for Grid ID "B" (row 2 of the table) and change the value from "360" to "240".

Set Grid Dimensions (Plan), Step 5

*Completed **Coordinate System** form for the X grid.*



5. Click a blank cell in the grid table (say row 5, column 1) to update the pictorial display of the grid. The result should look like the figure.

6. Click the **OK** button. We do not need to change the Y grid spacing.

Set Story Dimensions

Next we define the vertical dimensions of the building. Continuing in the **Building Plan Grid System and Story Data Definition** form:

1. We start by selecting **Simple Story Data**, then entering “4” for the **Number of Stories** and “150” (inches) for the **Story Height**.

Note:

See the section titled “Similar Story Levels” in Chapter 23 of the User’s Manual for additional information.

2. Next we modify the story dimensions by selecting **Custom Story Data** and clicking the **Edit Story Data** button. This opens the **Story Data** form.

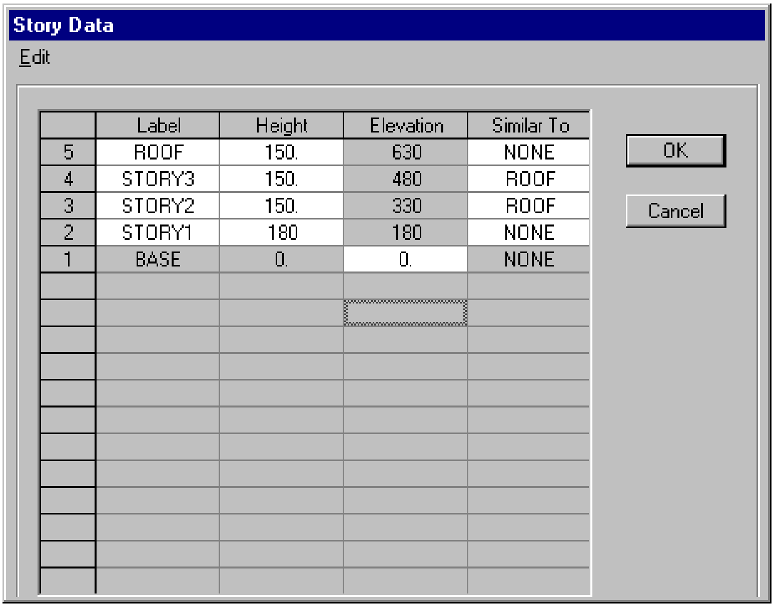
3. Change the **Label** of “STORY4” to “ROOF”.

4. Change the **Height** of “STORY1” to “180” inches

5. Note that “STORY1”, “STORY2”, and “STORY3” are declared to be similar to “ROOF”. Because of the setback our

Set Story Dimensions, Step 6

*Completed **Story Data** form.*



Story Data

Edit

	Label	Height	Elevation	Similar To
5	ROOF	150.	630	NONE
4	STORY3	150.	480	ROOF
3	STORY2	150.	330	ROOF
2	STORY1	180	180	NONE
1	BASE	0.	0.	NONE

OK

Cancel

"STORY1" is going to be different from the upper stories. Change the **Similar To** value for "STORY1" to "NONE".

6. Click a blank cell in the table to update all values. The result should look like the figure.

7. Click the **OK** button to close the form.

Add Structural Objects

So far we have only laid out a grid in the vertical and plan dimensions. Now we will add the beams, columns, and steel decks. These are called "structural objects" in the model. These objects are the starting point for the model; we will some make changes later:

Continuing to work in the **Building Plan Grid System and Story Data Definition** form:

1. Click the **Steel Deck** button under **Add Structural Objects**. This opens the **Steel Floor System** form.

2. First we define the floor system:

- Under **Overhangs**, change all four values to "0". This is the distance the floor extends beyond the perimeter grid lines. Using zero will simplify our model, and this is recommended for small values of overhang to help avoid poor aspect ratios in your slab mesh.

- Under **Secondary Beams**, make sure that **Secondary Beams** box is checked, select the span **Direction** to be "X", and set **Number** of beams per bay to be "3".

- Lastly, for the floor itself, we will define the loading acting in the two default load cases:

  Case "DEAD" is the default dead-load case that automatically includes the self-weight of all material in the structure. We will not add any additional floor load to this case. Later we will create a new load case to handle superimposed dead load for composite-floor design.

  Case "LIVE" is the default live-load case. Initially it has zero load in it. Under **Loading**, set the **Live Load** value to "0.000347". This value represents 50 psf (lb/ft^2), converted to kip-in units.

3. Next we define the framing system:

   - Select **Structural System Type: Intersecting Moment Frame**. This indicates that all columns and beams (except the secondary beams) contribute to the lateral-force-resisting system.

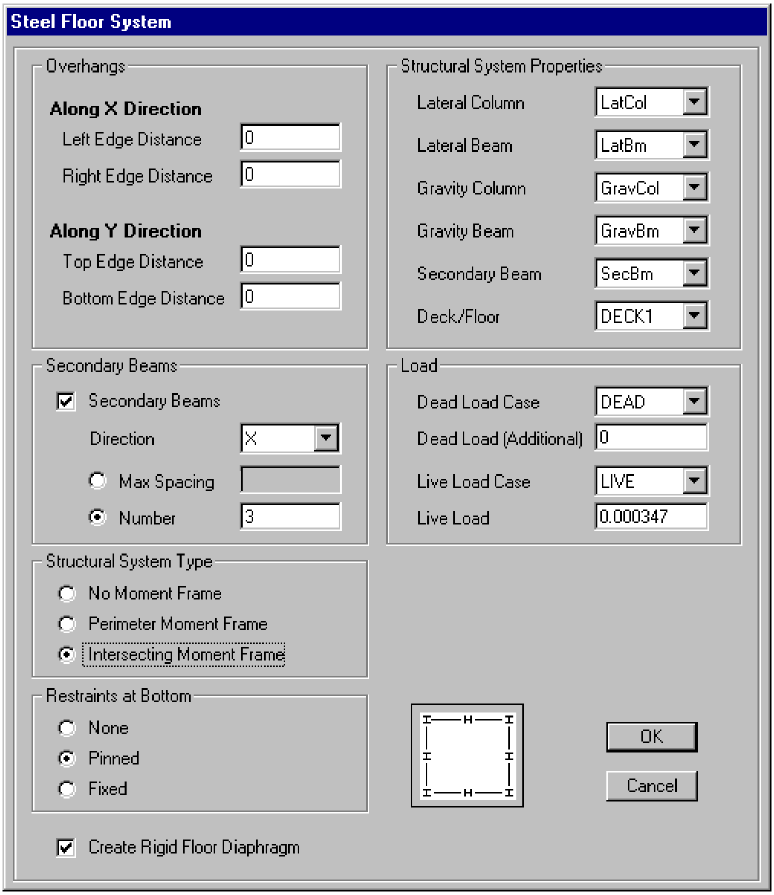
   - Select **Restraints at Bottom: Pinned**.

   - Make sure the **Create Rigid Floor Diaphragm** box is checked. This will create a constraint at each floor level so that the floor moves horizontally as a rigid body, and will be needed to use automated seismic loads with eccentricity.

4. Lastly, we define the **Structural System Properties** to be used by the different structural objects. You may select from properties that are predefined by the program. We will ex-

Add Structural Objects, Step 5

Completed ***Steel Floor System*** *form.*



amine the definitions of these properties later in this tutorial. You always have the option of modifying and adding to these property definitions. For this example, we will use the default values as follows:

- **Lateral Column**: Select “LatCol”. This is a set of steel sections, called an **auto select section list**, to be used for the columns of the lateral-force-resisting system. The program will select the optimum members from this set during steel frame design. We will examine the definition of this auto select section list later.

- **Lateral Beam**: Select “LatBm”. This defines an auto select section list to be used for the beams of the lateral-force-resisting system.

- **Gravity Column**: This is not used since all columns of our intersecting moment frame are part of the lateral-force-resisting system.

- **Gravity Beam**: This is not used since all beams (except the secondary beams) of our intersecting moment frame are part of the lateral-force-resisting system.

- **Secondary Beam**: Select “SecBm”. This defines an auto select section list to be used for the secondary beams of the flooring system from which the program will select the optimum members during design.

- **Deck/Floor**: Select “DECK1”. Note that this is a single property, not a set of multiple properties. Auto select section lists are only available for steel members.

5. When you are done, the **Steel Floor System** form should look like the figure above.

6. Click **OK** to close the **Steel Floor System** form.

7. Click **OK** to close the **Building Plan Grid System and Story Data Definition** form. Two views of the structure should now appear, as shown in the figure below.

We have completed the initial definition of the structural model.

Save the Model

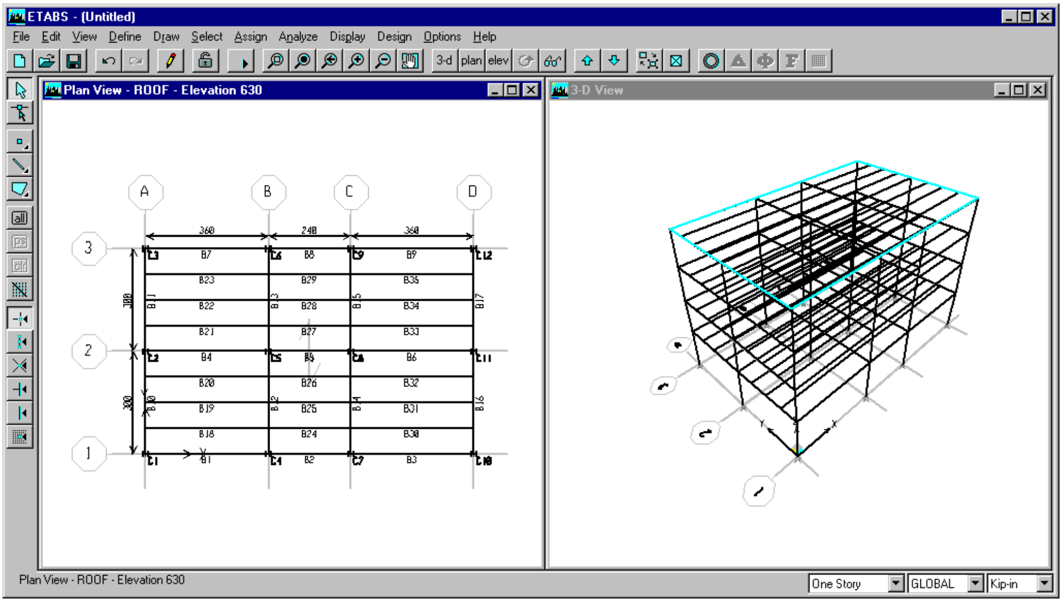
Tip:

Save your model often. You may also want to occasionally save a backup copy of your model with a different name.

It’s a good idea to save your model often to prevent loss of data that can occur due to computer failures or mistakes that you might make. Let’s do our first save now:

1. Select the **File menu > Save** command. Because it is the first time this model has been saved, this opens the **Save Model File As** form. This is a standard Windows file-saving form.

2. Using standard Windows operations, select the folder where you want to save this file. Then enter a file name, such as “Tutorial 1”, in the **File name** edit box. ETABS will automatically add the file extension “.EDB” to the file name.



Add Structural Objects, Step 7 *Initial Model.*

3. Click the **Save** button. This saves the file and closes the form.

Close and Restart ETABS

We will close ETABS, restart the program, and re-open our current model file. You may do this again at any point later in the tutorial if you need to take a break:

1. Select the **File > Exit** command to close ETABS. (If you had made any changes since the last time you saved the model, you would be given a chance to save your model before the program is closed.)

2. Restart the program by clicking on the appropriate desktop shortcut or by selecting ETABS from your Windows Start menu.

3. Select the **File menu > Open** command. This opens the standard Windows file-opening form.

4. Using standard Windows operations, select the folder where you previously saved the file “Tutorial 1.EDB”.

5. Click on the file “Tutorial 1.EDB”, or type “Tutorial 1” in the **File Name** edit box. ETABS will automatically add the file extension “.EDB” to the file name.

6. Click the **Open** button.

We are now ready to continue.

View the Model

By default, ETABS displays two views of the structure. In the left window is shown the plan view of the top story, “ROOF”, and in the right view is shown a 3-D view of the whole building.

Tip:

You can change the active view by clicking anywhere on the title bar of the window you want to make active, or by clicking in the window itself. Clicking on the title bar avoids accidentally selecting something while you are activating the window.

Only one view can be active at a time. You can change the active view by clicking the title bar of the desired window. The title bar will then be highlighted. Any changes made to viewing options will only affect the active view.

Let’s try some viewing options:

1. Click the title bar of the left window to make sure the plan view is active. Notice that a bounding rectangle is shown in the 3-D view showing which floor is displayed in the active view.

2. Move the mouse around in this view. Notice how the coordinates of the mouse are shown on the status bar at the bottom of the main ETABS window. The Z coordinate doesn’t change since we are at the fixed elevation of the top story.

3. On the top toolbar, click the **Move Down in List** button, , repeatedly to change the plan view to different story levels. Note how the bounding rectangle changes in the 3-D view to the right. Clicking the **Move Up in List** button, , reverses this process.

4. Select the **View menu > Set Elevation View** command (or click the **Elevation View** button, , on the top toolbar) to open the **Set Elevation View** form.

5. Select “1” under **Elevations**, and click **OK** to close the form. The elevation along grid line “1” is displayed in the active view.

6. Click the **Move Up in List** button, , and/or the **Move Down in List** button, , repeatedly to view the seven different elevations. Note how the bounding rectangle changes in the 3-D view to the right.

Note:

See the subsection titled “Perspective Views” in Chapter 10 of the User’s Manual for additional information.

7. Click the **Perspective Toggle** button, , on the top toolbar to toggle between a perspective view based on the chosen elevation and back to the 2-D elevation view. Note that a 2-D view shows only a single plane with no depth.

8. Click the title bar of the right window to make the 3-D view active. Notice that the bounding rectangle disappears.

9. On the top toolbar, click the **Set Building View Options** button, . This opens the **Set Building View Options** form.

Note:

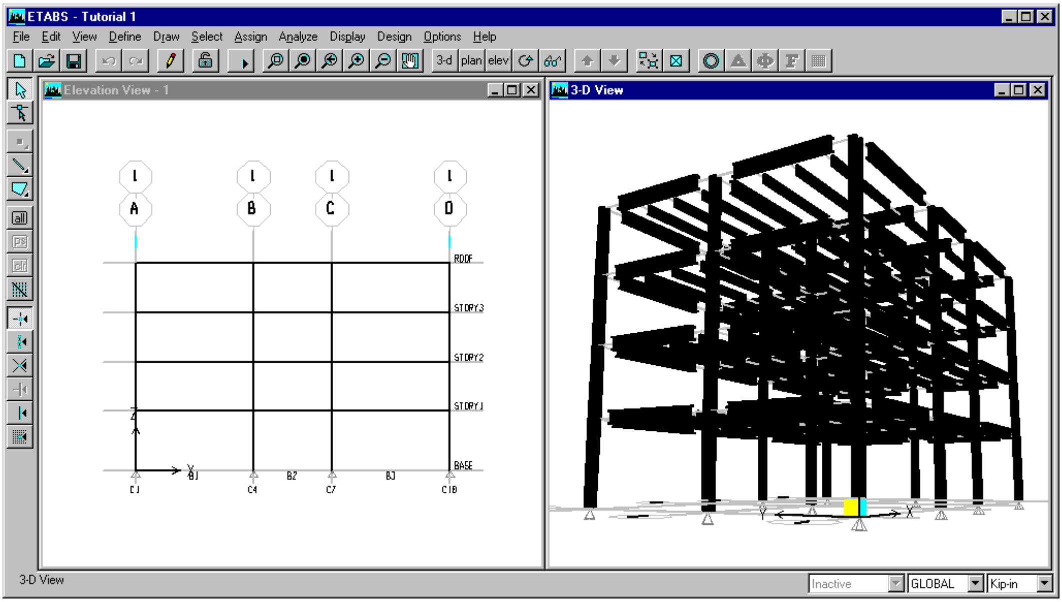
*Each of the options available in the **Set Building View Options** form is discussed in the section titled “Building View Options” in Chapter 10 of the User’s Manual.*

10. Under **Special Effects**, check the **Extrusion** box so that we can see the actual shape of the beam and column sections.

11. Under **Object Present in View**, uncheck the **Floor (Area)** box so that the floor does not obstruct our view of the framing.

12. Click **OK**. The 3-D view should now show the extruded view of the beams and columns. Note that the beams appear shortened for the sake of clarity. However, they actually extend to the column centerlines.

13. Click the **Rotate 3D View** button, , on the top toolbar. Then move the mouse cursor into the right window, click and hold the left mouse button while moving it around the screen. Movements to the left and right rotate the structure about the vertical viewing axis, and movements up and down



View the Model, Step 13 *Elevation "1" and Extruded 3-D View.*

move the structure about the horizontal viewing axis. When you release the mouse button, the structure will redraw. To rotate again, you must click the button again. See the figure above.

14. *Important:* If you perform this operation without first clicking the **Rotate 3D** button, the program may *select* objects, which we don't want to do right now. Click the **Clear Selection** button, , on the left toolbar to cancel this selection.

Tip:

You can also use the aerial view to zoom in on your model. See the section titled "The ETABS Aerial View" in Chapter 4 of the User's Manual for more information.

15. Let's zoom in. Click the **Rubber Band Zoom** button, , on the top toolbar. Then move the mouse cursor into the right window just outside the upper-left corner of the structure. Click and hold down the left button while dragging the mouse down and to the right until the dotted rectangle that appears encloses a small part of the structure, say one bay. When you release the mouse button, the enlarged portion of the structure will redraw.

16. *Important:* Again, if you drag the mouse like this without first clicking the **Rubber Band Zoom** button, the program

selects the objects in the dotted rectangle, which we don't want to do right now. Click the **Clear Selection** button, , on the left toolbar to cancel this selection.

17. To move around the neighborhood of the zoomed-in area, click the Pan button, , on the top toolbar. Then move the mouse cursor into the right window somewhere near the center. Click and hold down the left button while dragging the mouse around to pan the view. Notice that there is a limit to how far you can pan. Once you release the mouse button, you are no longer in pan mode.

Note:

See the subsection titled "Pan Feature" in Chapter 10 of the User's Manual for additional information.

18. To return to the full structure, click the **Restore Full View** button, , on the top toolbar. The first click undoes the pan operation. Click the button again to draw the full structure.

19. To return to the original 3-D view, click the **3D View** button, .

20. For what we will do later, we want to see the floors in the extruded view. Click the **Set Building View Options** button, , check the **Floor (Area)** box under **Object Present in View**, then click **OK**.

21. All of the viewing actions we have performed here using buttons on the top toolbar can also be accessed from the **View menu**. Additional viewing features are also available from this menu.

Define Material Properties

Default material properties are already defined for steel and concrete. Let's take a quick look at them, but not make any changes:

1. Select the **Define menu > Material Properties** command to open the **Define Materials** form.

2. Under **Materials**, select the property "CONC", which will be used by our floors and walls. Click the **Modify/Show**

Material button to bring up the **Material Property Data** form. In this form you will see:

- **Analysis Property Data**, which affects the load on the structure and the calculated response to the load. The material has been selected to be isotropic. These properties are:

  The mass per unit volume used for dynamic analysis.

  The weight per unit volume used for self-weight gravity loading.

  The modulus of elasticity and Poisson's ratio used for stiffness calculation.

  The coefficient of thermal expansion used for thermal loading.

  The shear modulus, which is computed by the program from the modulus of elasticity and Poisson's ratio (you cannot directly edit this item.)

- The **Type of Design**, which has been set to **Concrete**. The choices are **Concrete**, **Steel**, and **None**.

- **Design Property Data**, which affects the design-code checks performed by the program. These data generally do not affect the behavior of the structure under load. For concrete, these properties are:

  The specified compressive strength of the concrete.

  The yield stress of the bending reinforcing steel.

  The yield stress of the shear reinforcing steel.

  An option to specify a shear-strength reduction factor for lightweight concrete.

3. Since we are not changing anything, click the **Cancel** button to return to the **Define Materials** form.

4. Now select then property "STEEL", which will be used by all of our framing. Click the **Modify/Show Material** button to bring up the **Material Property Data** form. In this form you will see:

   - **Analysis Property Data**, which has the same type of data as did the concrete material.

   - The **Type of Design**, which has been set to **Steel**.

   - **Design Property Data**, which for steel are:

     The minimum yield stress.

     The minimum ultimate tensile stress.

     The cost per unit weight, which is used for composite beam design.

5. Since we are not changing anything, click the **Cancel** button to return to the **Define Materials** form.

6. Click the **Cancel** button to close the **Define Materials** form.

As a general rule, it is not a good idea to make changes to the two default properties, "CONC" and "STEEL". You can use the **Define Materials** form to define one or more new material definitions with the desired properties.

Define Frame Sections

Frame sections are named combinations of material and geometric cross-sectional properties that can be assigned to beams, columns, and other line objects. There are many different types of frame section properties that can be defined. Let's examine some frame sections and define a new one:

1. Select **Define menu > Frame Sections** command to open the **Define Frame Properties** form. You will see that a large number of predefined section properties already exist. More can be added.

2. Under **Properties**, select the property "W10X112". This is a wide-flange type of section. Click the **Modify/Show Property** button to bring up the **I/Wide Flange Section** form. Note the following:

   - The data for this section was obtained from the external file shown under **Extract Data from Section Property File**.

   - The material assigned to this section is "STEEL".

   - The geometric dimensions of the section are shown under **Dimensions** and illustrated in the figure on the form. Note that the section has two local axes, axis 2 being the major axis, and axis 3 being the minor axis.

   - Because this section came from an external file, no data on this form can be changed except for the material and the color. We will not change them.

   - Click the **Section Properties** button to open the **Property Data** form. This shows the geometric section properties calculated from the given dimensions. After reviewing this form, click **OK** to close it.

Note:

See the sections titled "Frame Section Properties" in Chapters 11 and 24 of the User's Manual for additional information.

3. Click **OK** to close the **I/Wide Flange Section** form and return to the **Define Frame Properties** form.

4. Under **Properties**, now select the property "LatCol". Recall that this is the property we specified for the columns in the model. This property is called an **auto selection section list**. It is a collection of several wide-flange sections from which the program will select one during design. Click the **Modify/Show Property** button to bring up the **Auto Selection Sections** form.

5. The two scroll boxes together list all the individual sections that are currently defined for this model that use a material specified for steel design. In the box on the right, labeled **Auto Selections**, are the sections included in this auto selection section list. In the box on the left, labeled **List of Sections**, are the sections *not* included in "LatCol".

6. Scroll through the **Auto Selections** on the right to see the sections available for the columns.

7. Click the **Cancel** button to close this form and return to the **Define Frame Properties** form.

8. Let's create our own auto select section list for later use by the braces. Click in the second drop-down box on the right that says "Add I/Wide Flange".

9. Scroll to the bottom of the list and click "Add Auto Select List". This opens the **Auto Selection Sections** form.

10. Change the **Auto Section Name** from "AUTO1" to "BRACE".

11. Scroll to the top of the **List of Sections** box and click on the topmost section, "W10X112".

12. Scroll down to the last of the W12 sections, namely "W12X96". Click on "W12X96" *while holding down the **Shift** key*. This selects all the W10 and W12 sections.

13. Click the **Add** button, which moves the selected sections to the **Auto Selections** box.

Note:

See the section titled "Using the Mouse" in Chapter 4 of the User's Manual for additional information on selecting multiple items in list boxes.

14. We are now going to remove the smaller sections from this list. In the **Auto Selections** box, click on "W10X12".

15. Scroll down slightly. Then, *while holding down the **Ctrl** key*, click on sections "W12X14", "W12X16", and "W12X19". Four sections should now be selected. If you made a mistake, keep holding down the **Ctrl** key while you click to select or deselect sections.

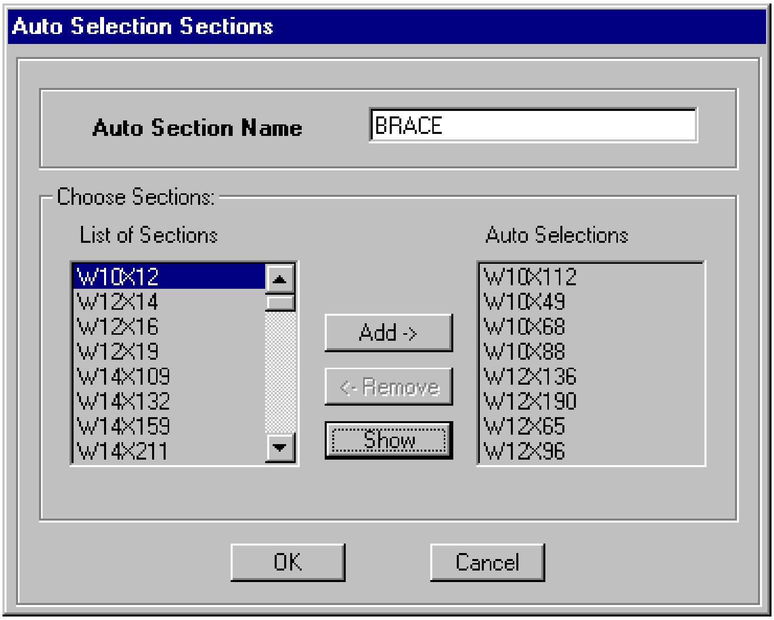
16. Click the **Remove** button. There should now be eight sections in the **Auto Selections** box. The form should look like the figure below.

17. Click **OK** to accept this definition of property "BRACE" and close the **Auto Selection Sections** form.

18. Click **OK** again to accept the changes to the properties and close the **Define Frame Properties** form.

Define Frame Sections, Step 16

Completed ***Auto Selection Sections*** *form for new property, "BRACE".*



Define Deck Section

Note:

Typically deck sections are assigned to floor- or ramp-type area objects.

Deck sections are named combinations of material and geometric cross-sectional properties that can be assigned to area objects. We will take a quick look at the default deck section that we are using, but not make any changes.

1. Select the **Define menu** > **Wall/Slab/Deck Sections** command to open the **Define Wall/Slab/Deck Sections** form. Wall, slab, and deck sections are three different types of properties that can be assigned to area objects.

2. Under **Properties**, select the deck property "DECK1". Recall that this is the property we specified for the floor when we started this model. Click the **Modify/Show Section** button to bring up the **Deck Section** form.

Tip:

Alternatively, you can refer to the User's Manual for help. The User's Manual has an extensive index and table of contents to help you locate information.

3. There is a lot of data specified on this form. Press the **F1** key on your keyboard to open the ETABS Help facility, which will provide a description of all items on this form. Pressing **F1** at any time will provide help on the currently displayed form.

4. Select the **File menu > Exit** command *in the Help window* when you have finished reading the help information.

5. Click **Cancel** to close the **Deck Section** form.

Leave the **Define Wall/Slab/Deck Sections** form open for what we will do next.

Define Wall Section

Wall sections are named combinations of material and geometric cross-sectional properties that can be assigned to area objects. Typically you assign wall sections to wall area objects. We will review the default property, "WALL1", and create a new wall property for our shear walls:

Note:

See the section titled "Wall/Slab/Deck Section Properties" in Chapter 11 of the User's Manual for additional information.

1. If the **Define Wall/Slab/Deck Sections** form is not already open, select the **Define menu > Wall/Slab/Deck Sections** command to open it.

2. Click on "WALL1" in the **Sections** list to highlight it, and then click the **Modify/Show Section** button. This opens the **Wall/Slab Section** form to display the section properties for "WALL1". This form is considerably simpler than the floor form.

3. Review the properties defined for "WALL1", and note in particular that it is 12 inches thick. Click the **Cancel** button to close the **Wall/Slab Section** form.

4. We will now create a second wall section definition. Click the drop-down box on the right side of the **Define Wall/Slab/Deck Sections** form and select "Add New Wall". This again opens the **Wall/Slab Section** form.

5. Note the new **Section Name** is "WALL2", which we keep.

6. Under **Thickness**, change the **Membrane** and **Bending** values both to “8” inches. Normally these two values should be the same.

7. Leave the **Type** set to **Shell** and click **OK** to close the form.

8. Click **OK** to close the **Define Wall/Slab/Deck Sections** form and save our new section definition.

Define Static Loads

Two default static-load cases, “DEAD” and “LIVE”, have already been defined by the program to model dead load and live load, respectively. Currently, case “DEAD” includes the self-weight of all material in the structure, and case “LIVE” includes the 50 psf that we added to the deck.

You can add as many static-load cases as you want. We will now create five more cases, one to represent additional dead load, and four to represent code-defined seismic lateral loads:

Tip:

You should typically only include the self weight in one load case. Otherwise you may end up double-counting the self weight in a load combination. See the section titled “Static Load Cases” in chapter 11 of the User’s Manual for more information.

1. Select the **Define menu > Static Load Cases** command to open the **Define Static Load Case Names** form.

2. Note the two predefined cases:

   - Case “DEAD” is defined to be of type “DEAD” for design purposes, and has a self-weight multiplier of “1”. The self-weight multiplier is a scale factor that multiplies the weight of all material in the structure and applies it as a load in the direction of gravity, which is always –Z.

   - Case “LIVE” is defined to be of type “LIVE” for design purposes, and has a self-weight multiplier of “0”.

3. Click in the edit box labeled **Load**, delete the entry there, and type in the name of our new load case, say “SUPDL”.

4. In the drop-down box labeled **Type**, select design type “SUPER DEAD”. This is superimposed dead load, which is a special type for composite floor design. For other types of design, it will be treated simply as additional dead load.

5. In the **Self Weight Multiplier** edit box, enter the value “0”.
6. Click the **Add New Load** button to actually create the new load case and add it to the table. Don’t forget this step!
7. Click in the edit box labeled **Load**, delete the entry there, and type in the name of another new load case, say “QUAKEX1”.
8. In the drop-down box labeled **Type**, select design type “QUAKE”.
9. In the **Self Weight Multiplier** edit box, enter the value “0”.
10. In drop-down box labeled **Auto Lateral Load**, select “UBC 97”.
11. Click the **Add New Load** button to actually create the new load case and add it to the table. Don’t forget this step!

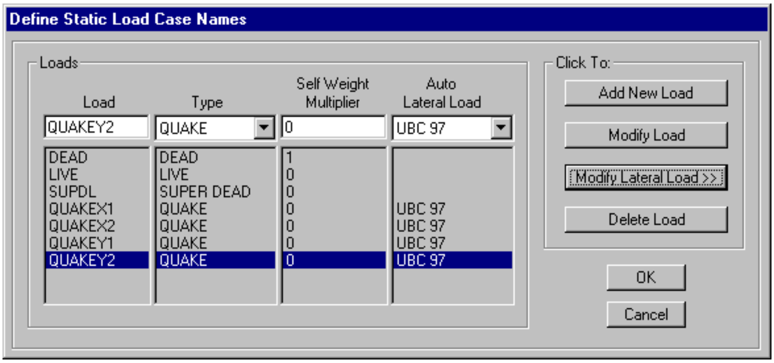
Note:

See the section titled “Defining Automatic Seismic Load Cases” and the subsection titled “1997 UBC Seismic Loads” in Chapter 28 of the User’s Manual for additional information.

12. With case “QUAKEX1” highlighted in the table, click the **Modify Lateral Load** button to open the **1997 UBC Seismic Loading** form.
13. Under **Direction and Eccentricity**, select **X Dir + Eccen Y**. This specifies an X-direction load applied with a positive Y-direction eccentricity. Rigid diaphragms are not needed to apply automated seismic loads, but they are needed to use the automated eccentricities.
14. By default, the eccentricity has the magnitude specified in the **% Eccen** edit box. This value is “0.05” (5%) of the maximum dimension of each diaphragm, measured in the direction of the eccentricity. We will use this default.
15. Under **Factors**, change the **Overstrength Factor, R** to “4.5”. This is the appropriate factor for ordinary moment-resisting frames, which we will consider for our design.
16. Review the rest of the form. We are not going to make any further changes.
17. Click **OK** to close the form.

Define Static Loads, Step 23

Completed ***Define Static Load Case Names*** *form”.*



18. To create the second lateral load, change the entry in the edit box labeled **Load** to “QUAKEX2”, make sure the three values to the right are “QUAKE”, “0”, and “UBC 97”, and click the **Add New Load** button.

19. With case “QUAKEX2” highlighted in the table, click the **Modify Lateral Load** button.

20. Select **X Dir – Eccen Y** under **Direction and Eccentricity**, set the overstrength factor to “4.5”, and then click **OK** to close the form.

21. In a similar fashion, define case “QUAKEY1” with **Direction and Eccentricity** set to **Y Dir + Eccen X** and the overstrength factor set to “4.5”.

22. In a similar fashion, define case “QUAKEY2” with **Direction and Eccentricity** set to **Y Dir – Eccen X** and the overstrength factor set to “4.5”.

23. This gives us four seismic cases in all, two in each lateral horizontal direction, with two different signs of eccentricity. You can verify the definition of each seismic load case by highlighting it, clicking the **Modify Lateral Load** button, reviewing the **1997 UBC Seismic Loading** form, and then clicking **OK** or **Cancel**.

24. Review the **Define Static Load Case Names** form. It should look like the figure above.

25. Click **OK** to accept the new load case definitions and close the form.

Save the Model

Tip:

Don't forget to save your model often.

Since we've made a few changes, now might be a good time to save the model again. We'll use the same file name, overwriting our previously saved model.

1. Select the **File menu > Save** command, or click the **Save Model** button, , on the top toolbar.

2. Because the model has been saved before, the file is saved without any further action from you.

ETABS®

Chapter 3

Modify the Model

This chapter continues the tutorial from Chapter 2. Here we will create the setback and add bracing and shear walls to the first story.

Delete Objects

We are going to create the setback by deleting some beams and columns from the upper stories, and then modifying the floor area to fit the reduced size of the upper stories.

The procedure we will use is typical of how many changes are made to the model:

- Select one or more objects in the model
- Perform an operation on the selected objects.

For now, the operation we will be performing is deleting selected objects. Proceed as follows:

1. Make sure the right display window shows an extruded 3-D view of the structure, with all objects visible.

2. Click the title bar of the left window to make that view active.

3. Click the **Elevation View** button, elev, on the top toolbar, Select **Elevation** "A", and click **OK**.

Note:

See the subsection titled "ETABS Snap Options" in Chapter 12 of the User's Manual for a complete description of the available snap options.

4. Move the mouse cursor very slowly over the model in the left window. As you move toward the intersection of a beam and a column, a red dot appears at the intersection point, flagged with the notation "Point". If this doesn't happen, click the **Snap to Points** button, , on the left toolbar, and try again.

5. Now move the mouse cursor along the beams, and note the presence of points where the secondary beams frame in, even though they can not be seen in this view.

6. Our selection operation will be easier without the snap feature. Click the **Snap to Points** button, , on the left toolbar to turn it off. The red dot will no longer appear as you move toward a beam/beam or beam/column intersection.

Tip:

You must select an object first before performing an action on it, such as, deleting it or making an assignment to it.

7. We will now select the nine column members and 6 beam members above "STORY1". There are many ways to select objects. We will begin with the window-type select:

    - First make sure that ETABS is in selection mode by clicking the **Pointer** button, , on the left toolbar.

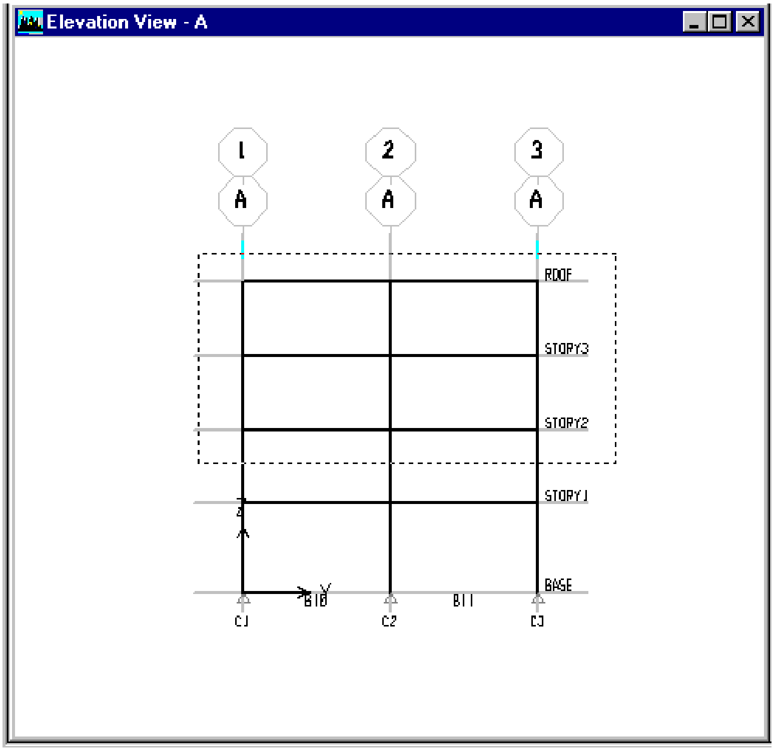
    - Move the mouse cursor to a blank spot in the left window just above and to the left of the top of column "C1".

    - Click and hold down the left mouse button.

    - While holding down the left mouse button, move the mouse cursor to the right of the structure, and down to a point between the floors of "STORY1" and "STORY2". Six column members and six beam members should be fully enclosed within the dashed rectangle that appears

Delete Objects, Step 7

Elevation "A" showing selection rectangle before releasing the mouse button.



while dragging the mouse. It should look like the figure above.

- Release the left mouse button. The twelve members should show as dashed, indicating that they have been selected. This is also indicated in the 3-D view to the right.

- In addition, 27 points at the beam/beam and beam/column intersections are also selected, since these were also enclosed within the selection rectangle. These are indicated by dashed X's. The selection of these points will not affect our upcoming operation.

- Note that column members between "STORY1" and "STORY2" were not selected because they were not *fully* contained within the selection rectangle.

8. Verify your selection by checking the message in the status bar at the lower left corner of the main ETABS window. It

should say "27 Points, 12 Lines selected". If it says anything else, click the **Clear Selection** button, , on the left toolbar, and try Step 7 again. It is a good habit to check this message every time you make a selection.

9. We will continue to add to our selection using another selection method — simply clicking on the objects:

   - One at a time, click on the three column members between "STORY1" and "STORY2". As each object is clicked, the image should become dashed, indicating that it has been selected.

   - If you accidentally select the wrong object, click on it again to deselect or reselect it.

10. Verify the selection status in the lower left corner of the ETABS window. It should say "27 Points, 15 Lines selected". If it says anything else, click the **Clear Selection** button, , on the left toolbar, and try Steps 7 and 9 again.

11. Now that we have our selection, we are ready to perform the operation. Select the **Edit menu** > **Delete** command (or press the **Delete** key). This removes the objects from the model.

12. Note that the deleted members disappear from both display windows.

Note:

The Undo feature works for most actions and assignments in ETABS. Typically Undo doesn't work for definitions. It does not work for the Define menu items and the Edit Grid and Edit Story items.

13. Suppose we had made a mistake. Select the **Edit menu** > **Undo** command (or click the **Undo** button, , on the top toolbar.) Observe that the deleted objects reappear. In general, you can undo all operations performed on the objects in your model back to the last time you saved the file

14. Suppose we had not made a mistake. Select the **Edit menu** > **Redo** command from the menus (or click the **Redo** button, , on the top toolbar.) Observe that the previously selected objects are deleted again. The model is back to where it was at the end of Step 12. In general, you can redo all operations that you undo.

15. With the left window still active, click the **Plan View** button, , on the top toolbar, select **Plan** "ROOF", and click **OK**.

16. We will again use the select-and-delete procedure while working in the "ROOF" plan view. *However, it is important to note that when working on plan views, selection can affect one or more stories, subject to your control.*

> ***Tip:***
>
> *The similar stories feature is only active when you are working in a plan view.*

17. In the story-option drop-down box on the bottom right of the ETABS screen, select "Similar Stories". (If this doesn't work, make sure the left window is active, click the **Clear Selection** button, , on the left toolbar, and try again.) The similar-stories option makes sure that our subsequent operations affect only the stories that are similar to the current plan view, "ROOF". Recall that these were previously defined to be "STORY2" and STORY3".

18. We will now select beams in the left bay using a third selection method:

> ***Tip:***
>
> *You must click the **Set Intersecting Line Select Mode** button each time you want to make an intersecting line selection. You do not remain in the intersecting line select mode after making an intersecting line selection.*

- Click the **Set Intersecting Line Select Mode** button, , on the left toolbar.

- Position the mouse in the left window to a point above beam "B7". See the figure below.

- Click and hold down the left mouse button.

- While holding down the mouse button, move the mouse cursor straight down to a point below beam "B1". The dotted line that appears while dragging the mouse should cross all nine beams in the left bay. It should look like the figure below

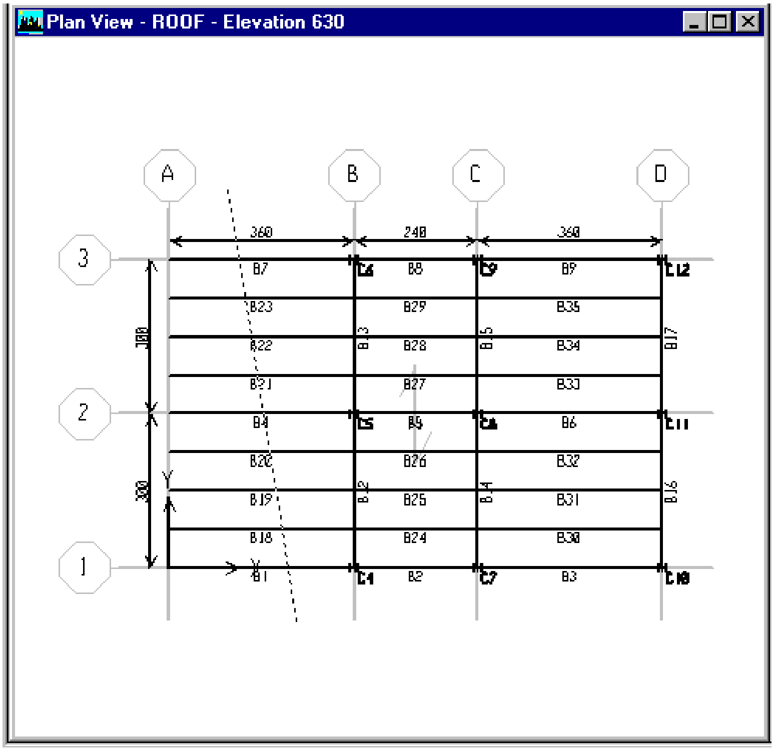
- Release the left mouse button. The nine beam objects should show as dashed, indicating that they have been selected.

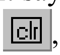
- In addition, the floor area was selected as shown by the dashed line just inside the perimeter of the story.

- Observe in the 3-D view to the right that the upper three stories were affected by this selection.

Delete Objects, Step 18

Plan "ROOF" showing intersecting line selection before releasing the mouse button.



- Verify that the selection status say "27 Lines, 3 Areas selected". If it says anything else, click the **Clear Selection** button, , on the left toolbar, and try this step again.

Note:

See Chapter 13 of the User's Manual for information on selection (and deselection) of objects.

19. We must now deselect the floor area. Move the mouse in the plan view to any point surrounded by four beams, but away from the beams themselves or the corners. Click the left mouse button. The dashed line around the floor should disappear as it is deselected.

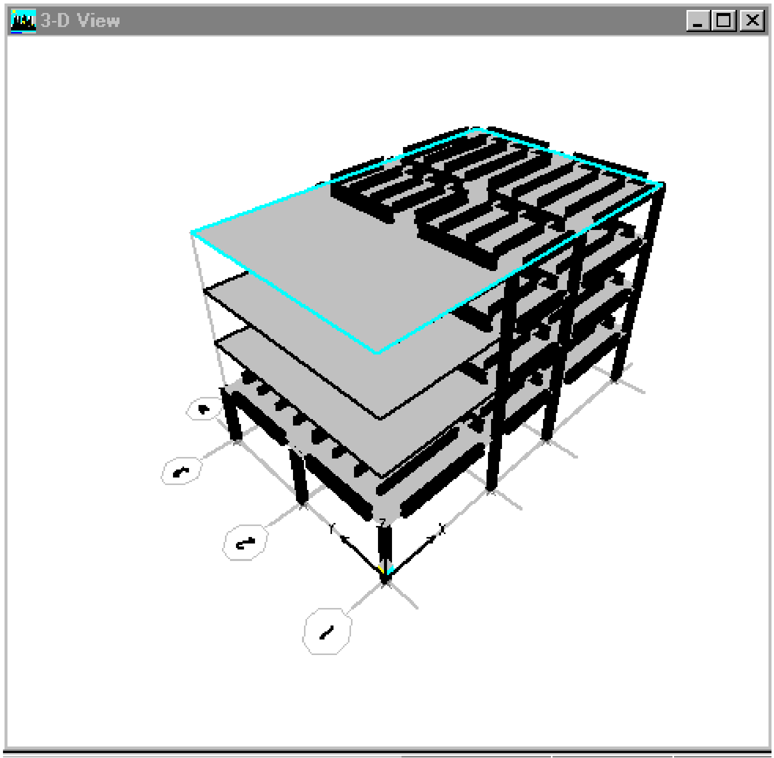
20. If you accidentally select or deselect the wrong object, click on it again to deselect or reselect it.

21. Verify that the selection status says "27 Lines selected".

22. Select the **Edit menu** > **Delete** command (or press the **Delete** key). This removes the selected objects from the model. See the figure below.

Delete Objects, Step 22

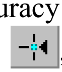
3-D view showing deletion of beam and column objects from the setback.



We have now removed the beams and columns from the setback, but the floors still stick out. We will remedy that next.

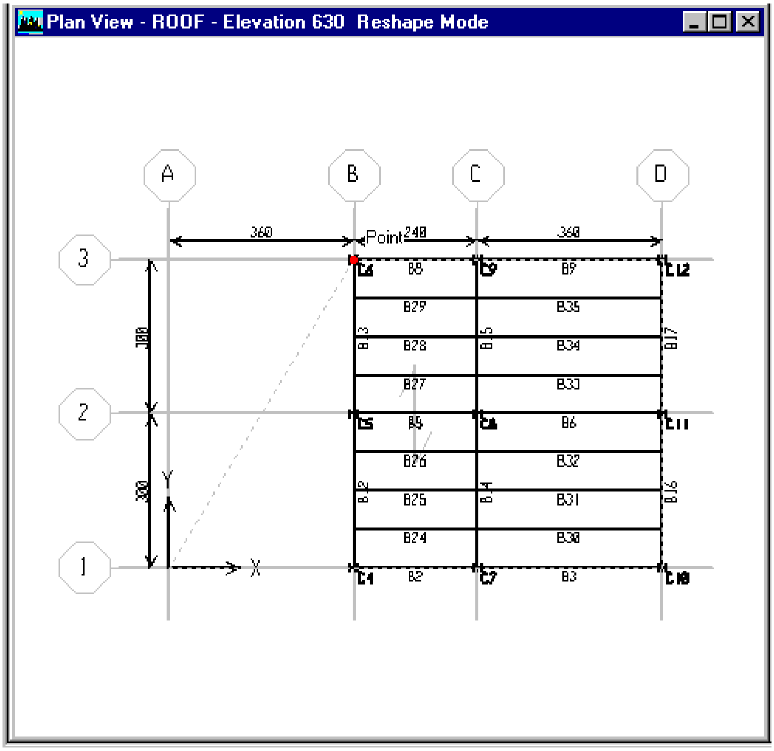
Reshape the Floors

We will now modify the upper floors so that they are only as large as the right two bays. We will use the **reshaper** tool, which is very powerful but somewhat subtle. To learn more about it, consult the ETABS Help facility, or see the *ETABS User's Manual*. Let's begin:

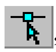
1. We will need to use the snap feature to assure accuracy for our next operation. Click the **Snap to Points** button, , on the left toolbar. Verify that it is working by moving the mouse cursor very slowly over the model. As you move toward the intersection of a beam and a column, a red dot should appear at the intersection point, flagged with the notation "Point".

Reshape the Floors, Step 8

Plan "ROOF" showing the floor after moving one corner.



2. Make sure that "Similar Stories" is still selected at the bottom right of the main ETABS window.

3. Select the **Draw menu** > **Reshape Object** command (or click the **Reshaper** button, , on the left toolbar.)

4. In the plan view on the left, move the mouse to a point on the floor but away from beams and corners.

5. Click the left mouse button. The boundary of the floor area should appear dashed, and four square "handles" should appear at the corners. If this does not happen, click on a blank spot in the left window but outside of the structure, and then try Steps 4 and 5 again.

6. Move the mouse cursor to the upper left handle, located at grid intersection "A-3". The cursor should change to a pair of crosshairs.

7. Click and hold down the left mouse button. Drag the mouse to the right and move the handle until it snaps to point "C6" at grid intersection "B-3".

Tip:

Instead of dragging the handle to its new location you can alternatively right click on the handle and then type in new coordinates for it in the resulting pop-up form.

8. Release the mouse button. The floor area should appear trapezoidal in the plan view on the left. This is shown in the figure above. The 3-D view on the right will not show the results of this operation.

9. Repeat Steps 6, 7, and 8, but move the lower left handle from grid intersection "A-1" to point "C4" at grid intersection "B-1". This completes our reshaping of the three upper floors.

10. Click the **Pointer** button, , on the left toolbar to end reshape mode and return to selection mode.

11. Click the title bar of the right window to make it active, then click the **Refresh Window** button, , on the top toolbar to draw the new extruded view, as shown in the figure below.

This completes the modeling of the setback.

Save the Model under a New Name

Since we've made some major changes, let's save the model again. This time we'll use a different file name, so as not to overwrite file "Tutorial 1" in case we later want to go back to that version of the model:

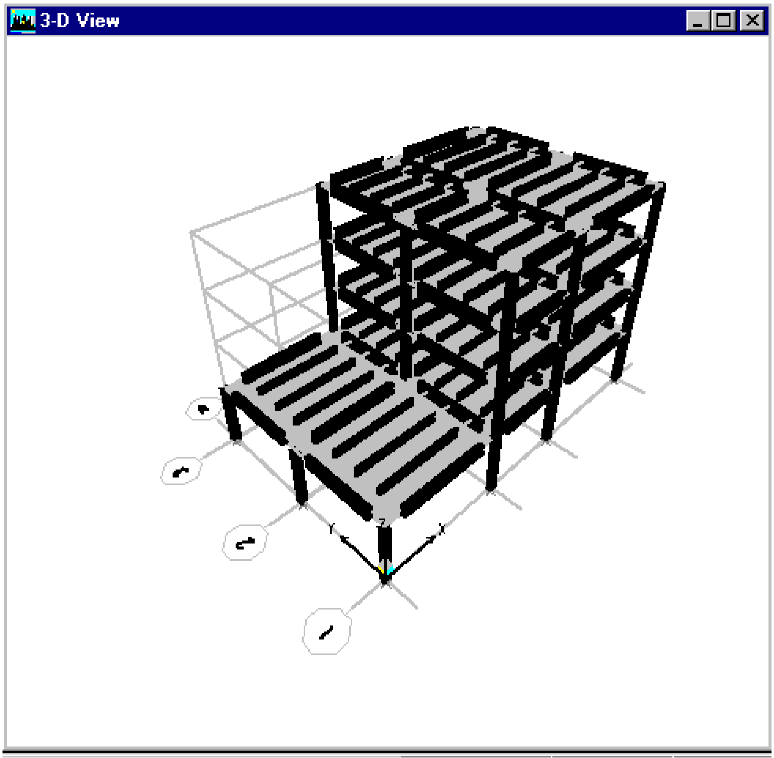
1. Select the **File menu > Save As** command to open the **Save Model File As** form.

2. This should already show the folder where we saved the first file. If not, select the folder where you want to save this file. Then enter a new file name under **File name**, such as "Tutorial 2". ETABS will automatically add the file extension ".EDB" to the file name.

3. Click the **Save** button. This saves the file and closes the form.

Reshape the Floors, Step 11

Extruded 3-D view showing the completed setback.



4. All future **File menu > Save** commands will use the new file name until you change it with another **File menu > Save As** command.

Draw Braces

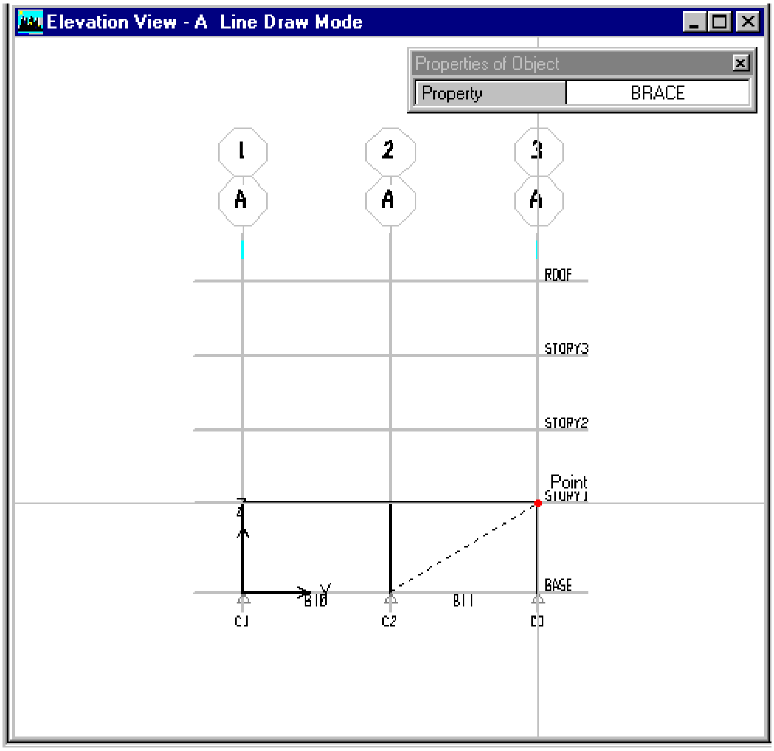
We are now going to add braces at the bottom story level in the Y direction:

1. Click the title bar of the left window to make it active. Click the **Elevation View** button, , on the top toolbar, select **Elevation** “A”, and click **OK**.

2. Make sure that **Snap to Points** is still on. If not, click the **Snap to Points** button, , on the left toolbar.

Draw the Braces, Step 6

Elevation "A" while drawing the first brace, snapped to the second point, and ready to double-click.



Note:

See the section titled "The Two Modes of ETABS" in Chapter 4 of the User's Manual for discussion of the mouse cursor.

3. Select the **Draw menu > Draw Line Objects > Draw Lines** command. The cursor changes to indicate that the program is in draw mode rather than select mode.

4. A small, floating form labeled **Properties of Object** appears. This determines the section properties assigned to the objects to be drawn.

5. Click the lower right data area, scroll to the top, and select "BRACE". Recall that "BRACE" is the name of the auto select section list that we defined earlier.

6. Draw each brace by clicking at the start location of the member, then double-clicking at the end location, as follows:

   - Move the mouse until it snaps to the base of column "C2" (grid line "2") and click.

   - Move the mouse around, and observe how a dashed line is shown indicating where the member will be drawn.

- Move the mouse until it snaps to the top of column “C3” (grid line “3”,) at “STORY1”. See the figure above.

- Double-click. This draws the first brace. Note: A single click would end one member and begin another connected member; the double-click ends the series so we can draw another unconnected member.

- In the same way, draw the second brace from the base of column “C3” to the top of column “C2”.

Note:

As an alternate to double-clicking to finish drawing the brace you can single click and then press either the Enter key or the Esc key on your keyboard.

7. If you make a mistake: click the **Pointer** button, , on the left toolbar to return to selection mode; click the Undo button, ,, on the top toolbar to make the correction(s); then start over at Step 3.

We could use the same steps to draw two more braces in elevation “D”, but let’s try something different instead.

Replicate the Braces

To illustrate the powerful replicate command, we are going to copy the two braces we just drew in elevation “A” to the same location in elevation “D”:

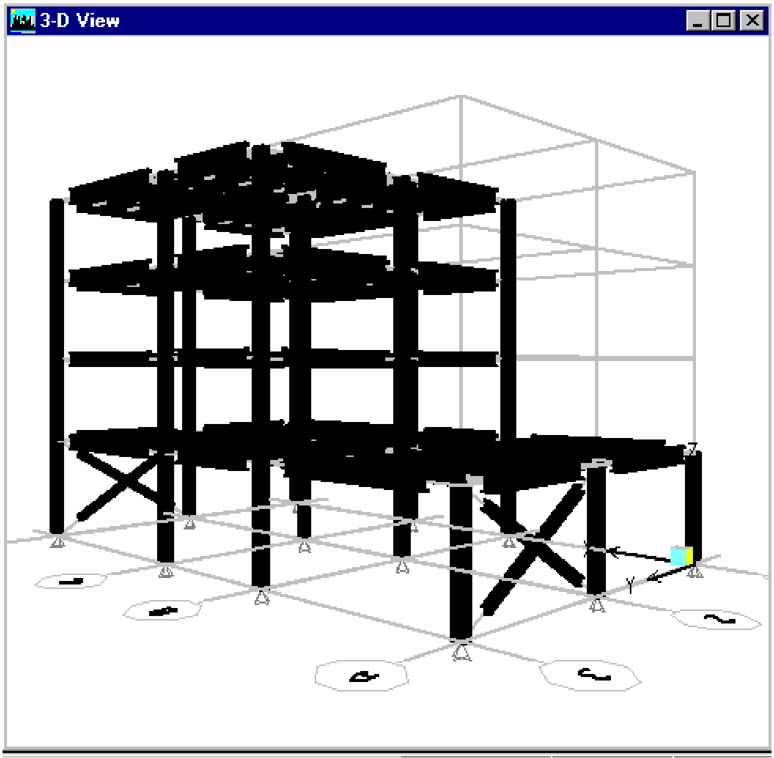
1. Click the **Pointer** button, , on the left toolbar to end draw mode and return to selection mode.

2. Click on each of the two braces to select them.

3. Check the selection status at the bottom left of the main ETABS window. It should say “2 Lines selected”. If not, click the **Clear Selection** button, , on the left toolbar, and try the selection again.

4. Select the **Edit menu > Replicate** command to open the **Replicate** form.

5. The four tabs at the top of the form are for selecting the type of replication to use. Click the **Linear** tab.



Draw the Braces, Step 10

Extruded 3-D view showing the completed braces. Note the change in viewing angle.

6. We are making one copy, and moving it 960 inches in the X direction. Enter “960” (inches) for **dx**, “0” for **dy**, and “1” for **Number**.

7. Make sure the **Delete Original** box is *not* checked.

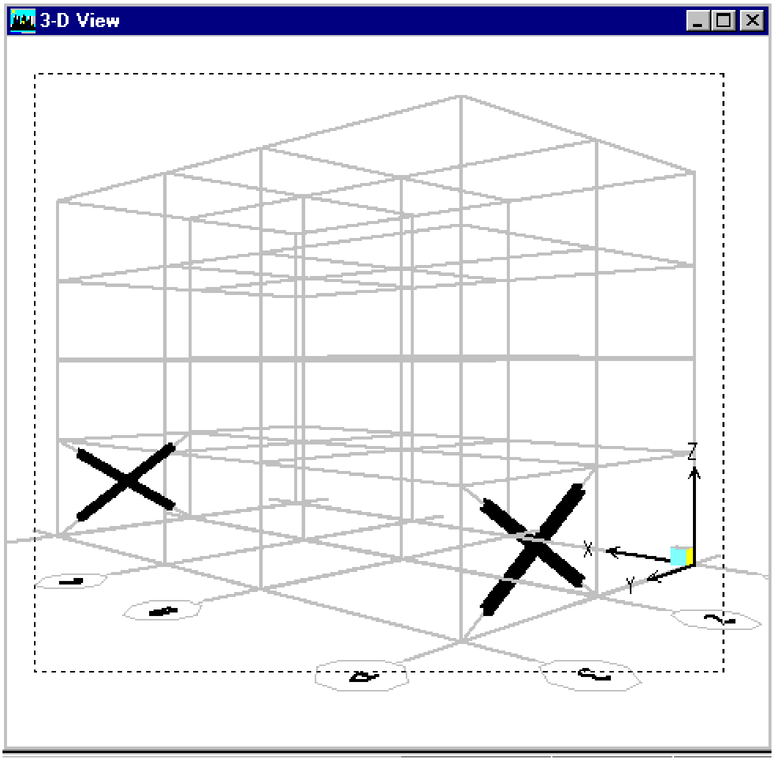
8. Click **OK** to close the form and perform the replication.

9. Note that the 3-D view on the right changes to a line drawing and you can see the replicated braces.

10. Click the title bar of the right window to make the 3-D view active. Click the **Refresh Window** button, , on the top toolbar. Rotate, pan, and zoom as necessary to satisfy yourself that the braces are where you want them to be. You can use the buttons on the top toolbar, or try using commands from the **View menu**. See the figure above.

Pin the Braces, Step 6

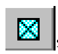
Selecting the braces in the 3-D view. Only the braces are present in the view. The selection rectangle has been dragged around the entire structure, but only the objects present can be selected.



Pin the Braces

By default, all line objects (beams, columns, braces, etc.) are continuously connected at their ends. We will now release the moments at the ends of the braces we just drew, making them pin-connected:

1. This time we'll work in the 3-D view on the right. Make sure this view is still active by clicking on its title bar.

2. Click the **Set Building View Options** button, , on the to toolbar to open the **Set Building View Options** form.

3. Under **Object Present in View**, uncheck the boxes for **Floor (Area)**, **Column (Line)**, **Beam (Line)**, and **Point Objects**.

4. Click **OK** to close the form. Only the braces are shown.

5. Move the mouse into the 3-D view to the outside of one corner of the building.

6. Click and hold down the left mouse button and drag it to the opposite corner of the building so that the entire structure is enclosed in the selection rectangle. See the figure above.

7. Release the mouse button.

Tip:

*Another way to select the braces is simply to use the **Select menu > Select by Line Object Type > Braces** command.*

8. Check the selection status at the bottom left of the main ETABS window. It should say "4 Lines selected". If not, click the **Clear Selection** button, , on the left toolbar, and start over at Step 2.

9. Here's the important point: *When selecting objects using the mouse in a window, only those objects that are present in that view can be selected.* Even though we windowed the whole building, only the braces were present to be selected.

10. Select the **Assign menu > Frame/Line > Frame Releases/Partial Fixity** command to open the **Assign Frame Releases** form.

11. We are going to release the bending moments at both ends. Check both the **Start** and **End** boxes for both **Moment 22** and **Moment 33** (four boxes altogether). Leave the corresponding **Frame Partial Fixity Spring** values as zero.

Tip:

*Alternatively, to clear the display of assigns, you can click the **Show Undeformed Shape** button, , on the top toolbar.*

12. Click **OK** to close the form and make the assignment.

13. Note that the view changes to show the braces as shrunk away from the ends, with green dots indicating the presence of end releases. The extrusions have been turned off.

14. Select the **Assign menu > Clear Display of Assigns** command to turn off the display of the most recent assignment.

15. Click the **Set Building View Options** button, , on the to toolbar to open the **Set Building View Options** form.

16. Click the **Defaults** button. Check the **Extrusion** box under **Special Effects**. Then click **OK** to close the form.

Note that if we had pinned the first two braces before replicating them, the releases would have been replicated also.

By the way, be aware that the braces are *not* connected to each other where they cross.

Define Reference Plane and Reference Lines

Before drawing the shear walls, we are going to define one reference plane and two reference lines. These items will assist us in drawing the shear wall along grid line "1" that includes a door opening.

1. Click the title bar of the left window to make it active. Click the **Elevation View** button, , on the top toolbar, select **Elevation** "1", and click **OK**.

Tip:

Reference planes and lines can help you to accurately locate the objects you draw in your model.

2. Select the **Edit menu > Edit Reference Planes** command to open the **Edit Reference Planes** form.

3. Enter a **Z-Ord** value of "84" and click the **Add** button. This creates a horizontal reference plane with a Z ordinate of 84 inches, and adds it to the table. Click **OK** to close the form.

4. Note that the reference plane is visible between the "BASE" and "STORY1" in both the elevation and the 3-D views. We will later use this reference plane to define the top of the door opening in the shear wall.

5. Select the **Edit menu > Edit Reference Lines** command to open the **Edit Reference Lines** form.

6. Enter an **X-Ord** value of "456" and a **Y-Ord** value of "0". Click the **Add** button. This defines a vertical reference line with location in plan of (X,Y) = (456,0) inches, and adds it to the table.

7. Now enter an **X-Ord** value of "504" and leave the **Y-Ord** value as "0". Click the **Add** button. This defines a second vertical reference line, and adds it to the table.

8. Make sure that both reference lines are present in the table, and then click **OK** to close the form.

9. Note that the reference lines are visible along grid line “1” between grid lines “B” and “C” in both the elevation and the 3-D view. We will use these reference lines next to define the edges of the door opening in the shear wall.

Draw Shear Walls

We are now going to add walls at the bottom story level in the X direction:

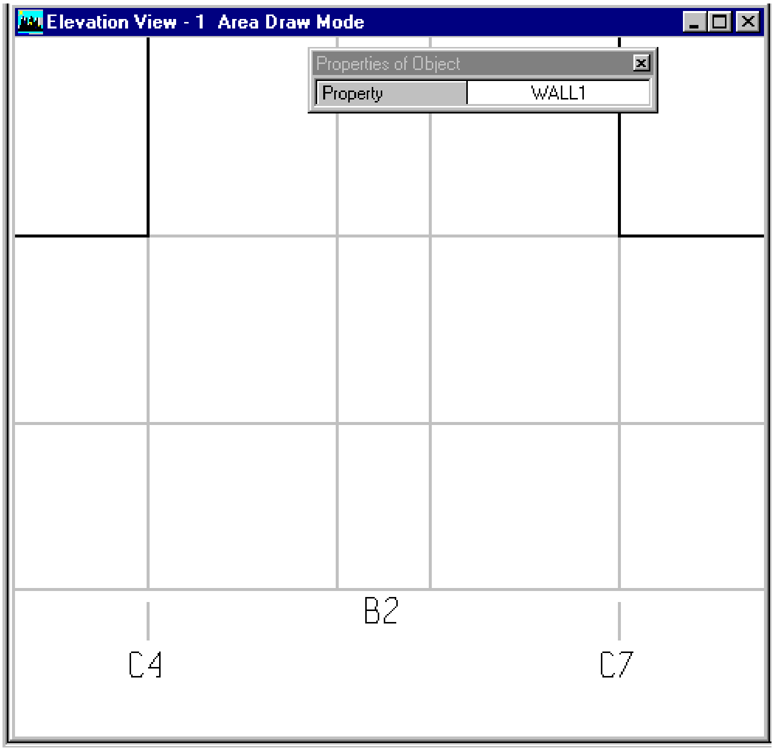
1. Click the title bar of the left window to make it active. Click the **Elevation View** button, , on the top toolbar, Select **Elevation** “3”, and click **OK**.

> ***Tip:***
>
> *You can also initiate drawing of objects by clicking the appropriate button on the side toolbar. See the back inside cover of the User’s Manual for a short description of the function of each toolbar button.*

2. Select the **Draw menu > Draw Area Objects > Draw Rectangular Areas** command. The cursor changes to indicate that the program is in draw mode rather than select mode.
3. A small, floating form labeled **Properties of Object** appears. This determines the section properties assigned to the objects to be drawn.
4. Click the lower right data area, scroll to the top, and select “WALL2”, the new property (8 inches thick) we defined earlier.
5. Make sure that snap-to-points is on. If not, click the **Snap to Points** button, , on the left toolbar to turn it on.
6. We will draw the solid wall by clicking at one corner of the wall, then clicking at the opposite corner:
   - Move the mouse until it snaps to the base of column “C6” (grid line “B”) and click.
   - Move the mouse until it snaps to the point where column “C9” (grid line “C”) intersects the floor at “STORY1”, and click again.
   - A red rectangle appears to show the extent of the wall. It should be one bay wide and one story tall.

Draw Shear Walls, Step 10

Elevation "1" after zooming in, ready to draw the shear wall with the door opening. Note the six areas created by grid lines "B" and "C", story levels "BASE" and STORY1", the reference plane, and the two reference lines.



7. Click the **Move Down in List** button,  , to twice to display elevation "1".

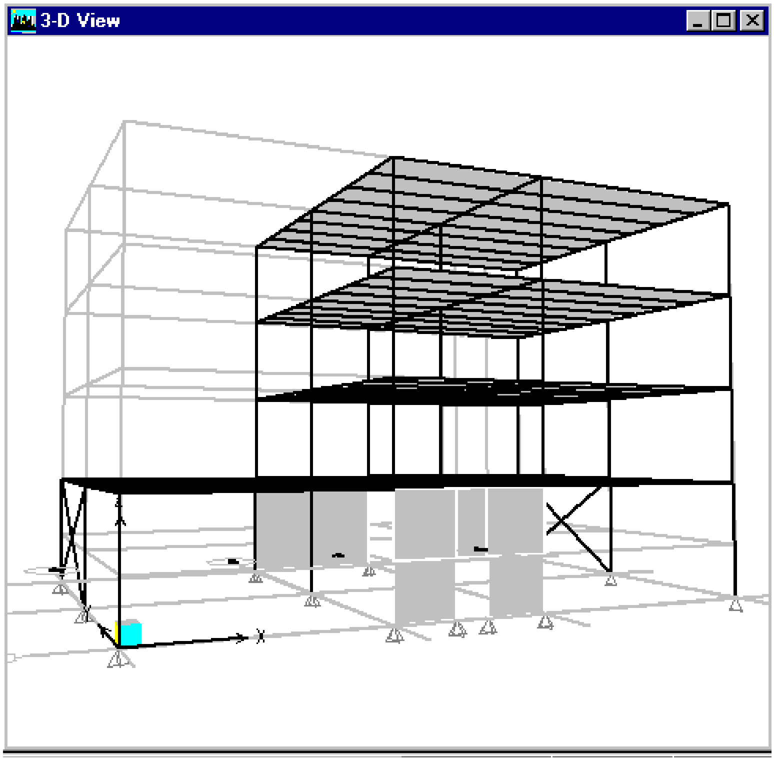
8. Select the **Draw menu > Draw Area Objects > Create Areas at Click** command.

9. In the floating form labeled **Properties of Object,** click the lower right data area and select "WALL1", the default property (12 inches thick) we reviewed earlier.

10. Observe how the bay along grid line "1" that is bounded by grid lines "B" and "C" and by story levels "BASE" and "STORY1" is broken up into six areas by the reference plane and reference lines. Use the **Rubber Band Zoom** button, , on the top toolbar to zoom into this bay. Make sure all six areas in this bay are in the view, as shown in the figure above.

Draw Shear Walls, Step 16

3-D view, using object fill, showing the completed shear walls.



11. The bottom-center area represents the door opening. The other five areas around the door represent the shear wall.

12. Click once in the center of each of the five areas around the door to draw five area objects representing the shear wall.

13. Click the **Pointer** button, , on the left toolbar to end draw mode and return to selection mode.

14. Change the 3-D view in the right window to show extrusions, then rotate, pan, and zoom to satisfy yourself that the walls are where you want them to be. If not, use the **Undo** command as necessary and try again.

15. Let's check the walls in the 3-D view. This time we'll try **object fill** instead of extrusion:

    - Click the title bar of the right window to make the 3-D view active.

- Click the **Set Building View Options** button, , on the top toolbar.
- Under **Special Effects**, *uncheck* the **Extrusion** box.
- Under **Special Effects**, check the **Object Fill** box.
- Click **OK** to close the form.

16. Rotate, pan, and zoom to satisfy yourself that the walls are where you want them to be, as shown in the figure above. If not, use the **Undo** command as necessary and try again.

Assign Pier and Spandrel Labels

We are now going to assign pier and spandrel labels to portions of our shear walls. This is necessary so that we can view forces associated with the wall piers and spandrels and so that we perform simplified wall design for them.

1. Click the title bar of the left window to make it active. Click the **Elevation View** button, elev, on the top toolbar, select **Elevation** “3”, and click **OK**.
2. Click once on the wall to select it. Verify that the selection status indicates “1 Area selected”.
3. Select the **Assign menu > Shell/Area > Pier Label** command to display the **Pier Names** form. Note that a default pier label, “P1”, is already defined.
4. Before performing our assignment, we are going to define to new labels for future use:
   - Type “P2” in the edit box under **Wall Piers** and click the **Add New Name** button to add this label to the list.
   - Type “P3” in the edit box and click the **Add New Name** button.
   - Add label “P4” in a similar manner.

- Verify that there are four labels in the table, plus "NONE".

Tip:

You assign pier and spandrel labels so that ETABS recognizes a group of wall-type area objects as a pier or spandrel. ETABS can neither display pier and spandrel forces, nor design piers and spandrels, until you assign pier and spandrel labels.

5. Make sure the "P4" label is highlighted, and then click **OK** to close the form and assign the pier label "P4" to the selected wall area.

6. Click the **Move Down in List** button, , twice to display elevation "1".

7. Zoom in to the bay with the shear wall and door opening.

8. Click the wall object on the left side of the door opening to select it. Select the **Assign menu > Shell/Area > Pier Label** command to open the **Pier Names** form. Highlight the "P1" label by clicking it, and then click **OK** to make the assignment.

9. Similarly, assign pier label "P2" to the wall object on the right side of the door opening.

10. Select all three of the wall objects above the opening, then assign pier label "P3" to them. Note that pier "P3" consists of three separate wall objects. The selection status should say "3 Areas selected". The forces in all three areas will be summed together when reporting forces for pier "P3".

11. Select the center wall object above the door.

12. Select the **Assign menu > Shell/Area > Spandrel Label** command to display the **Spandrel Names** form. Note that a default spandrel label, "S1", is already defined.

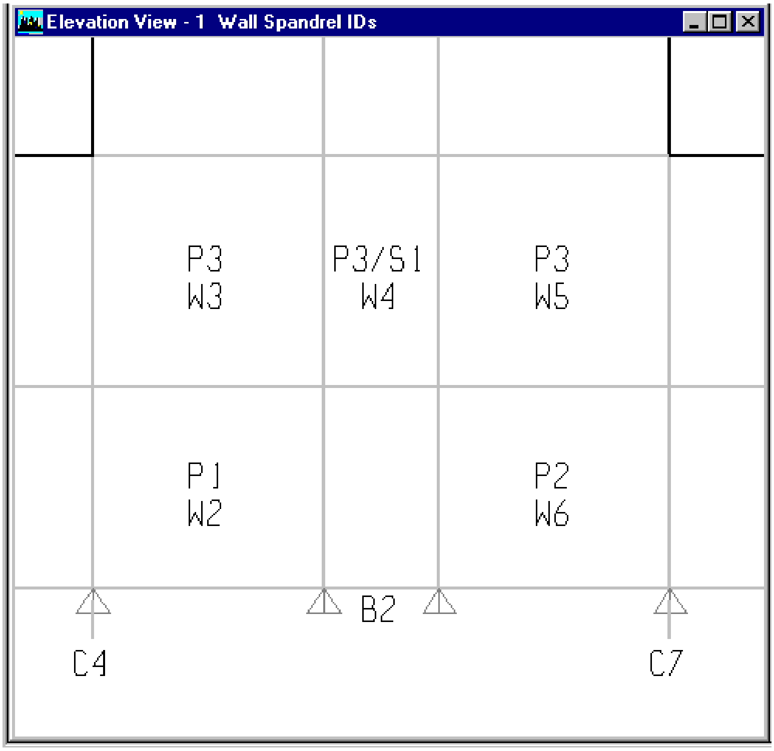
13. Highlight the "S1" label by clicking it, and then click **OK** to make the assignment.

14. Observe that the object above the door is part of a pier and a spandrel. Each wall object may be part of, at most, a single pier and a single wall.

15. When you are done, the display should show the pier and spandrel assignments as shown in the figure below. If not, use the **Undo** command as necessary and try again. Note that the labels "W2" to "W6" are the labels of the wall objects

Assign Pier and Spandrel Labels, Step 15

Elevation "1", zoomed in, showing the pier and spandrel labels assigned to the wall with the door opening.



themselves. Your wall labels may differ from the figure, depending on the order in which you drew the walls.

16. Select the **Assign menu > Clear Display of Assigns** command.

Change Column Orientations

We will now change the orientation of some of the columns. By default, the major axis (local axis 2) of the columns is parallel to the X direction. We will change some columns so that the major axis is parallel to Y. These columns are:

- Column "C7" at grid intersection "C-1".
- Column "C5" at grid intersection "B-2".
- Column "C11" at grid intersection "D-2".

- Column “C9” at grid intersection “C-3”.

Each of these columns consists of four members, one for each of the stories.

We will try selecting the columns in both elevation and plan views:

1. Make sure that ETABS is in selection mode by clicking the **Pointer** button, , on the left toolbar.
2. Click the title bar of the left window to make that view active.
3. Click the **Elevation View** button, , on the top toolbar, select **Elevation** “2”, and click **OK**.
4. Select the four members of column “C5” (grid line “B”) by clicking on them, one-by-one.
5. Select the four members of column “C11” (grid line “D”) by dragging a selection window around them.
6. Verify that the selection status says “5 Points, 8 Lines selected” (the points came from the window selection.) If it says anything else, click the **Clear Selection** button, , on the left toolbar, and try the selection again.
7. Select the **Assign menu > Frame/Line > Local Axes** command to open the **Axis Orientation** form.
8. For vertical columns, the angle is measured clockwise (when viewed from above) from the X axis to the column’s major axis (i.e., its local 2 direction). Select **Angle** and enter a value of “90”, or select **Column major direction is Y**.
9. Click **OK** to accept the assignment and close the form.
10. The display in the left window will change to show the local axes for all line objects in the elevation view. Observe the following:
    - For each line object, the colors Red, White, and Blue *always* correspond to local axes 1, 2, and 3, respectively.

Note:

See the subsection titled “Local Axes Assignments to Line Objects” in Chapter 14 of the User’s Manual for additional information (including sketches of local axes.)

- The local 1, 2, and 3 axes form a right-handed coordinate system.

Tip:

In ETABS the local axes 1, 2 and 3 are always red, white and blue, respectively. One way to remember this is that the local axes are the same colors as the American flag: red, white and blue.

- The local 1 axis (red) always points along the length of the line object, from start to end.
- For columns, the default orientation is for the local 2 axis to point along +X, hence the local 3 axis points along +Y (pointing into the screen.)
- For column "C7", we changed the local 2 axis to point into the screen along +Y, so the local 3 axis points along –X.
- For beams, the default orientation is for the local 2 axis to point upward, along +Z. The direction of the local 3 axis depends on the axial direction of the beam. For the beams in this view the local 3 axis points toward us, in the –Y direction.

11. Now let's try this in a plan view. Click the **Plan View** button, , on the top toolbar, select any **Plan** except "BASE", and click **OK**.

Tip:

If the active window is not a plan view then the story-option drop-down box is not active.

12. In the story-option drop-down box on the bottom of the ETABS screen, select "All Stories". This makes sure that our subsequent operations affect all four story levels. (If this doesn't work, make sure the left window is active, click the **Clear Selection** button, , on the left toolbar to make sure nothing is selected, and try again)
13. Click on point "C7" at grid intersection "C-1". Note the status bar indicates "5 Points selected". Windowing around the point gives the same result. This is because, *by default, selection in a plan view does not include the columns.*
14. Try the selection again, this time holding down the **Ctrl** key while you click on point "C7". This opens the **Selection List** form showing all the objects that can be selected at that point. This includes the point, the three beams, the column, and the floor deck.

Note:

See the section titled "Using the Mouse" in Chapter 4 of the User's Manual for discussion of selecting objects that are located one on top of another.

15. Click "Column C7" in the selection list. The status bar at the bottom of the ETABS window should show "4 Lines selected" (any points are OK).

16. Using the **Ctrl** key again, click on column "C9" at grid intersection "C-3". Click "Column C9" in the selection list that appears. The status bar should now indicate "8 Lines selected" (any points are OK).

17. Repeat Steps 7, 8, and 9 above to change the orientation of the two selected columns.

18. Review the local axes of the columns in this plan view, zooming in if necessary.

19. To get a better view of the columns, do the following:

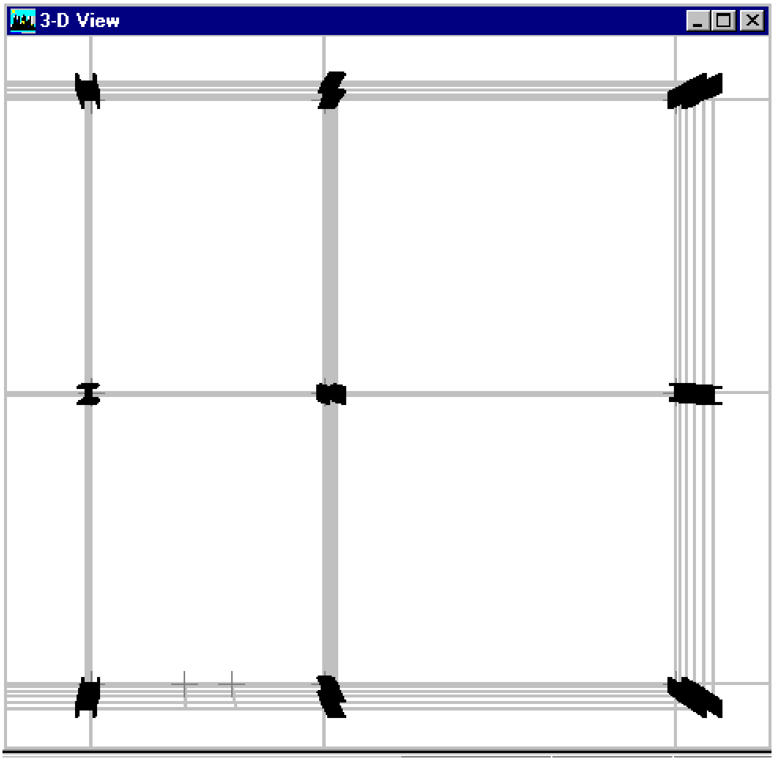
    - Click on the title bar of the right window to make the 3-D view active.
    - On the top toolbar, click the **Set Building View Options** button, .
    - Click the **Defaults** button.
    - Under **Special Effects**, check the **Extrusion** box.
    - Under **Object Visibility**, *uncheck* the **Floor (Area)**, **Wall (Area)**, **Beam (Line)**, and **Brace (Line)** boxes.
    - Click **OK**. The 3-D view should now show the extruded view of the columns only.

20. Select the **View menu > Set 3D View** command to open the **Set 3D View** form. This form give you more control over the orientation of a 3-D view:

    - Set the **Plan** angle to "270". This means we are looking from the –Y direction.
    - Set the **Elevation** angle to "90". This means we are looking from above (+Z).

Change Column Orientations, Step 21

3-D view from above showing the column orientations in the four-story portion of the building.



- Set the **Aperture** angle to "10". This reduces the perspective effect from the default value of "60".

- Press the **F1** key to open the help facility for more information about the meaning of these angles.

- Select the **File menu > Exit** command *on the Help form* to close the help facility.

- Click **OK** to close the **Set 3D View** form and draw the 3-D view.

21. Pan and zoom in the 3-D view to clearly see all the column orientations. See the figure above.

22. To return to the original 3-D view, click the **3D View** button, 

.

23. To return to the extruded view of the whole building, click the **Set Building View Options** button, , click the **Defaults** button at the bottom of the **Set Building View Options** form, check the **Extrusion** box, and click **OK**.

Assign Floor Load

We will assign floor load to the superimposed dead load case, and reduce the load acting on the roof. Recall that all four stories had 50 psf added to load case “LIVE”. First we will assign 30 psf to case “SUPDL” for all floors but the roof. Then we will set the load in cases “SUPDL” and “LIVE” to 20 psf for story “ROOF”.

1. Click the title bar of the left window to make that view active.

2. Click the **Plan View** button, , on the top toolbar, select **Plan** “ROOF”, and click **OK**.

3. In the story-option drop-down box on the bottom of the ETABS screen, select “One Story”. This makes sure that our subsequent operations only affect the “ROOF” story. (If this doesn’t work, make sure the left window is active, and click the **Clear Selection** button, , on the left toolbar to make sure nothing is selected, and try again)

4. We will now select all the floors. Rather than using the mouse:

   - Select the **Select menu > Select by Wall/Slab/Deck Sections** command. This opens the **Select Sections** form.

   - Under **Select**, click “DECK1” to highlight it, and then click **OK**. Recall that “DECK1” is the name of the property assigned to all floors.

   - Note that all floors are indicated as being selected in both views by dashed lines just inside their perimeters.

- Verify that the selection status indicates “4 Areas selected”. If not, clear the selection and try again.

5. We will now deselect the “ROOF” floor. You could do this by clicking the floor area in the plan view of the “ROOF”, but let’s try something different:

   - Select the **Select menu > Deselect > by Story Level** command. This opens the **Select Story Level** form.

   - Under **Select**, click “ROOF” to highlight it, and then click **OK**.

   - Verify that the selection status indicates “3 Areas selected”. If not, clear the selection and try again, starting with Step 4.

6. Set the units to pounds and feet, “lb-ft”, using the drop-down box in the lower right corner of the ETABS screen.

7. Select the **Assign menu > Shell/Area Loads > Uniform** command to open the **Uniform Surface Loads** form.

8. Under **Load Case Name**, select “SUPDL”.

9. Under **Uniform Load**, enter “30” for the **Load** and select “Gravity” for the **Direction**.

10. Make sure that the **Options** are set to “Replace Existing Load”.

11. Click **OK** to make the assignment and close the form.

12. We will now select the “ROOF” level by another method:

    - Select the **Select menu > Select on XY Plane** command.

    - In the 3-D view in the window on the right, move the mouse until it snaps to any point on story level “ROOF”.

    - Click the point to select all objects at this elevation.

- Verify that the selection status indicates "27 Points, 24 Lines, 1 Areas selected". If not, clear the selection and try again.

- Note that even though our selection includes point, lines, and areas, the assignment we are going to do next affects only areas, so the inclusion of the other objects is unimportant.

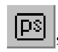
13. Select the **Assign menu > Shell/Area Loads > Uniform** command to open the **Uniform Surface Loads** form.

14. Under **Load Case Name**, select "SUPDL".

15. Under **Uniform Load**, enter "20" for the **Load** and select "Gravity" for the **Direction**.

16. Make sure that the **Options** are set to "Replace Existing Load".

17. Click **OK** to make the assignment and close the form.

18. Note that the selection is cleared after making the assignment. Click the **Restore Previous Selection** button, , on the left toolbar to repeat the same selection.

19. Select the **Assign menu > Shell/Area Loads > Uniform** command to again open the **Uniform Surface Loads** form.

20. Under **Load Case Name**, select "LIVE" this time, verify that the other data is the same, and click **OK**.

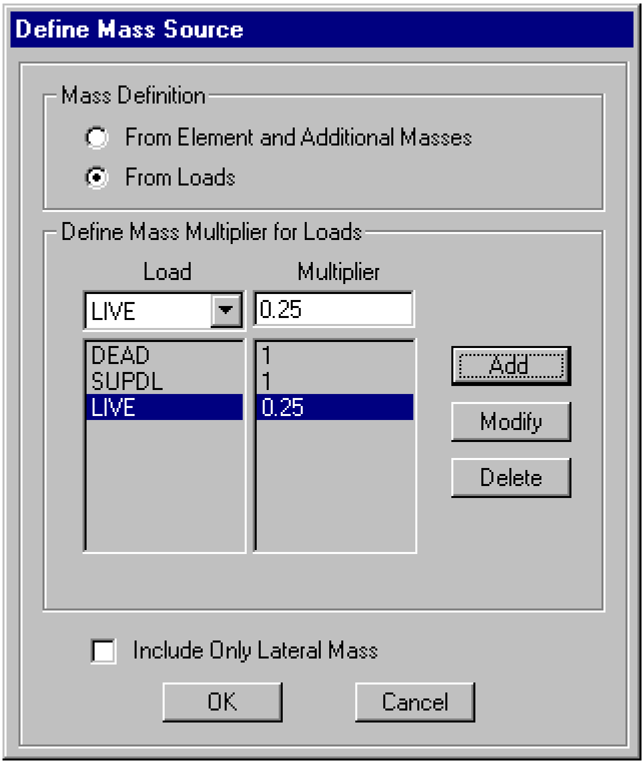
Define Mass Source

Mass and weight are considered as separate properties in ETABS. Mass is used for inertia in dynamic analyses and to define acceleration loads for seismic ground motion. Weight is used for static gravity loading.

When defining material properties in ETABS, you specify mass density and weight density separately, although normally they are related by the value of gravitational acceleration in the current length units (e.g., 386.4 in/s^2 or 9810 mm/s^2.)

Define Mass Source, Step 5

Completed ***Define Mass Source*** *form.*



In our example, load case "DEAD" includes the weight of all the materials in the model, case "SUPDL" accounts for the weight of the nonstructural components of the building, and case "LIVE" accounts for the weight of the live load.

So far, we have only included mass from the materials in the model. We still need to account for the mass of the nonstructural components and the live load. There are two principal options for accomplishing this:

- Assign additional mass to point, line, and/or area objects.

- Specify that all mass be computed from the gravitational component of one or more static load cases.

We will use the second option here, and specify that our mass be computed from the sum of 100% of case "DEAD", plus 100% of case "SUPDL", plus 25% of case "LIVE":

1. Select the **Define menu > Mass Source** command to open the **Define Mass Source** form.

2. Under **Mass Definition**, select **From Loads**.

3. Under **Define Mass Multiplier for Loads**:

   - Select "DEAD" in the **Load** drop-down box, enter "1" in the **Multiplier** edit box, and click the **Add** button to add the load to the table.

   - Select "SUPDL" in the **Load** drop-down box, enter "1" in the **Multiplier** edit box, and click the **Add** button to add the load to the table.

   - Select "LIVE" in the **Load** drop-down box, enter "0.25" in the **Multiplier** edit box, and click the **Add** button to add the load to the table.

4. Leave the **Include Only Lateral Mass** box unchecked.

5. Make sure that the form looks like the figure above.

6. Click **OK** to close the form.

This completes our initial model.

Save the Model

Since we've made more changes, let's save the model again. You can overwrite the current file, "Tutorial 2", or save the file as a third, new name.

Chapter 4

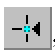
Analyze the Model

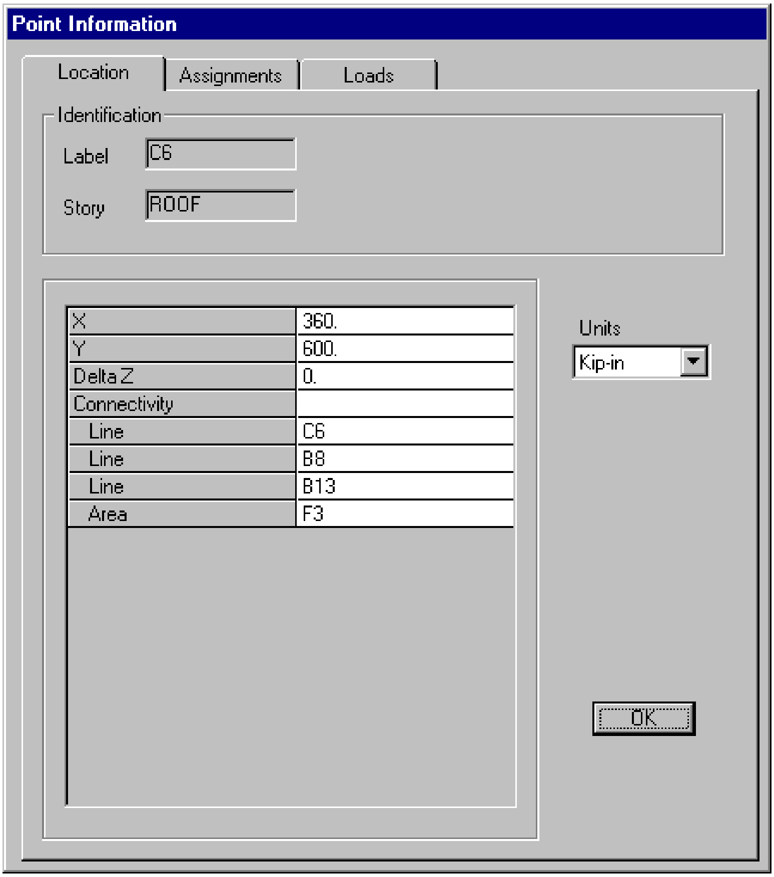
This chapter continues the tutorial from Chapter 3. Now that our model is complete, we will perform the analysis to determine the response of the building to its loading. The actual analysis proceeds quickly. We will spend most of our time in this chapter looking at object information, both before and after the analysis.

View Object Information

As a general rule, clicking the right mouse button while pointing to an ETABS object will provide information about that object. The type of information provided will depend on the type of display shown in the window (undeformed model, displaced shape, moment diagram, stress contours, etc.).

Let's first work in the left window, which should show the 2-D plan view of story "ROOF":

1. Make sure the point-snap option is on. If not, click the **Snap to Points** button, , on the left toolbar.



View Object Information, Step 3

*Point Information for point "C6" at level "ROOF", showing the **Location** tab.*

2. Move the mouse toward the upper left corner of the elevation, at the top of column "C6" (grid intersection "B-3"). ***Note – the labels in your model may differ!*** While the red dot is showing, click the right mouse button. This opens the **Point Information** form. This form is not editable; it is for display purposes only.

Note:

See the section titled "Right Click Information for Point Objects" in Chapter 25 of the User's Manual for additional information.

3. Three tabs are available at the top of the form. See the figure above. With the **Location** tab active, you can see the following:

   - The point object is identified by **Label** "C6" and **Story** "ROOF" (all five point objects in column "C6" are identified by the **Label** "C6"; only the **Story** value differs between them.)

- The coordinates **X** and **Y** are measured from the origin. Coordinate **Delta Z** is measured from the story level. Note that the current units should still be “lb-ft”.

- This point is connected to column line object (“C6”), to two beam line objects (“B8” and “B13”), and to one floor area object (“F3”.)

4. Click the **Assignments** tab at the top of the form. You can see that this point is part of the rigid diaphragm “D1” that was created for this story, that it is part of the default group “ALL”, and that no other assignments have been made. These items need not concern us now.

5. Click the **Loads** tab at the top of the form. No loads have been assigned to the point object.

6. Click **OK** to close the **Point Information** form.

7. Move the mouse just rightward to the middle of beam “B8”, so that no red dot appears at any point object. ***Note – the labels in your model may differ!***

Note:

See the section titled “Right Click Information for Line Objects” in Chapter 24 of the User’s Manual for additional information.

8. Click the right mouse button. This opens the **Line Information** form. With the **Location** tab active, you can see the following:

   - The line object is identified by **Label** “B8” and **Story** “ROOF”.

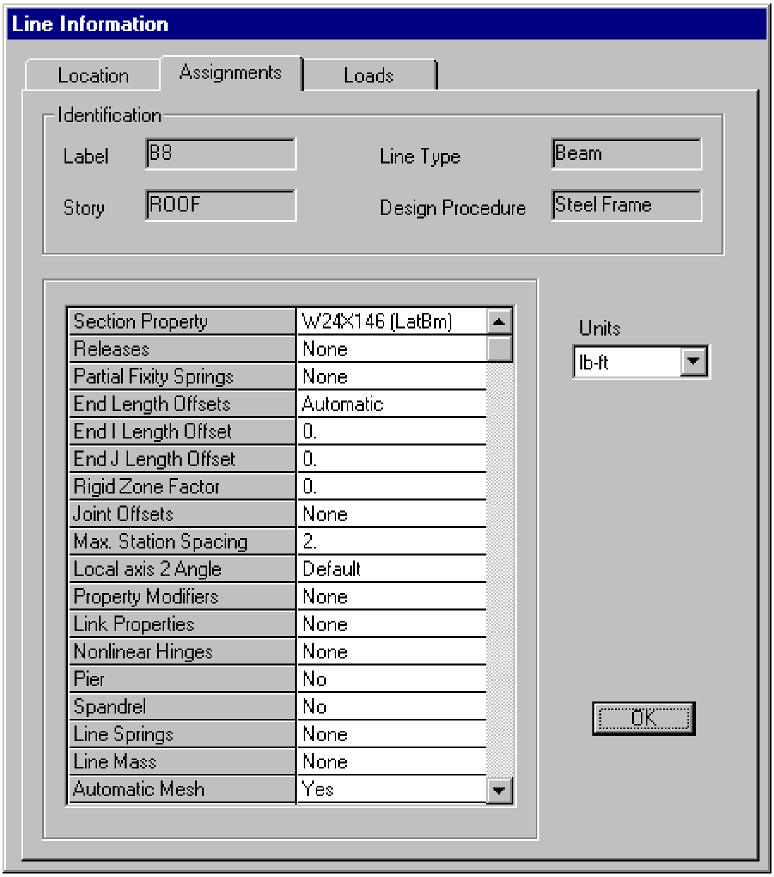
   - The length of the line is given, along with the identification and coordinates of the two points that define the ends of the line.

9. Click the **Assignments** tab at the top of the form. You can see that a large number of assignments are possible for line objects, as shown in the figure below.

10. Note in particular that the **Section Property** is given as “W24X146 (LatBm)”. This means that the program has chosen one section to be used for analysis purposes from the available sections in auto select section list “LatBm”. This initial choice is the section with the median weight. This assignment may change after design.

View Object Information, Step 9

*Line Information for beam "B8" at level "ROOF", showing the **Assignments** tab.*



11. Click the **Loads** tab at the top of the form. No loads have been directly assigned to the line object. Recall that we assigned the loads to the floor area objects.

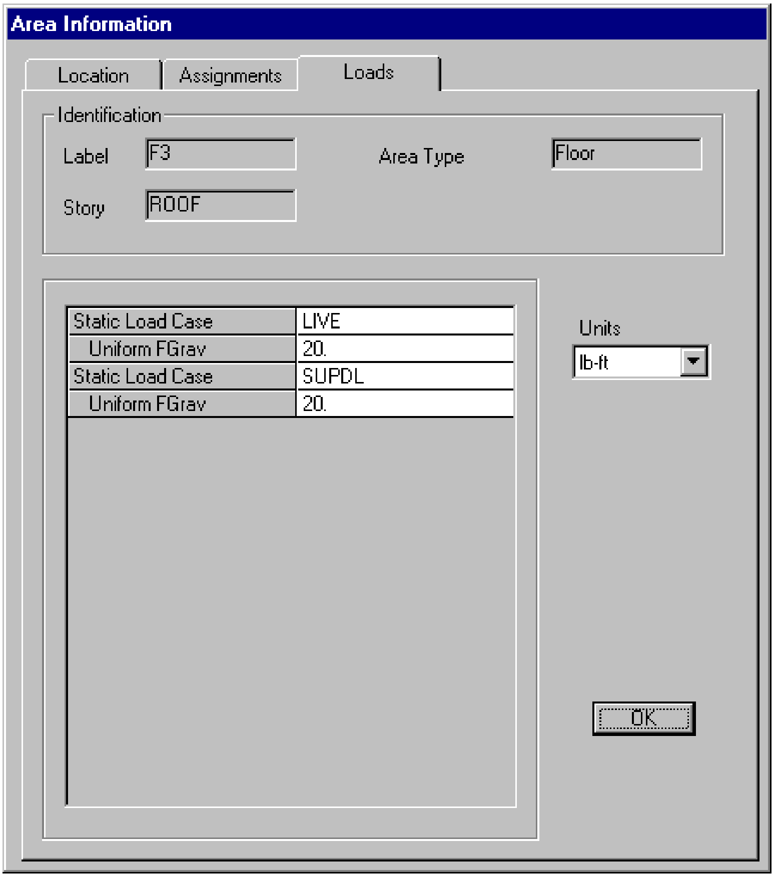
12. Click **OK** to close the **Line Information** form.

13. Move the mouse slightly downward to the empty space between beams "B8" and "B29". This point is part of the floor "F3". ***Note – the labels in your model may differ!***

14. Click the right mouse button. This opens the **Area Information** form. With the **Location** tab active, you can see the following:

View Object Information, Step 16

*Area Information for floor “F3” at level “ROOF”, showing the **Loads** tab.*



Note:

See the section titled “Right Click Information for Area Objects” in Chapter 23 of the User’s Manual for additional information.

- The area object is identified by **Label** “F3” and **Story** “ROOF”. (The other similar floors are also labeled “F3”; the floor at “STORY1” is labeled “F1”.)
- Various geometric properties are given, including the area, perimeter, centroid, and polar moment of inertia. Also, the number of points that define the area is given, along with their identification and coordinates.

15. Click the **Assignments** tab at the top of the form. You can see that a number of assignments are possible for area objects. Note, in particular:

- The **Section Property** is “DECK1”.

- This object is part of diaphragm constraint “D1” for this story.

16. Click the **Loads** tab at the top of the form. Here we can see that the uniform load values we assigned to load cases “SUPDL” and “LIVE” are both 20 psf, as shown in the figure above.

17. Click **OK** to close the **Area Information** form.

18. Change the plan view to “STORY3”, right click on the floor area, and check that the loads are 30 psf in case “SUPDL” and 50 psf in case “LIVE” (it won’t be exactly “50” since we specified the load in kip-in units.)

19. Move the mouse toward point “C9” so that the red dot appears there.

20. If we click the right mouse button now, we will get information on point object “C9”. Instead, hold down the **Ctrl** key on the keyboard while clicking the right mouse button. This opens the **Selection List** form that allows us to choose among all the objects that have any part located at that point. This includes the point itself, three beams, a column, and the floor. Click on the line that reads “COLUMN C9”.

21. This opens the **Line Information** form. With the **Location** tab active, you can see the following:

    - The line object is identified by **Label** “C9” and **Story** “STORY3” (all four line objects in column “C9” are identified by the **Label** “C9”; only the **Story** value differs between them.) Note that each column line object is considered to be part of the story above it.

    - The length of the line is given, along with the identification and coordinates of the two points that define the ends of the line. The column for “STORY3” runs from “STORY2” to “STORY3”.

22. Click the **Assignments** tab at the top of the form. Note in particular that the **Local axis 2 Angle** indicates the change we made earlier to the orientation of this column.

23. Click the **Loads** tab at the top of the form. No loads have been assigned to the line object.

24. Click OK to close the **Line Information** form.

25. Change the units in the drop-down box at the bottom right of the ETABS window to “Kip-in”.

Set Analysis Options

Before running the analysis, let’s review the major options controlling the analysis:

Note:

See the section titled “Analysis Options” in Chapter 15 of the User’s Manual for additional information.

1. Select the **Analyze menu > Set Analysis Options** command to open the **Analysis Options** form.

2. Note under **Building Active Degrees of Freedom** that all six degrees of freedom are selected. This includes the translations (UX, UY, and UZ) along the three global axes and the rotations (RX, RY, and RZ) about the three global axes. Except in special modeling cases (e.g., a planar frame) all six degrees of freedom should always be checked.

3. Note that the **Dynamic Analysis** box has been checked. Click the **Set Dynamic Parameters** button to open the **Dynamic Analysis Parameters** form.

4. Quickly note that three eigenvectors have been requested by default, which will give us the first three natural vibration modes of the structure. Further discussion of this form is beyond the scope of this tutorial. Click the **Cancel** button to close this form.

5. Note that the **Include P-Delta** and the **Save Access DB File** boxes are not checked by default, which is OK for our example.

6. Click **Cancel** to close the **Analysis Options** form.

Perform Analysis

Our first version of the model is complete, and we are now ready to perform the analysis. During the analysis, ETABS will compute the deflections and internal forces and moments for the structure due to the loads.

To run the analysis:

1. Select the **Analyze menu > Run Analysis** command, or click the **Run Analysis** button, , on the top toolbar.

2. When the **Run Options** form appears, click the **Run** button. (You would typically use the **Run Minimized** button for larger structures or more complicated analyses.)

3. Before the analysis is run, ETABS saves the model under the current file name. If we had not already saved the model once, the **Save Model File As** form would open, prompting us to provide a filename and folder.

4. A window will then appear showing the progress of the analysis. When the analysis is complete, an **OK** button will appear at the bottom of this window.

5. Before clicking **OK**, it is a good idea to scroll through the analysis messages to see if any warning or error messages are present. All messages in the analysis window are saved in a file with the extension ".LOG".

6. Click the **OK** button to close the analysis window.

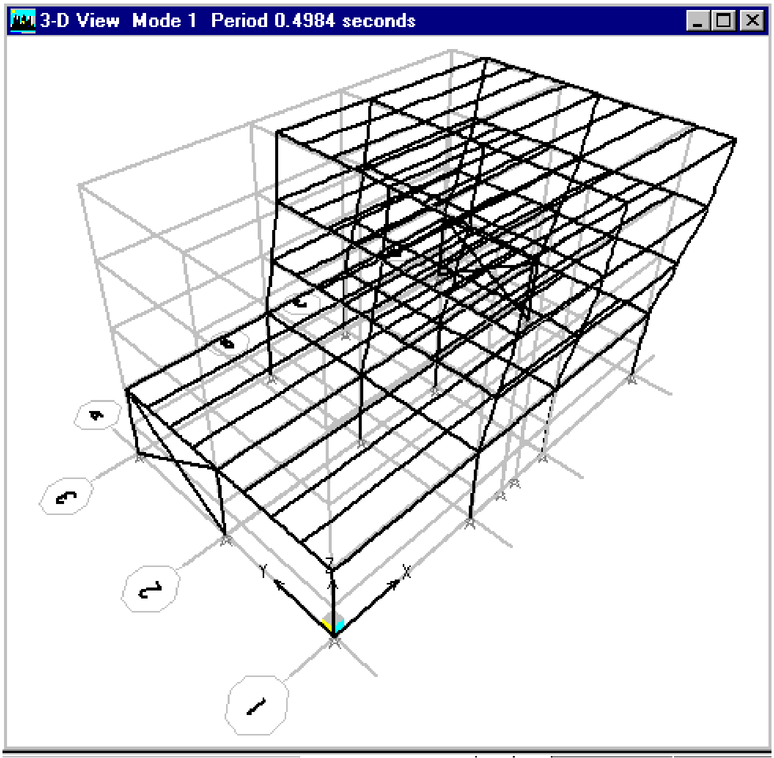
7. Two things happen:

   - The **Lock/Unlock Model** button, , changes to its locked mode, . This prevents us from making any changes to the model that would invalidate the analysis results. To change the model, we would have to click this button, which would unlock the model and delete the analysis results. We won't do that now.

   - The active window automatically displays the deflected shape of the structure for the first vibration mode

Perform Analysis, Step 7

3-D view of first mode shape.



computed during the analysis. This should be primarily a Y-translational mode, with its period shown at the top of the window. See the figure above.

Next we will examine other results of the analysis.

Display Mode Shapes

Even though we are not performing a dynamic seismic analysis, the vibration modes provide useful insight into the behavior of the structure. Since the first mode shape is already being displayed, let's examine the modes further:

1. Click the **Next Case/Step** button, >>, on the status bar at the bottom of the ETABS window. This changes the display to the second mode, which should show primarily X-direction displacement.

Note:

See the section titled "Mode Shape" in Chapter 16 of the User's Manual for additional information.

2. Repeated clicking on the **Next Case/Step** button and **Previous Case/Step** button, , will cycle you through the three modes calculated during the analysis.

3. Click the **Animate** button, , on the status bar to animate the currently displayed mode shape. Animation often makes it easier to visualize the deflected shape. During animation, a slide control appears to the left of the **Animate** button to control the animation speed. Try clicking the left and right arrows on this slide control.

4. Click the **Animate** button again to stop animation.

5. You can also select the mode shape to view by selecting the **Display menu > Show Mode Shape** command or clicking the **Display Mode Shape** button, , on the top toolbar. Do this now to open the **Mode Shape** form, enabling you to choose the mode number, the scaling, and whether to show cubic or straight-line deflected shapes of the line objects. Enter "1" for **Mode Number** and click **OK** to close the form.

6. Click the **Plan View** button, , on the top toolbar and select "ROOF".

7. Observe that mode 1 includes a torsional component due to the unsymmetrical nature of the model. This will be easier to see if you click the **Animate** button, .

8. Click the **Next Case/Step** button, , and observe that mode 2 has no significant torsional component.

9. Stop the animation.

10. Click the **Show Undeformed Shape** button, , on the top toolbar to return the display to the undeformed model.

Display Deformed Shape

We will now examine the deformed shape under the seven static loads. The procedure is essentially the same as viewing the mode shapes:

1. Make sure the right window is active and change it to show a 3-D view.

2. Select the **Display menu > Show Deformed Shape** command (or click the **Display Static Deformed Shape** button, , on the top toolbar) to open the **Deformed Shape** form.

3. Under **Load**, select "DEAD Static Load", and click **OK** to close the form and display the deformed shape. Note that mode shape and deformed shapes do not display extrusions.

4. You can rotate, pan, and zoom to change your view as desired.

Note:

See the section titled "Deformed Shape" in Chapter 16 of the User's Manual for additional information.

5. Right click on any point object (beam/beam or beam/column intersection). This will open the **Joint Displacements** form that shows the three translational and three rotational components of displacement at that point. The translations are in the current length units; the rotations are in radians.

6. You can move the **Joint Displacements** form as follows:

   - Move the mouse to the title bar of the **Joint Displacements** form.

   - Click and hold down the left mouse button and drag the form to the desired location.

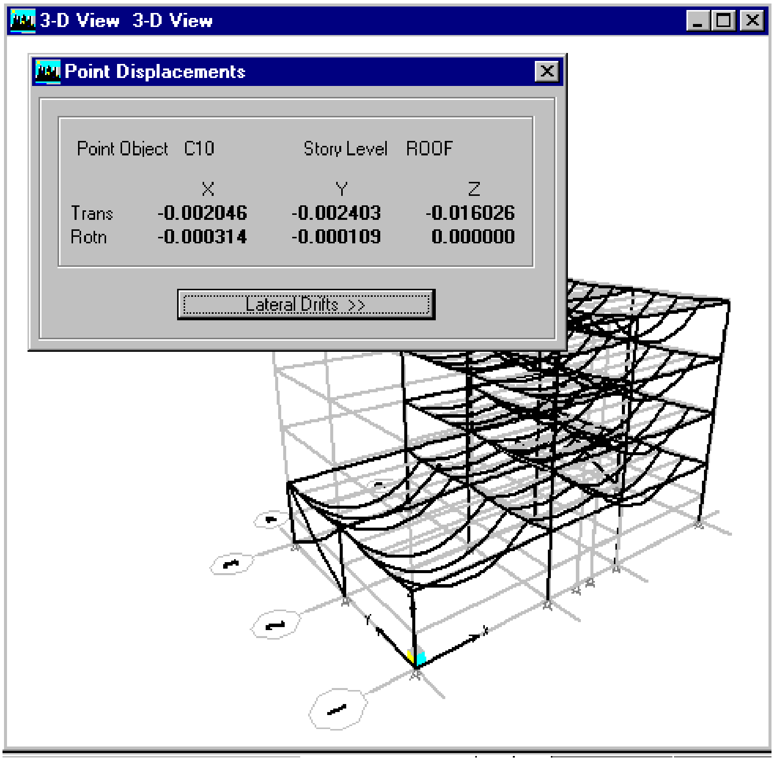
   - Release the mouse button.

7. Right click on another point. The **Joint Displacements** form will update to show the displacements at the new point.

8. Click the **Close Window** button, , in the upper right corner of the **Joint Displacements** form to close it.

Display Deformed Shape, Step 5

3-D view of the deformed shape for load case "DEAD", showing displacements at a point using the right-button click. The units are inches for the translations and radians for the rotations.



9. Clicking on the **Next Case/Step** button, >>, and the **Previous Case/Step** button, <<, will cycle you through the seven static load cases.

10. Clicking the **Animate** button, Start Animation, on the status bar will start and stop animation of the currently displayed deformed shape.

11. Click the title bar of the left window to make it active. Make sure it is displaying the plan view of "ROOF".

12. Click the **Display Static Deformed Shape** button, , on the top toolbar. Under **Load**, select "QUAKEY1 Static Load", and click **OK** to close the form and display the deformed shape.

13. Observe that there is some torsional rotation under this lateral load as we saw for Mode shape 1. This is due in part to

the asymmetry of the structure in the Y direction, and in part due to the eccentric load in this case.

14. Click on the **Next Case/Step** button, >>, and the **Previous Case/Step** button, <<, to see the other three lateral load cases. Observe how the eccentric loading causes some torsion.

15. Right click on a point object in the left view to open the **Joint Displacements** form. Move the form if necessary.

16. Click the **Lateral Drifts** button, Lateral Drifts >>, to open the **Displacements and Drifts** form. This displays a table of horizontal translations and lateral drifts for the given point object at all story levels. The lateral drift at a story is defined as the change in translation from the story below divided by the story height.

17. Click the **Close Window** button, ✕, in the upper right corner of the **Displacements and Drifts** form to close it.

Display Frame Axial Forces

We will now look at the axial forces in the beams and columns throughout the model:

1. For clarity, let's work in a single window. Select the **Options menu > Windows > One** command.

2. Click the **3D View** button, 3-d, on the top toolbar.

Note:

See the section titled "Frame Element, Pier and Spandrel Forces" in Chapter 16 of the User's Manual for additional information.

3. Click the **Set Building View Options** button, on the top toolbar, click the **Defaults** button, and click **OK**.

4. Select the **Display menu > Show Member Forces/Stress Diagram > Frame/Pier/Spandrel Forces** command (or click the **Display Member Force Diagram** button, on the top toolbar) to open the **Member Force Diagram for Frames** form.

5. Under **Load** select "DEAD Static Load"

6. Under **Component**, select **Axial Force**.

7. Under **Scaling**, select **Auto**.

8. Make sure that the **Fill Diagram** box is checked, and the **Show Values on Diagram** box is not checked.

9. Click **OK** to close the form and display the axial forces.

10. Note that the force diagrams for frame elements and for piers and spandrels display simultaneously. You could use the **Set Building View Options** button, , to turn off the wall objects. For now we will view them all simultaneously.

11. Click on the **Next Case/Step** button, >>, and the **Previous Case/Step** button, <<, to cycle through the seven static load cases. Rotate, pan, and zoom as necessary to clearly see the force diagrams in the various members.

12. As you are viewing the various cases, observe the following:

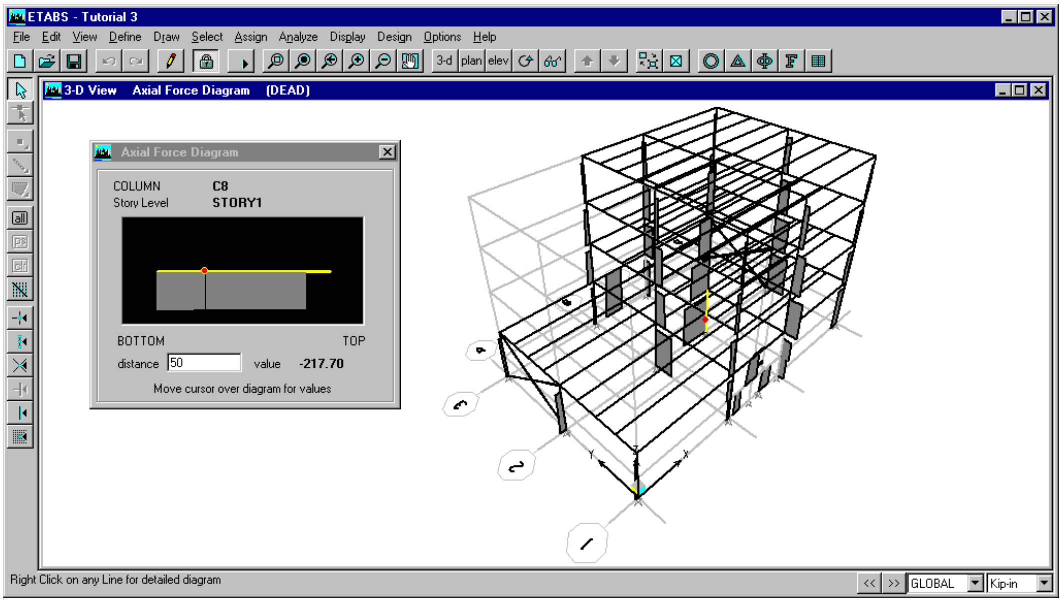
    - Compressive (negative) axial forces display as red and tensile (positive) axial forces display as yellow. (These colors can be changed using the **Options menu > Colors > Output** command.)

    - In 3-D views, compressive (negative) axial forces plot in the negative local 2 direction for each element, and tensile (positive) axial forces plot in the positive local 2 direction. Note that these directions differ from column to column since we rotated four of the columns.

13. Note the large axial forces in the braces for cases “QUAKEY1” and “QUAKEY2”.

14. Right click on any column object. (If necessary, turn off the snap-to-points option by clicking the **Snap to Points** button on the left toolbar.) This will open the **Axial Force Diagram** form that shows a detailed axial-force diagram for that object.

15. Click the title bar of this form, and while holding down the left mouse button, move the mouse to drag the form away from the structure for clearer viewing.



Display Frame Axial Forces, Step 18 *Axial force diagram for load case "DEAD".*

16. At the bottom of this form is an edit box showing the distance from the bottom to be "0", and the corresponding value of axial force at this location. This location is also indicated by the red dot in the diagram in the middle of the form, and by the red dot on the blinking column object on the structure.

17. Move the mouse cursor left and right in the picture box containing the diagram. As the red dot moves, the values of distance and axial force at the bottom of the form will change.

18. Click in the **distance** edit box on this form, and enter a value of "50". The red dot will move and the axial force value will change correspondingly, as shown in the figure above.

19. Observe that there is a region at one or both ends of the column object where no axial force is displayed. These are the end offsets of the member, and correspond to the portion of the object that overlaps the depth of the connected beams. These lengths have been automatically calculated by the program (you also have the option to manually assign these lengths.)

Note:

See the section titled "Frame Rigid Offset Assignments to Line Objects" in Chapter 14 of the User's Manual for additional information.

20. Right click on another line object. The **Axial Force Diagram** form will update to show the diagram for the new object. Note that the axial forces are zero for all beams because they are constrained against deformation by the floor diaphragms.

21. Click the **Close Window** button, , in the upper right corner of the **Axial Force Diagram** form to close the form, or click anywhere in the display window.

22. Click the **Elevation View** button, , on the top toolbar, select **Elevation** "1", and click **OK**.

23. Note that in 2-D views, the axial force is always visible even if the local 2 axis is perpendicular to the viewing plane, as it is for column "C7" on grid line "C".

24. Click the Perspective Toggle button, , repeatedly to change between 2-D and 3-D views and observe how the orientation of the axial force plot differs in these two types of views. Return to the 2-D view.

25. Zoom into the middle bay of the bottom story so we can look at the pier and spandrel forces.

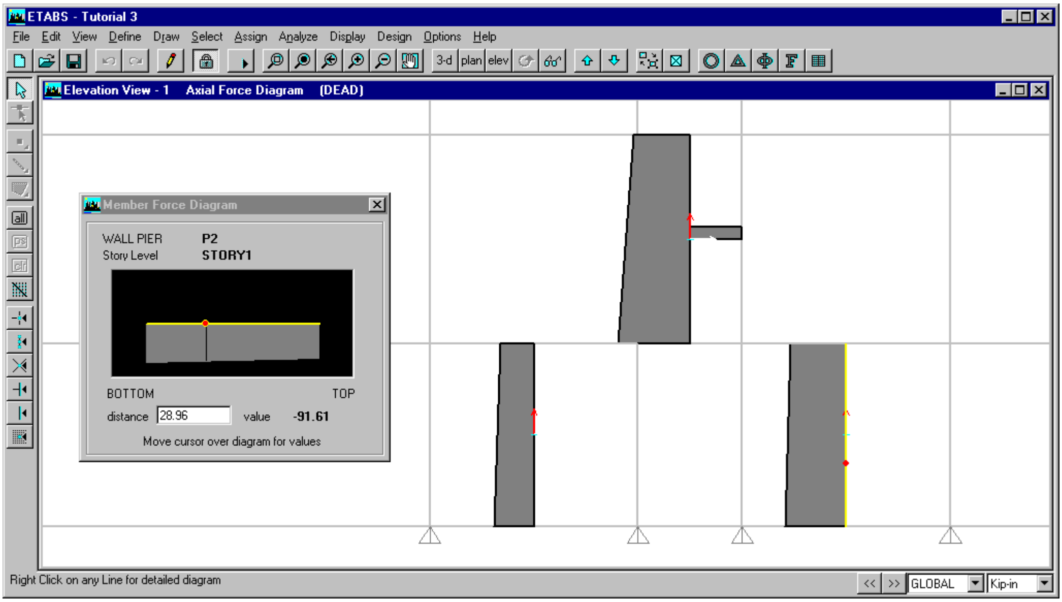
26. Click the **Set Building View Options** button, , on the top toolbar. *Uncheck* the **Column (Line)** and **Beam (Line)** boxes under **Object Present in View**. Check the **Pier Axes** box under **Piers and Spandrels**. Click **OK** to close the form.

27. Observe the following about the piers and spandrels:

    - Each pier and spandrel has a set of local axes similar to frame members; these axes are largely independent of the local axes of the wall.

    - The local 1 axis of a pier connects the centroid of the bottom edge of the pier to the centroid of the top edge.

    - The local 1 axis of a spandrel connects the centroid of one side of the spandrel (e.g., the left edge) to the cen-



Display Frame Axial Forces, Step 28

Axial force diagram for load case "DEAD", showing piers and spandrels.

troid of the other side (e.g., the right edge.) These axes are not currently being displayed.

- The local 2 axis is in the plane of the wall. The local 3 axis is normal to the wall.

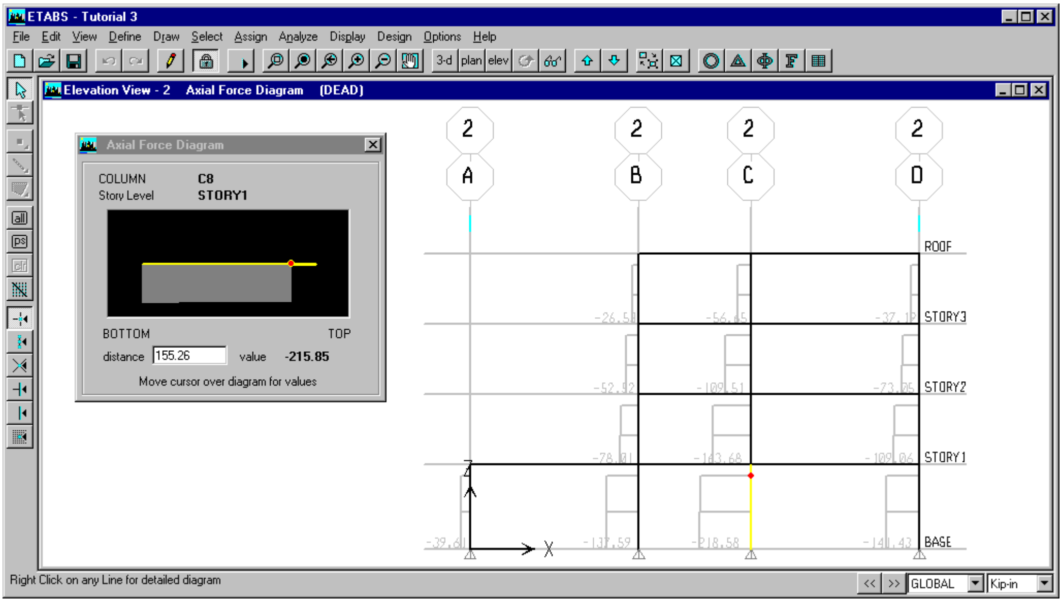
- The abscissa of the force diagram for a pier or spandrel is plotted along the local 1 axis, using exact force values at the two ends and linear interpolation in between.

- The force output display conventions for piers and spandrels are identical to those for frame members.

28. To see a detailed axial force diagram, right-click on the pier or spandrel, as shown in the figure above.

29. Click the **Close Window** button, , in the upper right corner of the **Axial Force Diagram** form to close the form.

30. Finally, let's examine how we can show the force values directly on the structure:



Display Frame Axial Forces, Step 30

Axial force diagram showing values on the directly on the structure for Elevation "2".

- Click the **3D View** button, 3-d, on the top toolbar.
- Click the **Set Building View Options** button, , on the top toolbar. Click the **Defaults** button. Click **OK** to close the form.
- Click the **Display Member Force Diagram** button, , on the top toolbar to open the **Member Force Diagram for Frames** form.
- Uncheck the **Fill Diagram** box.
- Check the **Show Values on Diagram** box.
- Click **OK** to close the form.
- Rotate, pan, and zoom as necessary to see some values.
- Click the **Elevation View** button, elev, on the top toolbar, select **Elevation** "2", and click **OK**.

- You can still right-click on members to see detailed force diagrams, as shown in the figure above.

31. Click the **Close Window** button, , in the upper right corner of the **Axial Force Diagram** form to close the form.

Axial torsion plots in exactly the same way as axial force, so we will not pursue it in this tutorial.

Display Frame Shear Forces

Shear force diagrams are similar to axial force diagrams, with two major differences:

- Shear force has two components; axial force has only one.

- The sign convention for shear force is dependent upon the direction of the local 1 axis of the member; the sign convention for axial force is independent of the local axes.

We will examine these difference in what follows:

1. Click the **3D View** button, 3-d, on the top toolbar.

2. Click the **Set Building View Options** button, , on the top toolbar. Click the **Defaults** button. Click **OK** to close the form.

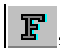
3. Follow the same steps as for displaying the axial force diagram, except select **Shear 2-2** under **Component** on the **Display Member Force Diagram** form. There are two shear components; we are selecting the major shear. For clarity, *uncheck* the **Show Values on Diagram** box, and *check* the **Fill Diagram** box.

4. Step through the different load cases and try different views, just as you did for axial forces.

5. Observe the large shear forces in the piers for load cases "QUAKEX1" and "QUAKEX2". The shear forces in the

piers are also significant for cases "QUAKEY1" and "QUAKEY2" due to torsional effects.

6. Each case is scaled automatically, so it is difficult to compare values graphically. Let's change to an absolute scale:

   - Display case "QUAKEX1".
   - Click the **Display Member Force Diagram** button, , on the top toolbar.
   - Under **Scaling**, change from **Auto** to **Scale Factor**.
   - The value shown in the **Scale Factor** edit box is the value the program chose automatically for this case. Do not change it.
   - Click **OK**.

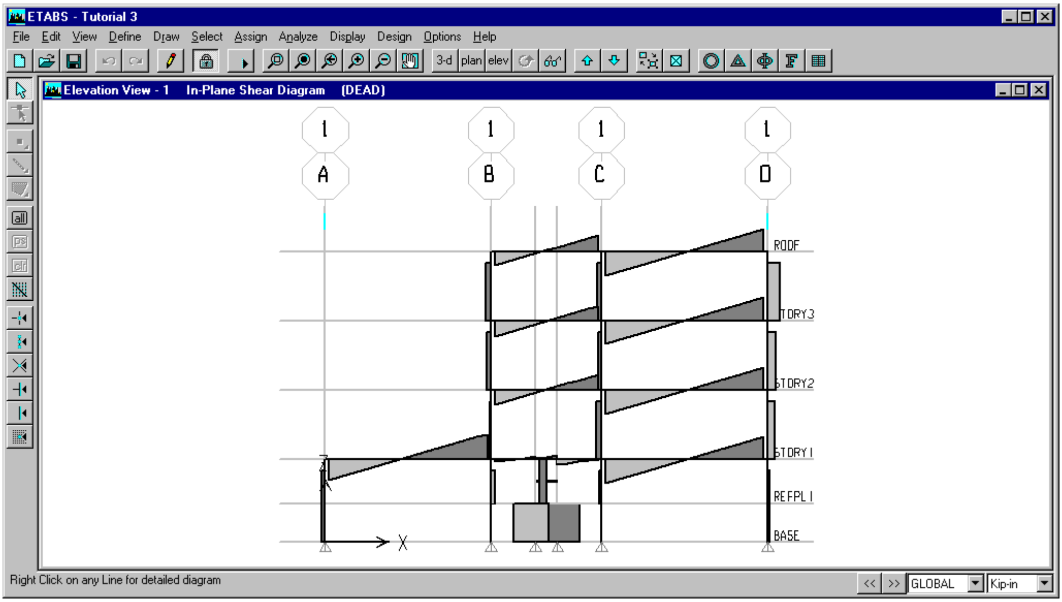
7. Now step through the four lateral load cases (without changing the scale factor), and observe the relative values of shear force in the piers.

8. Like axial forces, negative shear forces display as red and positive shear forces display as yellow. Unlike axial forces, the sign convention for major shear forces depends on the direction of the local axes 1 and 2. See Chapter 35 titled "Frame Element Output Conventions" in the *ETABS User's Manual* for the shear-force sign conventions.

9. In 3-D views, negative major shear forces plot in the negative local 2 direction for each element, and positive major shear forces plot in the positive local 2 direction. Minor shear plots similarly along the local 3 axis.

10. Right click on any beam, column, pier, or spandrel to open the **Shear Force 2-2 Diagram** form that shows a detailed shear-force diagram for that object. Use of this form is the same as for the **Axial Force Diagram** form.

11. Click the **Elevation View** button, , on the top toolbar, select **Elevation** "1", and click **OK**.



Display Frame Shear Forces, Step 17

In-plane shear force diagram in elevation "1" for load case "DEAD".

12. Note that in 2-D views, the shear-force diagram is always visible even if the local 2 axis is perpendicular to the viewing plane, as it is for column "C7". This shear may not all be acting in the viewing plane!

13. Click the Perspective Toggle button, , repeatedly to change between 2-D and 3-D views and observe how the orientation of the shear-force diagram differs in these two types of views.

14. Click the Perspective Toggle button, , if necessary to return to a 2-D view.

15. Click the **Display Member Force Diagram** button, , on the top toolbar to open the **Member Force Diagram for Frames** form.

16. Note that when a 2-D view is active, the option to select **In-plane Shear** is available under **Component**. Select this component, change back to automatic scaling, and click **OK** to close the form.

17. This displays the resultant shear-force vector (major plus minor shear), projected onto the viewing plane. All displayed shear force is acting in the viewing plane. See the figure above.

18. Click the Perspective Toggle button, , again. Because the in-plane shear is invalid in 3-D, the view reverts to showing the undeformed shape.

Display Frame Bending Moments

Bending-moment diagrams are similar to axial-force and shear-force diagrams, with differences that we will examine here:

1. Click the **3D View** button, 

, on the top toolbar.

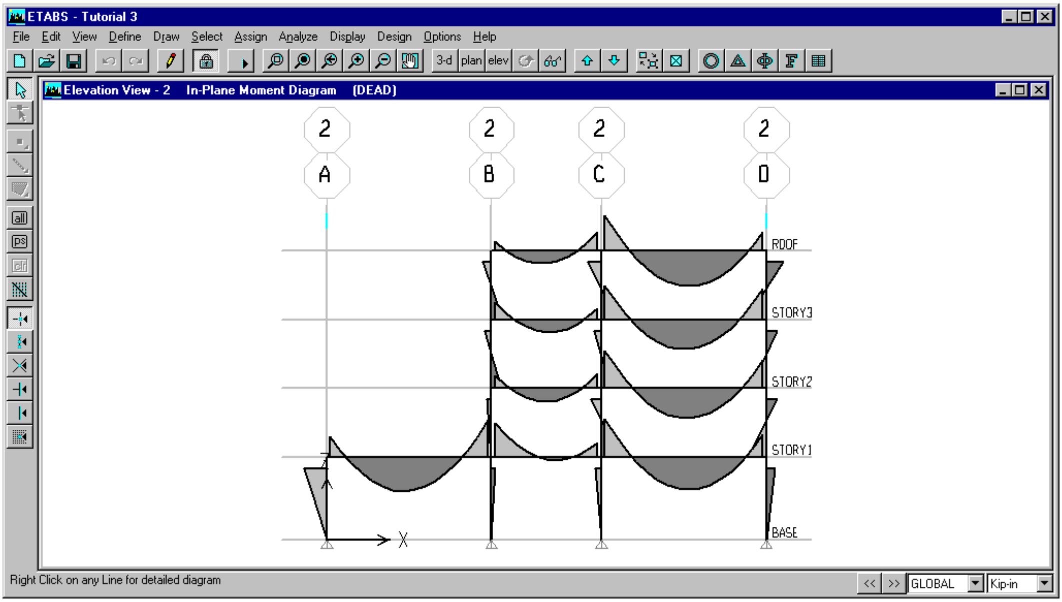
2. Follow the same steps as for displaying the shear-force diagram, except select **Moment 3-3** under **Component** on the **Display Member Force Diagram** form. There are two bending-moment components; we are selecting the major moment.

3. Step through the different load cases and try different views, just as you did for axial and shear forces.

4. Like the forces, negative bending moments display as red and positive bending moments display as yellow. Unlike axial force and shear forces, the sign convention for major bending moments depends only the direction of the local axis 2. See Chapter 35 titled “Frame Element Output Conventions” in the *ETABS User’s Manual* for the bending-moment sign convention.

5. In 3-D views, major bending moments plot along the local 2 axis, and minor moments plot along the local 3 axis.

6. By default, positive bending moments plot on the tension side of the beam, column, pier, or spandrel; negative moments plot on the compression side. You may switch this behavior by using the **Options menu > Moment Diagrams on Tension Side** command.



Display Frame Shear Forces, Step 14

In-plane moment diagram in elevation "2" for load case "DEAD".

7. Right click on any beam, column, pier, or spandrel to open the **Moment 3-3 Diagram** form that shows a detailed bending-moment diagram for that object. Use of this form is the same as for the **Axial Force Diagram** form.

8. Click the **Elevation View** button, elev, on the top toolbar, select **Elevation** "2", and click **OK**.

9. Note that in 2-D views, the bending moment is always visible even if the local 2 axis is perpendicular to the viewing plane, as it is for column "C7". This moment may not all be acting in the viewing plane!

10. Click the Perspective Toggle button, , repeatedly to change between 2-D and 3-D views and observe how the orientation of the bending-moment diagram differs in these two types of views.

11. Click the Perspective Toggle button, , if necessary to return to a 2-D view.

12. Click the **Display Member Force Diagram** button, , on the top toolbar to open the **Member Force Diagram for Frames** form.

13. Note that when a 2-D view is active, the option to select **In-plane Moment** is available under **Component**. Select it and click **OK** to close the form.

14. This displays the resultant bending-moment vector (major plus minor moment), projected onto the viewing plane. All bending moment is acting about the axis normal to the viewing plane. See the figure above.

15. Click the Perspective Toggle button, , again. Because the in-plane moment is invalid in 3-D, the view reverts to showing the undeformed shape.

Chapter 5

Design and Optimize the Model

This chapter continues the tutorial from Chapter 4. Now that the analysis is done, we are ready to check our model against code requirements. Based on the design results, we will modify the model and iterate on the analysis-and-design procedures to get our final structure.

Check Steel Frame Design

With the analysis complete, we can now check the structure to see if it satisfies design code requirements. If we had assigned specific sections to the beams and columns, this would be a one-time check. Instead, we assigned an auto select section list to each beam and column, so we need to perform an iterative process to get the "optimum" design.

ETABS differentiates between steel-frame design and composite-beam design. Steel-frame design considers *all* forces acting on the frame member (column, beam, or brace.) Composite-beam design applies only to infill beams, and considers only the major (M3) bending moments and the major (V2) shear forces.

We will now design the columns, braces, and moment-frame beams using the ETABS steel-frame design feature. Later, we will use the ETABS composite-beam design feature for the infill beams.

1. First we must select the design code we are using. Select the **Options menu > Preferences > Steel Frame Design** command to open the **Steel Frame Design Preferences** form.

2. Working in the right-hand column of this form, click on the individual data fields and do the following:

   - Select "UBC97-ASD" for **Design Code**.

   - Select "Ordinary MRF" for **Frame Type**.

   - Select "Zone 4" for **Zone**.

   - Enter "2.8" for Omega0.

   - The other fields don't apply in our case.

3. Click **OK** to close the form.

4. Select the **Design menu > Steel Frame Design > Select Design Combo** command to open the Steel frame **Design Load Combinations Selection** form. This form can be used to select which load combinations are used for design. We are just going to look.

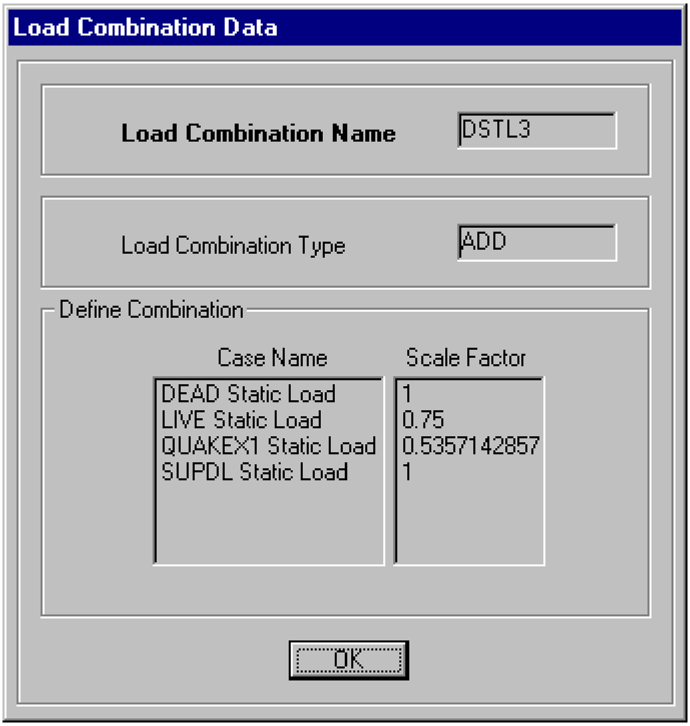
5. ETABS has automatically created twenty-six load combinations, based on the UBC code, which will be checked for design. These "combos" are listed in the scroll box on the right, labeled **Design Combos**.

6. Click on combo "DSTL1" and click the **Show** button. This opens the **Load Combination Data** form. You can see that this combo is just the sum of the two dead-load cases, "DEAD" and "SUPDL", both scaled by unity. Click **OK** to close the form.

7. Now click on combo "DSTL3" and click the **Show** button. This combo is the sum of four static load cases, as shown in the figure below:

***Check Steel Frame Design**, Step 7*

Load Combination data for combo "DSTL3".



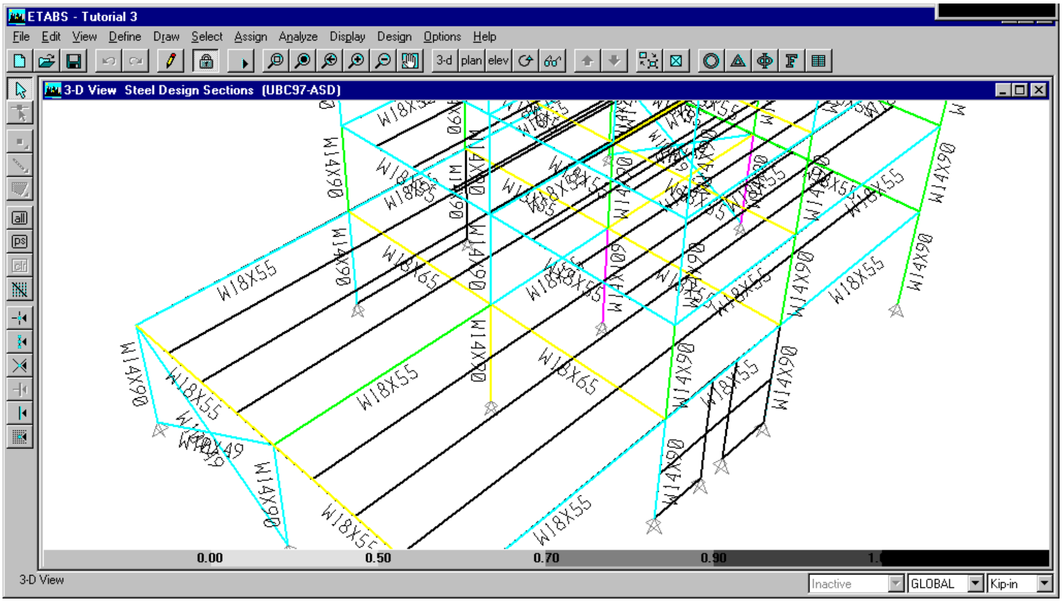
- "DEAD" multiplied by "1".
- "SUPDL" multiplied by "1".
- "LIVE" multiplied by "0.75".
- "QUAKEX1" multiplied by "0.5357" (this is 0.75 divided by 1.4).

8. Click **OK** to close this form.

9. Click on combo "DSTL4" and click the **Show** button. Observe that this is the same as "DSTL3" except for the sign of the scale factor for case "QUAKEX". Click **OK**.

10. Look at other combos if you wish to, then click **Cancel** to close the **Design Load Combinations Selection** form.

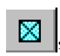
11. Make sure that the active display window is showing a 3-D view of the whole structure.



Check Steel Frame Design, Step 12 *Steel Frame Sections after first analysis.*

12. Select the **Design menu > Steel Frame Design > Start Design/Check of Structure** command. You will then see each of the beams, columns, and braces highlighted, in sequence, as the program performs the design check and optimization. For each member, all available sections are checked, and the section with the minimum weight that satisfies the design code is selected.

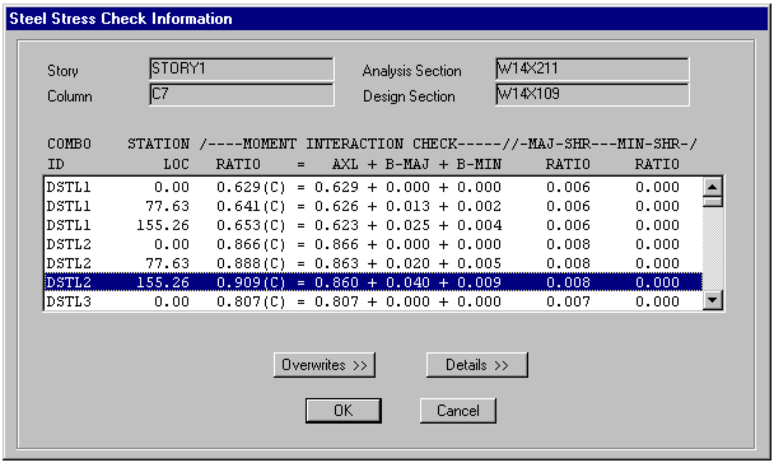
13. When done, the display changes to show the selected section for each member, and the color of each member indicates the P-M interaction ratio according to the legend shown at the bottom of the window.

14. Click the **Set Building View Options** button, , on the top toolbar. Under **Visible in View**, uncheck the **Grid Lines** box. Click **OK**. This will make view clearer.

15. Pan, zoom, and rotate as necessary to see that most of the columns are labeled "W14X90", most of the beams are labeled "W18X55", and all the braces are all labeled "W10X49". See the figure above.

Check Steel Frame Design, Step 16

Steel stress check information as obtained by right-clicking on a column member, highlighting the load combination and location in the member that controls the design.



16. Zoom into the structure and right-click on a column at the bottom (part of "STORY1".) This opens the **Steel Stress Check Information** form.

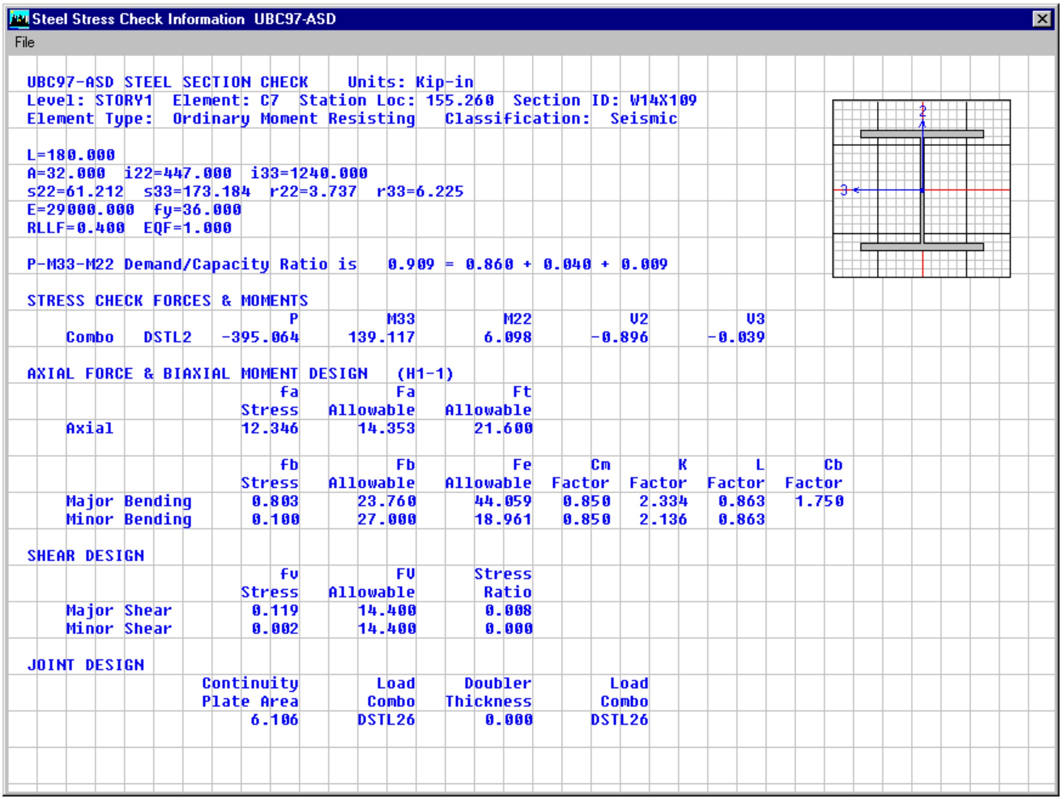
17. The top of the form should show the story and column label. The **Analysis Section** is "W14X211", and the **Design Section** is "W14X109". The important thing to understand here is that the analysis results were based on the weight and stiffness of the assumed "W14X211" section, and these analysis results were used to select the "W14X109" section. We will need to iterate later to validate this selection.

18. The scroll box in the middle of the form shows the results obtained at each location in the member for each of the design combos. These results include the P-M interaction ratios and the shear ratios. The highlighted line is the location and combo that controls the design.

19. Click the **Details** button to open the **Steel Stress Check Information UBC97-ASD** form, which shows all the details of the design-check calculations for the highlighted combo and location. See the figure below.

20. You can print this form from the **File** menu *on the form*. Click the **Close Window** button, , in the upper right corner of the form to close it.

21. Click the **Overwrite** button to open the **Steel Frame Design Overwrites** form. This shows the various parameters that are



Check Steel Frame Design, Step 19 *Detailed information for a single location in a single member under a single design combo.*

used for the stress check, and enables you to modify them on a member-by-member basis. Refer to the *ETABS Steel Design Manual* for more information on the meaning of these values. Click **Cancel** to close the form.

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

23. You may check other members if you wish by right-clicking on them. Note that we have not yet designed the secondary beams, so no information is available for them.

24. Now select the **Design menu > Steel Frame Design > Display Design Info** command to open the **Display Design Results** form.

25. Click **Design Output**, then select "P-M Ratio Colors & Values". Click **OK** to close the form. The display now shows the P-M ratios instead of the design sections for each member.

26. Click the **Elevation View** button, 

, on the top toolbar, select **Elevation** "1", and click **OK**.

27. Click the **Move Up in List** button, , repeatedly to view all seven different elevations. Check the P-M ratios and note that none of them exceed 1.0. You may right-click on any member for which you want to check the design details.

28. Select the **Design menu > Steel Frame Design > Verify all Members Passed** command. The resulting message indicates that all members passed the steel frame design check. Click **OK** to close the message.

29. Select the **Design menu > Steel Frame Design > Verify Analysis vs Design Section** command. This opens a message box that tells us that analysis and design sections differ for 96 members, the entire structure. Clearly we need to iterate on the analysis and design procedure.

30. Click **No** to close the form without selecting the members.

Check Composite Beam Design

Composite beams are checked separately from the steel frame, but the concepts are similar:

1. First we must select the design code we are using. Select the **Options menu > Preferences > Composite Beam Design** command to open the **Composite Beam Design Preferences** form.

2. There are five tabs on this form, each with several parameters. Click each tab in turn, and view some of the options.

3. We will use all default values. Click the **Reset All** button. This selects **Design Code** "AISC-ASD89" with default values for loading and deflection, ignores vibration, and opti-

mizes the design for weight rather than price. Click **OK** to close the form.

4. Select the **Design menu > Composite Beam Design > Select Design Combo** command to open the Composite Beam **Design Load Combinations Selection** form. This form can be used to select which load combinations are used for design. We are just going to look.

5. ETABS has automatically created six load combinations, based on the AISC code, which will be checked for design.

6. Click the **Construction** tab. You will see that two combos are listed on the right that will be used for design for construction considerations. All the combos listed on the left are defined, but will not be used.

7. Click the **Strength** tab and note that two different combos are used for strength considerations, and similarly for the **Deflection** tab.

8. To see the definition of any combo, click the combo name, then click the **Show** button. Click the **OK** button to close the form.

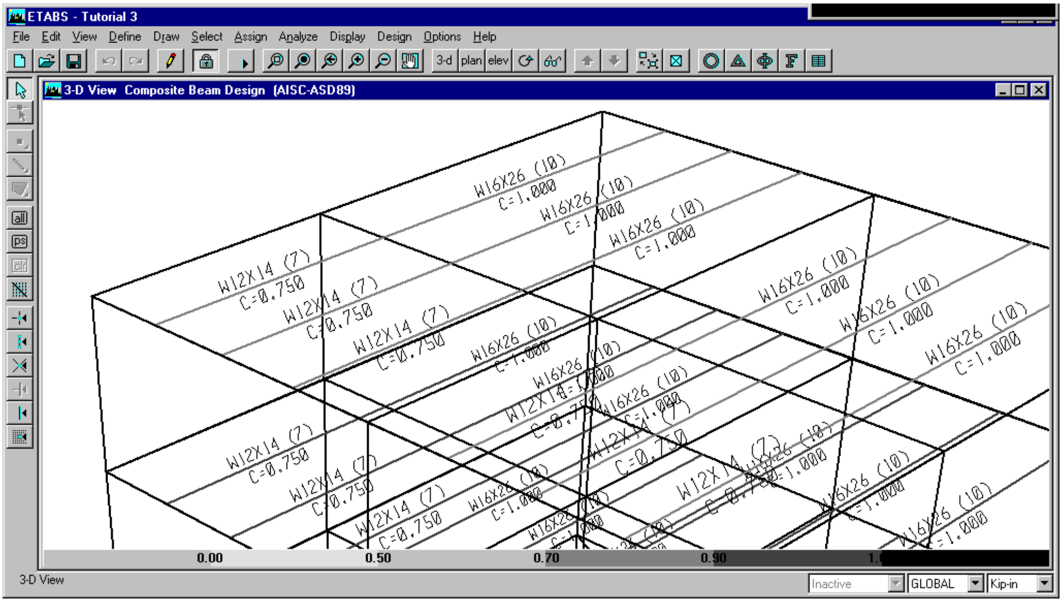
9. Click **Cancel** to close the **Design Load Combinations Selection** form.

10. Make sure that the active display window is showing a 3-D view of the whole structure.

11. Select the **Design menu > Composite Beam Design > Start Design/Check of Structure** command. You will then see each of the secondary beams highlighted, in sequence, as the program performs the design check and optimization. For each member, all available sections are checked, and the section with the minimum weight that satisfies the design code is selected.

12. When done, the display changes to show the following for each member:

    - The selected section.



Check Composite Beam Design, Step 12 *Composite beam section, number of shear studs, and camber (inches) after first analysis.*

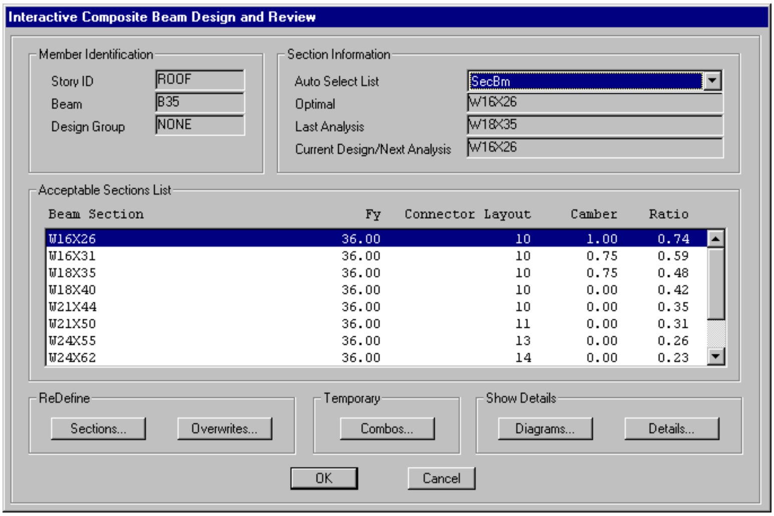
- The number of shear studs required.
- The camber required.
- The color of each member indicates the maximum moment, shear, deflection, or vibration ratio encountered during the design check, according to the legend shown at the bottom of the window.
- See the figure above.

13. Zoom into any portion of the structure, and right-click one of the secondary beams. This opens the **Interactive Composite Beam Design and Review** form, as shown in the figure below.

14. This is a very comprehensive form with many options, and it is deserving of more attention than can be given here. You may try the various features yourself. There are three important things to notice:

Check Composite Beam Design, Step 14

Interactive Composite Beam Design *form as obtained by right-clicking on a secondary beam. The optimal section is highlighted.*



- All available sections that have passed the design requirements are shown in the scroll box labeled **Acceptable Sections List** in the middle of the form. The minimum weight (or least cost) section is highlighted.

- This form is interactive, meaning that with each change you make to any of the settings, the design is immediately re-checked and the results are displayed. This applies only to the selected member.

- For more information, you can press the **F1** key for help or refer to the Chapter 4 titled "Interactive Composite Beam Design and Review" in the *ETABS Composite Beam Design Manual*.

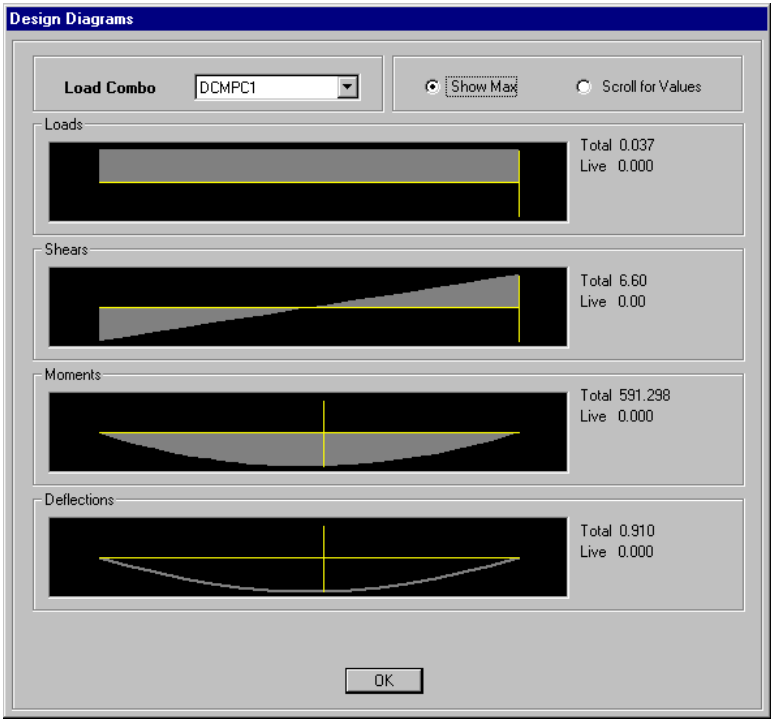
15. Click the **Diagrams** button under **Show Details**. Try using the scroll bar at the bottom, and try the **Show Max** option, as shown in the figure below. Click **OK** to close.

16. Click the **Details** buttons under **Show Details** to see all the design details. Click the **Close Window** button, ☒, in the upper right corner of the form to close it.

17. When you are all done, click **Cancel** to close the **Interactive Composite Beam Design and Review** form without saving any changes you may have made.

Check Composite Beam Design, Step 15

Design diagrams for a secondary beam, showing the load and response diagrams, with the maximum values indicated.



18. You may check other members if you wish by right-clicking on them.

19. Select the **Design menu > Composite Beam Design > Display Design Info** command to open the **Display Composite Beam Design Info** form.

Note:

The stress ratios are discussed in the section titled "Stress Ratios" in Chapter 25 of the Composite Beam Design Manual.

20. Click **Stress Ratios**, then check both the **Construction Load** and **Final Load** boxes. Click **OK** to close the form. The display now shows the maximum moment and shear ratios for construction and final loading for each member.

21. Click the **Plan View** button, 

, on the top toolbar, select "ROOF", and click **OK**.

22. Click the **Move Down in List** button, , repeatedly to view all four different elevations. Check the ratios and note that none of them exceed 1.0. You may right-click on any member for which you want to check the design details.

23. Select the **Design menu > Composite Beam Design > Verify all Members Passed** command. The resulting message indicates that all members passed the steel frame design check. Click **OK** to close the message.

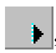
24. Select the **Design menu > Composite Beam Design > Verify Analysis vs Design Section** command. This opens a message box that tells us that analysis and design sections differ for 54 members, all composite beams in the structure. Clearly we need to iterate on the analysis and design procedure.

25. Click **No** to close the form without selecting the members.

Iterate for Final Design

For the first analysis, ETABS used the median-weight section for each auto-selection section property set. For subsequent analyses, the program will use the current design sections for the analysis section properties. In this example these are the minimum-weight sections chosen by the steel frame and composite beam design checking procedures. The analysis results may change since the new sections may have different weight and stiffness than the sections used previously, and this in turn may affect the design results. We proceed as follows:

1. Perform the analysis:

    - Click the **Run Analysis** button, , on the top toolbar. Click **Run** on the **Run Options** form.

    - When the analysis is complete, scroll through the analysis messages to see if any warning or error messages are present, then click the **OK** button to close the analysis window.

1. Perform the steel frame design check:

    - Select the **Design menu > Steel Frame Design > Start Design/Check of Structure** command.

- Select the **Design menu > Steel Frame Design > Verify all Sections Passed** command. If they all pass (they should for this example), click **OK**.

- If all members passed, select the **Design menu > Steel Frame Design > Verify Analysis vs Design Sections** command. If no sections differ, click **OK**. If any sections differ, click **No**.

2. Perform the composite beam design check:

   - Select the **Design menu > Composite Beam Design > Start Design/Check of Structure** command.

   - When this is done, select the **Design menu > Composite Beam Design > Verify all Sections Passed** command. If they all pass (they should for this example), click **OK**.

   - If all members passed, select the **Design menu > Composite Beam Design > Verify Analysis vs Design Sections** command. If no sections differ, click **OK**. If any sections differ, click **No**.

3. If any analysis sections differ from design sections for either steel-frame or composite-beam design, go back to Step 1 and perform another iteration. If the analysis and design sections are all the same, the iteration is complete.

For this example, the steel-frame design takes three iterations, and the composite-beam design takes two. During the iteration process, you can review design results for individual members as we did before. You could also change the model, loads, or design parameters as you wish, but in the end you should make sure that all members pass the design checks, and that the analysis and design sections are the same. You must also exercise your own engineering judgement as to the soundness of the design and the validity of the results produced by the program.

Our final design for the steel frame has resulted in almost all the columns being the same, and all the moment-frame beams being the same. This is a relatively small, stiff structure that is using the smallest available sections in the auto select section lists. There may be some additional savings to be found by including smaller sections in the auto select sections list for this model.

Note that the periods of the structure lengthened substantially during the iteration, since we moved toward smaller sections. In a different model, it could go the other way.

Check Shear Wall Design

For concrete members (beams, columns, walls) ETABS does not use auto select section lists or perform any kind of sizing. ETABS does determine the amount of reinforcing steel needed, and checks if the design is adequate.

1. First we must select the design code we are using. Select the **Options menu > Preferences > Shear Wall Design** command to open the **Shear Wall Design Preferences** form.

2. We will use all default values including the 1997 UBC code. Each of these preference items is described in the *ETABS Shear Wall Design Manual*. Click **OK** to close the form.

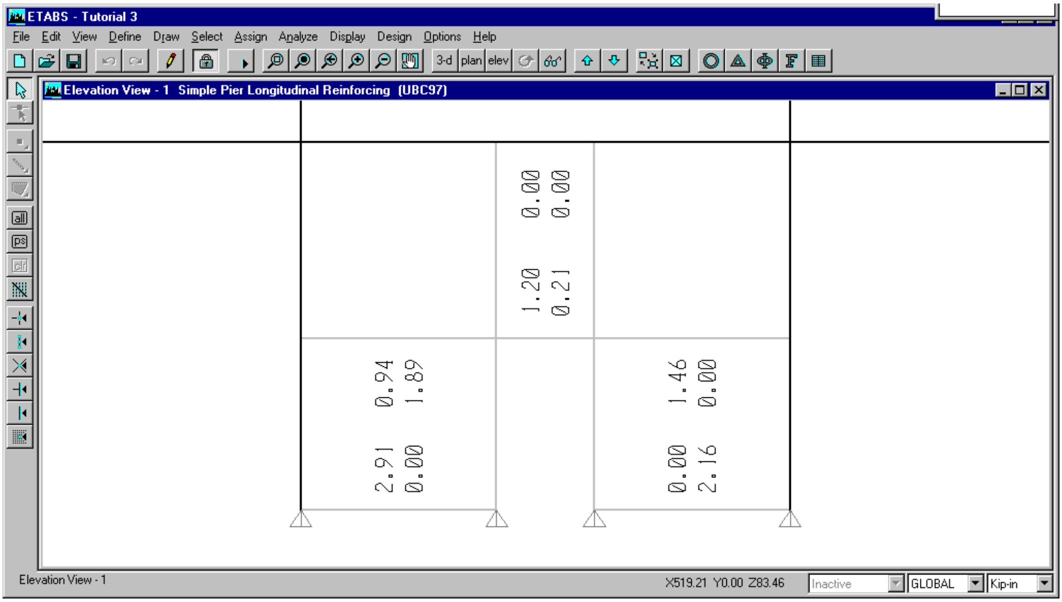
3. Select the **Design menu > Shear Wall Design > Select Design Combo** command to open the Shear Wall **Design Load Combinations Selection** form. This form is similar to that used for steel design. You may want to view a few of the load combinations now. Click **Cancel** to close the form.

4. Select the **Design menu > Shear Wall Design > Start Design/Check of Structure** command. The piers and spandrels are now designed.

5. When the design is complete, change the view so you can clearly see the piers and spandrel in the wall with the door opening. The numbers displayed are the larger of either the tension or compression reinforcing steel area required at the four corners of each pier. See the figure below. The amount of steel required in our model is small because of the presence of the columns on either side of the wall.

6. Select the **Design menu > Shear Wall Design > Display Design Info** command to open the **Display Design Results** form.

7. Click **Design Output**, and then select “Spandrel Longitudinal Reinforcing”.



Check Shear Wall Design, Step 5 *Maximum area of longitudinal steel (tension or compression) required in the four corners of each wall pier.*

8. Click **OK** to close the form and display the longitudinal steel required at both ends of the top and bottom of the spandrels.

9. In a similar fashion you can select other design information for display.

10. To get more details, right click on a pier or spandrel to open the **Pier Design** or **Spandrel Design** form, respectively. If you right-click on the wall object directly above the door, you will be prompted to select whether you want to view the pier or spandrel results. The design details shown on these two forms are documented in the *ETABS Shear Wall Design Manual*.

11. Click OK to close the form. Check other members if you wish.

For piers, ETABS can do a simplified overturning design that provides lumped reinforcing at each end of the wall, or it can perform a more complete design based on exact interaction curves for actual reinforcement layouts in the wall. In this example we have done only the simplified overturning design.

In an actual project you should follow-up to this simplified design by defining your actual wall section (including reinforcing) using the Section Designer feature of ETABS, assign that section to your wall pier, and then run the design/check to verify that the stress ratios are acceptable. More information on this process is provided in the *ETABS Shear Wall Design Manual*.

The addition of reinforcing will do little to change our analysis results, hence we do not need to iterate further. However, it was necessary to complete the sizing of the steel members before performing this final check on the shear walls.

Print Results

Output from the program can be obtained in many forms, including printouts of graphics and text, AutoCAD drawing exchange (.DXF) files, Windows enhanced metafiles for inclusion into documents, Microsoft Access database files, and output tables displayed on the screen. In this tutorial, we will only obtain a few hardcopy printouts:

1. First we will print a graphical display. Click the title bar of any display window to make it active.

2. Set the display to show the type of information you want to print. You can print the undeformed shape, deformed shape, force/moment diagrams, design information, or any other information that can be shown in a display window. For example, click the **Display Static Deformed Shape** button, , on the top toolbar, and select a load case.

3. Set the view you want to see (3D, plan, or elevation)

4. Use zoom, rotate, and other options to get the view that you want to see.

5. Select the **File menu > Print Graphics** command. This will send the current graphical display in the active window to your current default Windows printer.

6. You can get more control over the printout by using the **File menu > Print Setup** command and **File menu > Print Preview for Graphics** command. Control over the colors and line types is available by selecting the **Options menu > Colors > Output** command.

7. We will now print some tabular information. Select two or three members by clicking them in the active window. Check your selection in the status bar on the bottom left of the ETABS window.

8. Select the **File menu > Print Tables…** command. Note that you have a choice of printing input data, analysis results, or design results. Select the **Analysis Output** command. This opens the **Print Output Tables** form.

9. Under **Type of Analysis Results**, the **Displacements**, **Reactions**, **Beam Forces**, and **Column Forces** boxes should already be checked. You will only get these types of output if the appropriate types of objects have been selected.

10. Click the Select Loads button. This will open the **Select Output** form. You will see a scroll box containing the seven static load cases we defined plus all the load combinations the program used for design checking.

11. Click on “DCMPC2 Combo”.

12. While holding down the **Ctrl** key, click on “DSTL10 Combo”. Both “DCMPC2” and “DSTL10” should now be highlighted.

13. Scroll down to the bottom. While holding down the **Ctrl** key, click on “QUAKEY1 Static Load”. Scroll to check that three cases are now highlighted.

14. Click the **OK** button to close the form.

15. Make sure that the **Selection Only** box is checked (otherwise the entire model will print.)

16. Click **OK** to close the form and send the output to your default Windows printer.

17. You may choose other graphical or tabular output for printing.

Conclusion

We have covered many of the basic features of the ETABS program in this tutorial, and yet there are many more we have not had time to explore. You have been exposed to enough of the program to become quickly productive. The additional features will make certain tasks easier, allow you to model more complicated buildings, and provide many additional capabilities for analysis and design.

There is much information available in the help facility within the program, in the printed *ETABS User's Manual*, and in the electronic manuals that are installed on your computer with the program. Be sure to consult these resources as you try to expand your knowledge of the program. Don't hesitate to experiment and try different approaches to modeling — you can always use the **Undo** button.