Tutorial

Part I - Steel Building Example

This manual provides step-by-step instructions for building a basic ETABS model. Each step of the model creation process is identified, and various model construction techniques are introduced. If you follow the instructions, you will build the model shown in Figure 1.

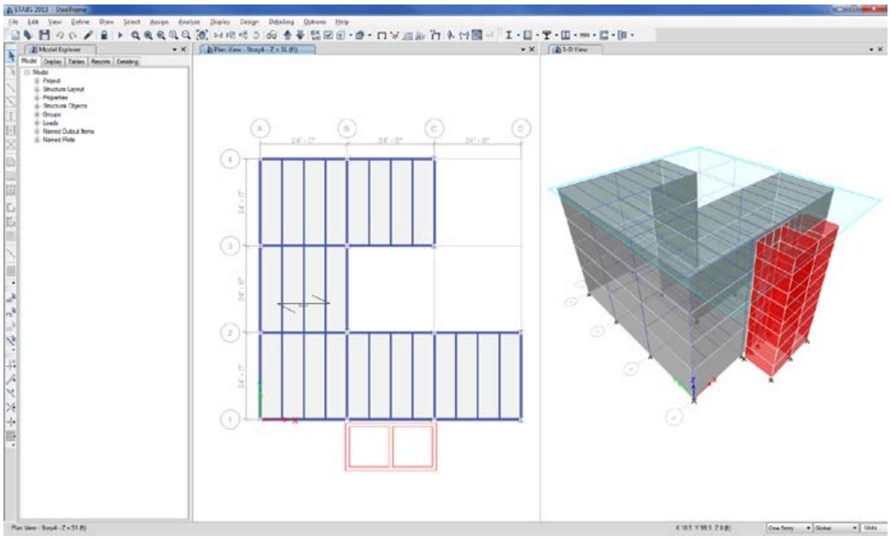


Figure 1
An Example of a Model

The Project

The example project is an irregularly shaped four-story building with an external elevator core. The first story is 15 feet high and stories 2, 3, and 4 are each 12 feet high. The bays are 24 feet in the X and Y directions.

The lateral force resisting system consists of intersecting moment frames (the elevator core is structurally isolated). The floors consist of 3 inches of concrete over a 3-inch-deep metal deck. The secondary (infill) beams are designed as composite beams. The lateral-force resisting beams that connect the columns are designed as noncomposite beams.

The architect for the building has requested that the maximum beam depth not exceed that of a W18 beam to allow sufficient clearance for ductwork running beneath the beams.

Step 1 Begin a New Model

In this Step, the story height and girds are set. Then a list of sections that fit the parameters set by the architect for the design are defined.

A. Start the program. The Start Page will display.

B. Click the **New Model** button on the Start Page and the Model Initialization form shown in Figure 2 will display.

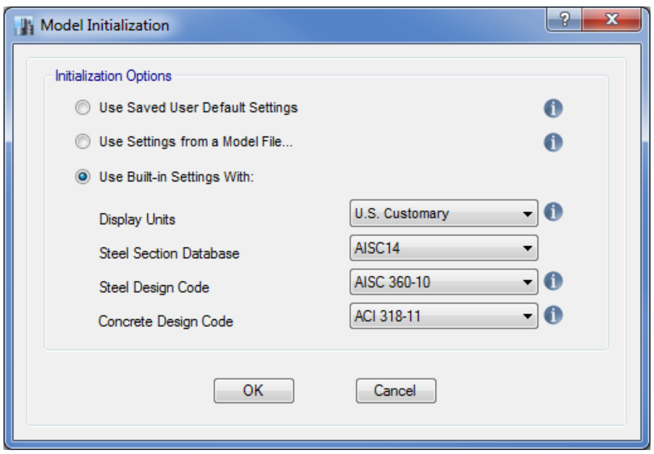


Figure 2 Model Initialization form

C. Choose the *Use Built-in Settings With:* option.

D. Select *U.S. Customary* base units from the Display Units drop-down list on the Model Initialization form. To review the display units hold the mouse cursor over the information icon . To change the units once initialized, click the **Options menu > Display Units** command.

E. Select *AISC14* from the Steel Section Database drop-down list.

F. Select *AISC360-10* from the Steel Design Code drop-down list on the Model Initialization form. Click the **OK** button and the New Model Quick Templates form shown in Figure 3 will display.

The New Model Quick Templates form is used to specify horizontal grid line spacing, story data, and template models. Template models provide a quick, easy way of starting a model. They automatically add structural objects with appropriate properties to the model. We highly recommend that you start your models using templates whenever possible. However, in this example, the model is built from scratch, rather than using a template.

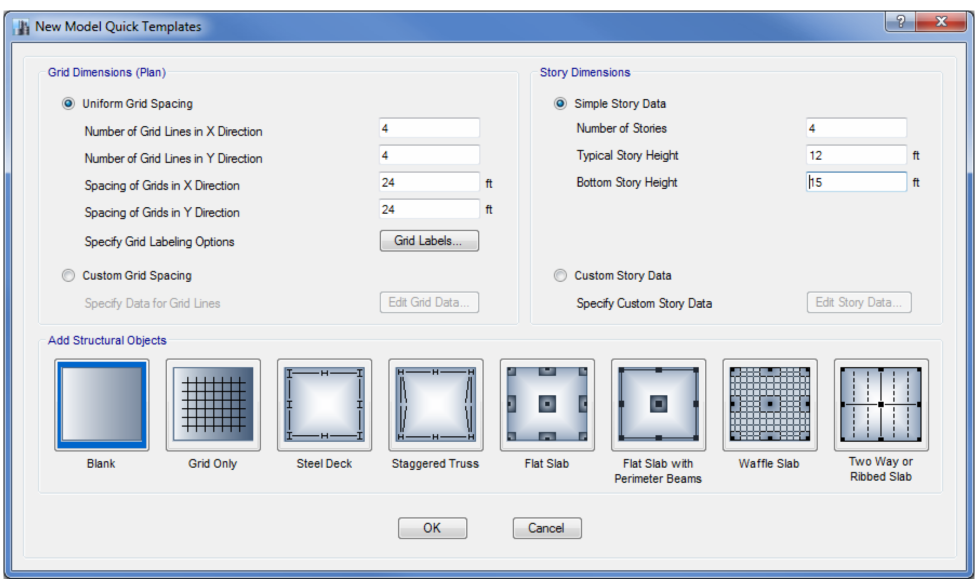


Figure 3
New Model Quick Templates form

G. Set the number of stories in the Number of Stories edit box to **4**.

H. Type **180 in** into the Bottom Story Height edit box and press the Enter key on your keyboard (be sure you type ***in***). Notice that the program automatically converts the 180 in to 15 because the input unit for this edit box is feet (180 inches = 15 feet).

I. Click the **Blank** button in the Add Structural Objects area - the button should be highlighted by a dark blue border.

J. Click the **OK** button to display the blank windows and origin.

In addition to the origin, the program also shows the horizon. We will shut off the horizon in the next steps so that the model grids will be more visible.

K. Click the **Set Display Options** button or use the **View menu** > **Set Display Options** command. The Set View Options form shown in Figure 4 will display.

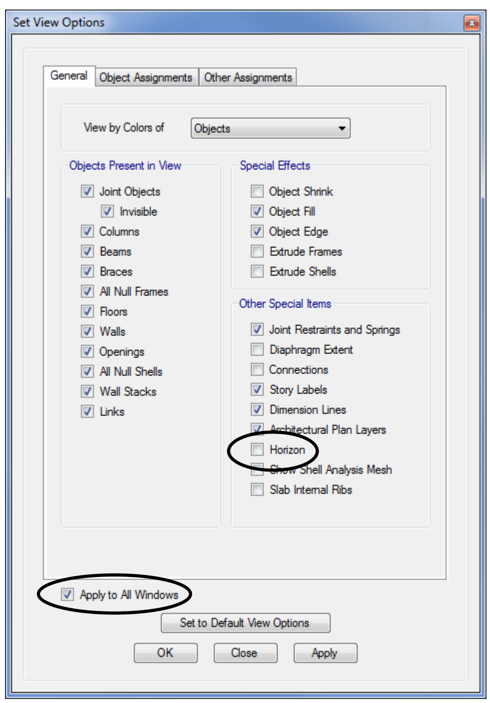


Figure 4
Set View Options form

L. Uncheck the *Horizon* option in the Other Special Items area of the General tab and check the *Apply to All Windows* option.

M. Click the **OK** button and the main ETABS window displays as shown in Figure 5.

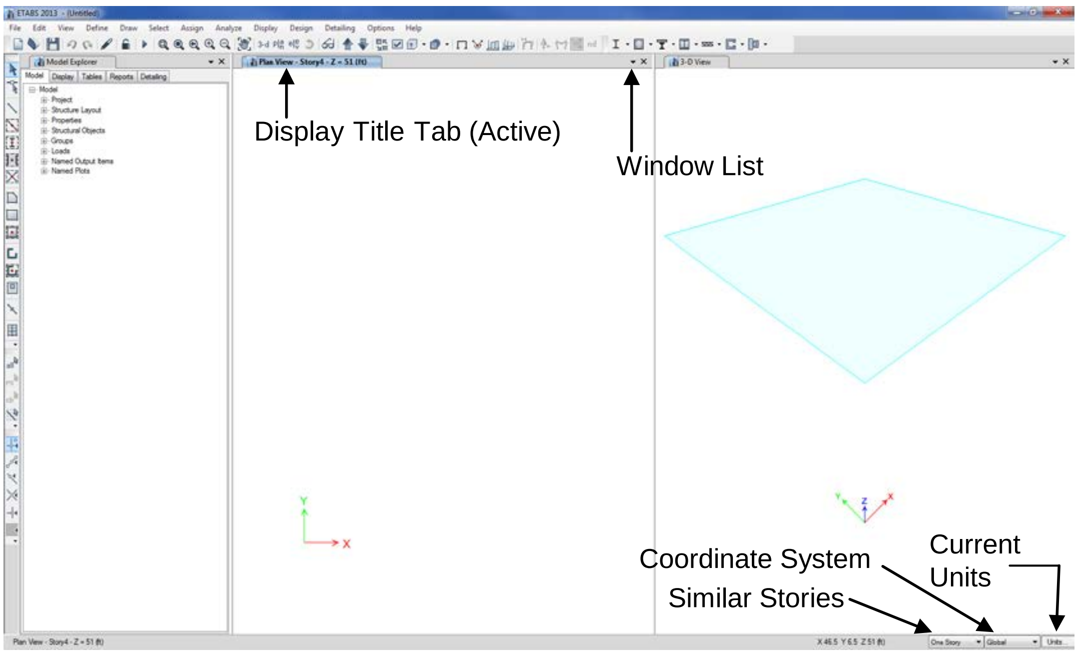


Figure 5
The ETABS main window

The model appears on screen in the main ETABS window with two view windows tiled vertically, a Plan View on the left and a 3-D View on the right, as shown in Figure 5. The number of view windows can be changed using the **Window List** button. View windows may be closed by clicking on the **Close [X]** button next to the Window List button.

Note that the Plan View is active in Figure 5. When the window is active, the display title tab is highlighted. Set a view active by clicking anywhere in the view window. The location of the active Plan View is highlighted on the 3-D View by a Bounding Plane. The Bounding Plane may be toggled on and off by using the **Options menu** > **Show Bounding Plane** command.

Although this tutorial will consist of only one tower, the default T1, ETABS allows multiple towers to exist in the same model. Additional towers may be defined by first using the **Options menu > Allow Multiple Towers** command and then the **Edit menu > Edit Towers, Stories and Grid Systems** command. Every object (columns, beams, walls, etc.) in the model will be associated with one, and only one, tower.

If you change the views, return to the default previously described, with the Plan View active as shown in Figure 5.

Edit the Horizontal Grid

Defining a grid system allows for the rapid and accurate placement of objects when drawing. Grid lines also determine object meshing and the location of elevation views.

A. Click the **Edit menu > Edit Stories and Grid Systems** command, which will display the Edit Story and Grid System Data form.

B. Highlight *G1* in the Grid Systems area and click the **Modify/Show Grid System** button to display the Grid System Data form.

C. On the Grid System Data form, click the **Quick Start New Rectangular Grids** button in the Rectangular Grids area, which will display the Quick Cartesian Grids form shown in Figure 6.

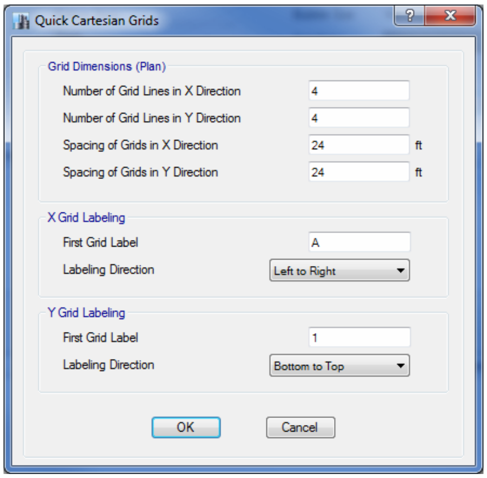


Figure 6
Quick Cartesian Grids form

D. On the Quick Cartesian Grids form, verify that the number of grid lines in each direction is set to **4**, and that the spacing of the grids in both the X and Y directions is set to **24** ft.

E. Click the **OK** button three times to display the grid.

Define an Auto Select Section List

An auto select selection list is simply a list of sections, for example, W18X35, W18X40, W21X44, W21X50 and W24X55. Auto select section lists can be assigned to frame members. When an auto select selection list is assigned to a frame object, the program can automatically select the most economical, adequate section from the auto select section list when it is designing the member.

The program has several built-in auto select section lists. Some of those lists will be used later in these instructions. Because the architect requested that the beams be no deeper than W18, it is useful to create an auto select section list that contains W16 and W18 beams now.

A. Click the **Define menu** > **Section Properties** > **Frame Sections** command, which will display the Frame Properties form shown in Figure 7.

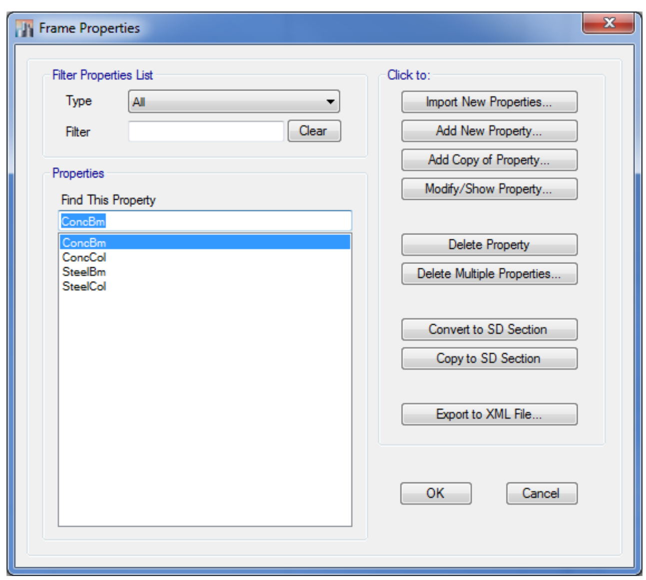


Figure 7
Frame Properties form

B. Click the **Import New Properties** button in the Click to area of the Frame Properties form. The Frame Property Shape Type form shown in Figure 8 appears.

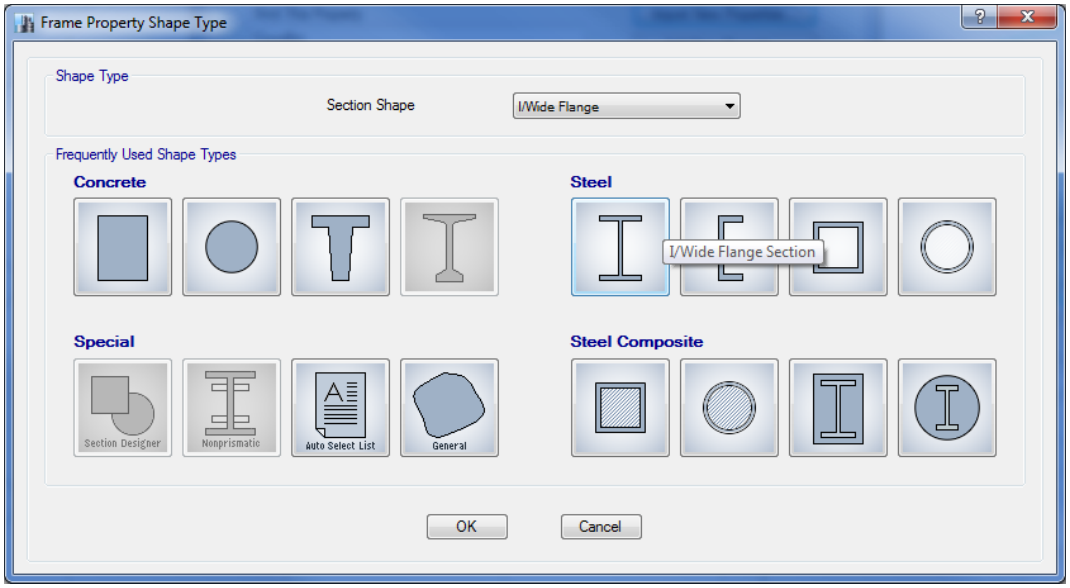


Figure 8
Frame Property Shape Type form

C. Select *Steel I/Wide Flange* from the Section Shape drop-down list in the Shape Type area and then click on the **OK** button, or click on the **I/Wide Flange Section** button under Steel in the Frequently Used Shape Types area of the Frame Property Shape Type form. The Frame Section Property Import Data form shown in Figure 9 appears.

D. Confirm that in the Filter area the Section Shape Type drop-down list shows *Steel I/Wide Flange*.

E. Scroll down the list of sections in the Select Section Properties To Import area to find the *W16X26* section. Click once on that section to highlight it. This is the first section in an auto select section list for lateral beams.

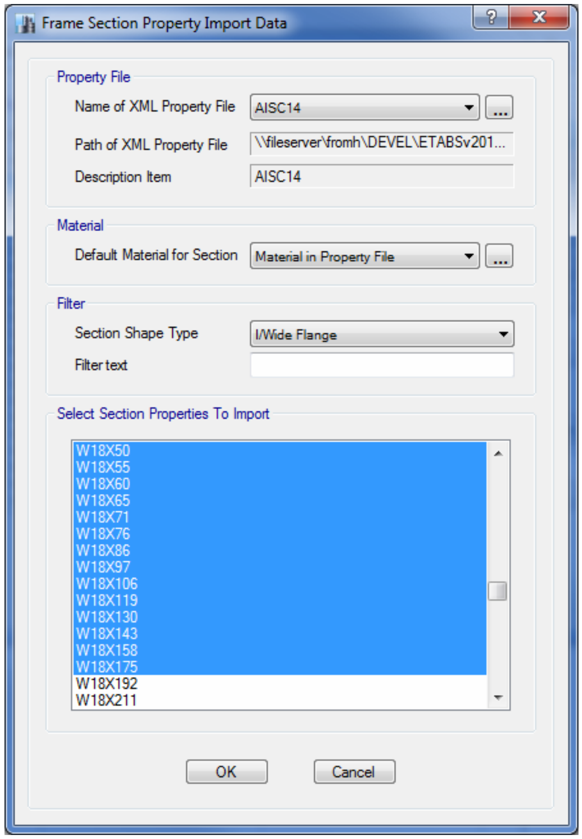


Figure 9
Frame Section Property Import Data form

F. Scroll further down the list of beam sections in the Select Section Properties To Import area to find the *W18X175* beam. Press the Shift key on your keyboard and then click once on the W18X175 beam. You should now have all of the beams between the W16X26 and the W18X175, inclusive, highlighted.

G. Click the **OK** button to return to the Frame Properties form. The Properties area should now list the sections just highlighted.

H. Click the **OK** button to close the Frame Properties form and accept the changes just made.

I. In the Model Explorer window, click on the ⊞ **Properties** node on the Model tab to expand the tree. If the Model Explorer is not displayed, click the **Options menu** > **Show Model Explorer** command.

J. On the expanded tree, <u>right</u>-click on the **Frame Sections** branch to display a context sensitive menu. On this menu, click on the **Add New Frame Property** command to display the Frame Property Shape Type form.

K. Select *Auto Select* from the Section Shape drop-down list in the Shape Type area and then click the **OK** button, or click on the **Autoselect Section List** button  under Special in the Frequently Used Shape Types area of the Frame Property Shape Type form. The Frame Section Property Data form shown in Figure 10 appears.

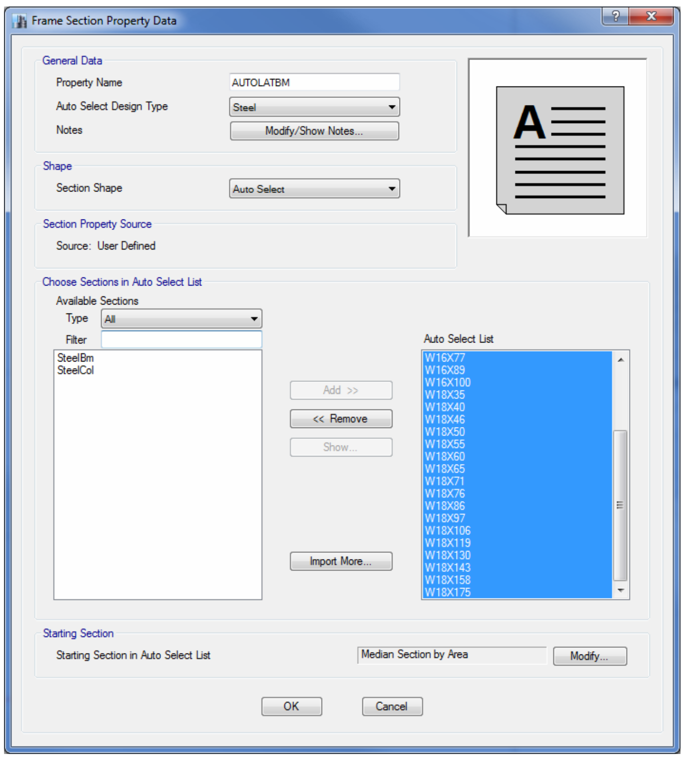


Figure 10
Frame Section Property Data form

L. Type **AUTOLATBM** in the Property Name edit box.

M. Click once on the *W16X26* section in the Choose Sections in Auto Select List area to highlight it.

N. Scroll further down the list of sections in the Available Sections to find the *W18X175* section. Press and hold the Shift key on your keyboard and then click once on the W18X175 section. You should now have all of the sections between the W16X26 and the W18X175, inclusive, highlighted.

O. Click the **Add** button to add the selected beams to the Auto Select List on the right side of the form.

P. Click the **OK** button.

Q. Click the **Define menu > Section Properties > Frame Sections** command to display the Frame Properties form.

R. Click the **Import New Properties** button to display the Frame Property Shape Type form.

S. Click on the **Autoselect Section List** button  under Special in the Frequently Used Shape Types area. The Frame Section Property Import Data form appears.

T. Click once on the *A-CompBm* section in the Select Section Properties To Import area, and while holding down the Ctrl key (*not* the Shift key) on your keyboard, click again on the *A-LatCol* section. These items are default auto select section lists provided by the program for composite beams and lateral columns, respectively.

U. Click the **OK** button to return to the Frame Properties form. The A-CompBm and A-LatCol auto select lists should be present in the properties area.

V. Click the **OK** button to accept your changes.

W. Click anywhere in the Plan View to make it active.

Step 2 Add Frame Objects

In this Step, the program is set up to add objects to multiple stories simultaneously. Then the structural objects are added to the model.

Set Up to Add Objects to Multiple Stories Simultaneously

Make sure that the Plan View is active. To make a window active, move the cursor, or mouse arrow, over the view and click the left mouse button. When a view is active, the Display Title Tab is in highlighted. The location of the Display Title Tab is indicated in Figure 5.

A. Click the drop-down list that reads *"One Story"* at the bottom right of the Main window, which is shown in Figure 5.

B. Highlight *Similar Stories* in the list. This activates the Similar Stories option for drawing and selecting objects.

C. To review the current Similar Story definitions, click the **Edit menu > Edit Stories and Grid Systems** command. The Edit Story and Grid System Data form appears. On this form, click the **Modify/Show Story Data** button to display the Story Data form shown in Figure 11. Note the Master Story and Similar To columns in the form.

   With the Similar Stories option active, as additions or changes are made to a story—for example, Story 4—those additions and changes will also apply to all stories that have been designated as Similar To Story 4 on the Story Data form. By default, the program has defined Story 4 as a Master story and, as shown in Figure 11, Stories 1, 2 and 3 are Similar To Story 4. This means that, with Similar Stories active, any drawing or selection performed on any one story will apply to all of the other stories. A story can be set as Similar To NONE so that additions or changes will not affect it.

D. We will not make any changes to the forms, so click the **Cancel** buttons two times to close the forms.

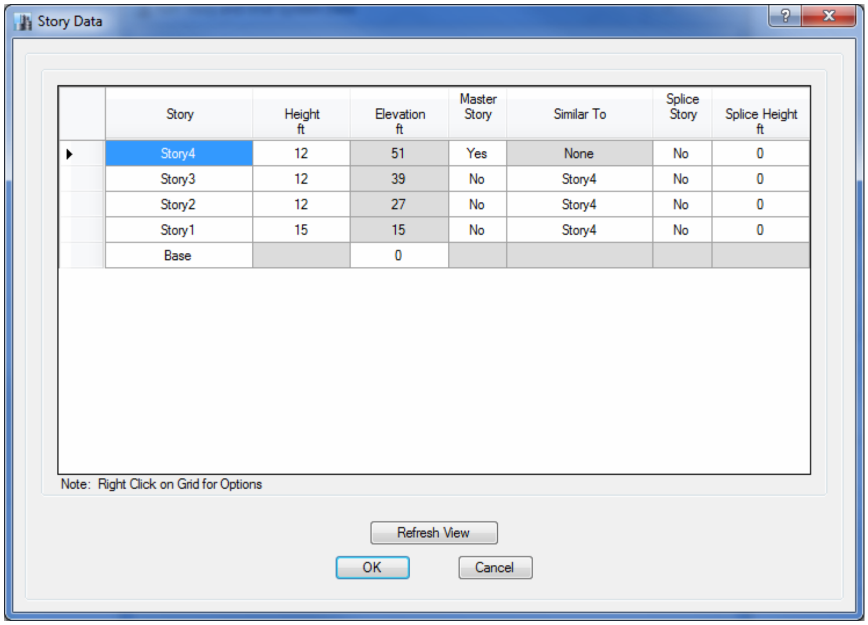


Figure 11
Story Data form

Draw Column Objects

Make sure that the Plan View is active.

A. Click the **Quick Draw Columns** button or use the **Draw menu > Draw Beam/Column/Brace Objects > Quick Draw Columns** command. The Properties of Object form for columns shown in Figure 12 will display "docked" in the lower left-hand corner of the program.

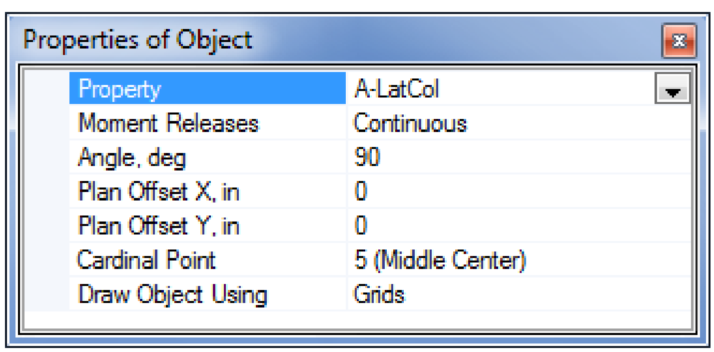


Figure 12
Properties of Object form for columns

Hold the left mouse button down on the Properties of Object tab to move the box elsewhere in the display, or to dock it at another location using the docking arrows.

B. Make sure that the Property item on the Properties of Object form is set to *A-LatCol*. If it is not, click once in the drop-down list opposite the Property item to activate and then select A-LatCol from the resulting list. A-LatCol is a built-in auto select section list intended to be used for lateral force resisting columns.

   If you want to review sections included in A-LatCol, or any of the other auto select section lists, (1) click the **Define menu** > **Section Properties** > **Frame Sections** command. The Frame Properties form will appear. (2) Highlight *A-LatCol* in the Properties list. (3) Click the **Modify/Show Property** button. The Frame Section Property Data form will display; the sections included in the A-LatCol auto select section list are listed in the Auto Select List area of the form. (4) Click the **Cancel** buttons to close the forms. Note that sections may also be reviewed using a right-click on the A-LatCol leaf under the Frame Sections branch in the Model Explorer and selecting the **Modify/Show A-LatCol** command.

C. Double click in the Angle edit box on the Properties of Object form and type **90** to set the angle to 90. This means that the columns will be rotated 90 degrees from their default position.

D. To draw the first column, left click once in the Plan View at the intersection of grid lines D and 1. An I-shaped column should appear at that point in the Plan View. Also, in the 3D View, note that the column is displayed extending over all story levels even though the column was drawn at only one story level. This occurs because the Similar Stories feature is active.

E. Click once in the Plan View at the intersection of grid lines D and 2 to draw the second column.

F. Now change the Angle item in the Properties of Object form from 90 to **0**.

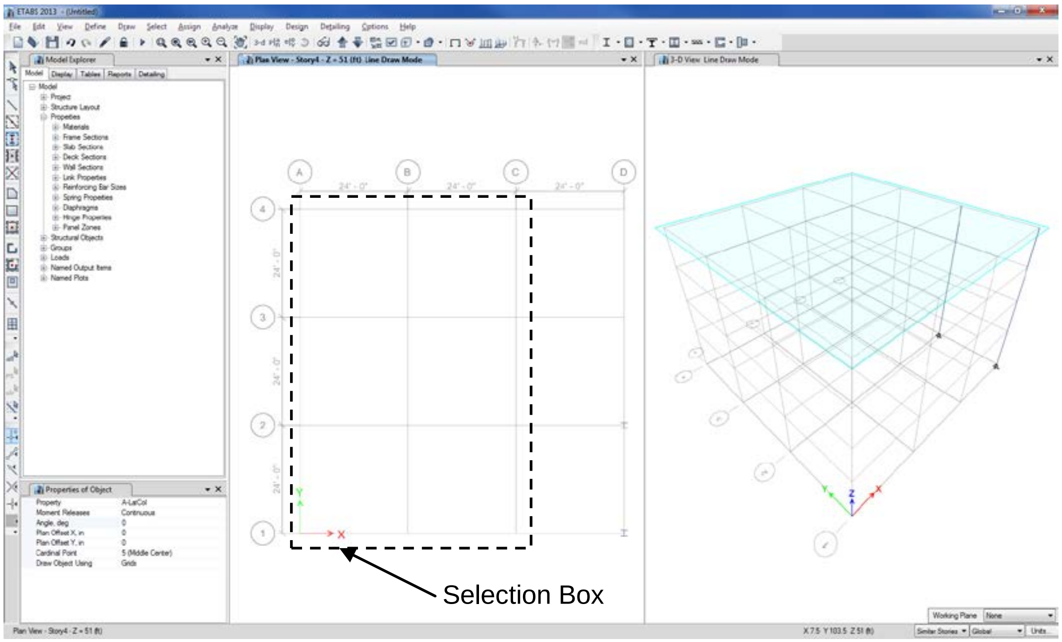


Figure 13
Drawing column objects in a windowed region

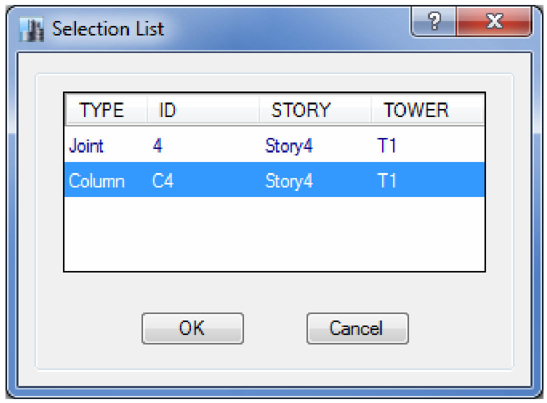
G. Now draw the remaining columns in one action by "windowing" around the grid intersections as shown in Figure 13. To "window," click the left mouse button above and to the left of grid intersection A-4 and then, while holding the left mouse button down, drag the mouse until it is below and to the right of grid intersection C-1. A selection box similar to that shown in Figure 13 should expand around the grid line intersections as the mouse is dragged across the model. Release the left mouse button and the program will draw the column objects at the grid line intersections.

   Note that these columns appear rotated 90 degrees from the first two.

H. Click the **Select Object** button, , to change the program from Draw mode to Select mode.

I. Hold down the Ctrl key on your keyboard and left click once in the Plan View on column A-2. A selection list similar to the one shown in Figure 14 pops up because multiple objects exist at the location that was clicked. In this example, a joint object and a column object exist at the same location. Note that the selection list will only appear when the Ctrl key is used with the left click.

Figure 14
Selection List form

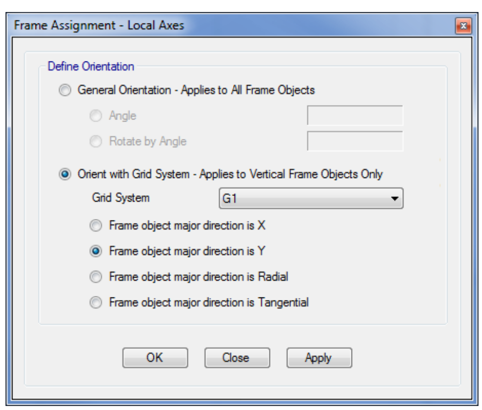


J. Select the column from the list by clicking on it and then on the **OK** button. The column at A-2 is now selected. It is selected over its entire height because the Similar Stories feature is active. Note that the status bar in the bottom left-hand corner of the main ETABS window indicates that 4 frames have been selected.

K. Repeat the selection process at B-2, A-3, C-3 and C-4. The status bar should indicate that 20 frames have been selected.

L. Click the **Assign menu** > **Frame** > **Local Axes** command to access the form shown in Figure 15.

Figure 15
Frame Assignment - Local Axis form



M. Click the *Orient with Grid System* option and then select the *Frame object major direction is Y* option in the form and then click the **OK** button. The selected columns rotate 90 degrees.

Notice the colored arrows associated with each column. Those arrows indicate the local axes directions. The red arrow is always in the local 1 direction, the green arrow is in the local 2 direction and the blue arrow is in the local 3 direction. Currently, the red arrow is not visible because it (and thus the column local 1-axis) is perpendicular to the screen.

Click the **Assign menu** > **Clear Display of Assigns** command to clear the display of the arrows.

N. Click the **Set Display Options** button . When the Set View Options form displays, check the *Extrude Frames* check box in the Special Effects area and check the *Apply to All Windows* check box followed by the **OK** button.

The model should now appear as shown in Figure 16.

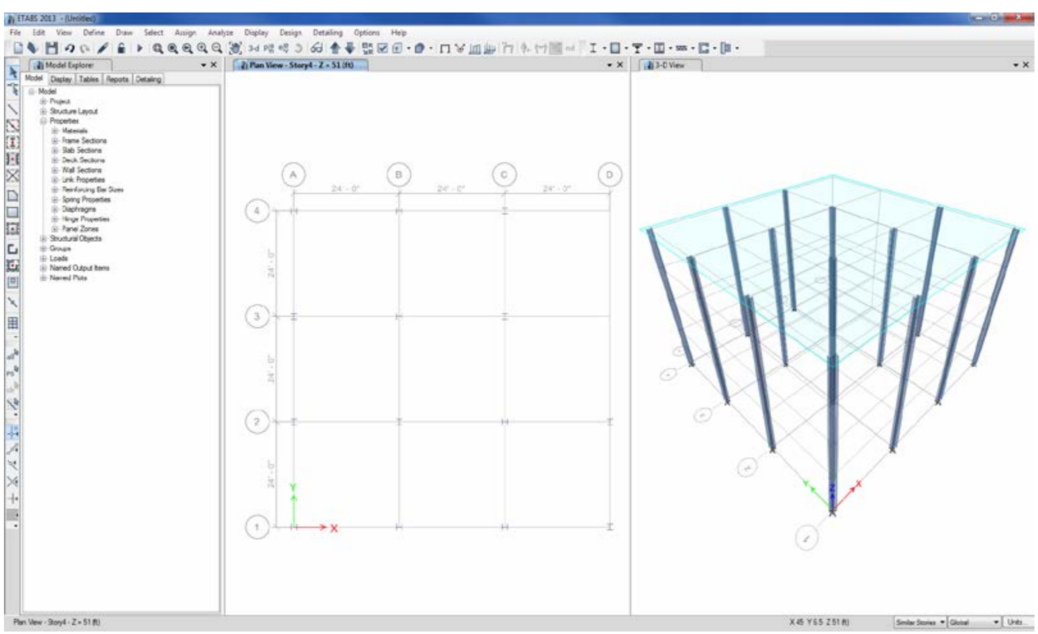


Figure 16
The example model with the columns drawn

Save the Model

During development, save the model often. Although typically you will save it with the same name, thus overwriting previous models, you may occasionally want to save your model with a different name. This allows you to keep a record of your model at various stages of development.

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

Draw the Lateral Force-Resisting Beam Objects

Make sure that the Plan View is active. Draw the beams between the columns using the following Action Items.

A. Click the **Quick Draw Beams/Columns** button, or the **Draw menu** > **Draw Beam/Column/Brace Objects** > **Quick Draw Beams/Columns** command. The Properties of Object form for frame objects shown in Figure 17 will display "docked" in the lower left-hand corner of the main window.

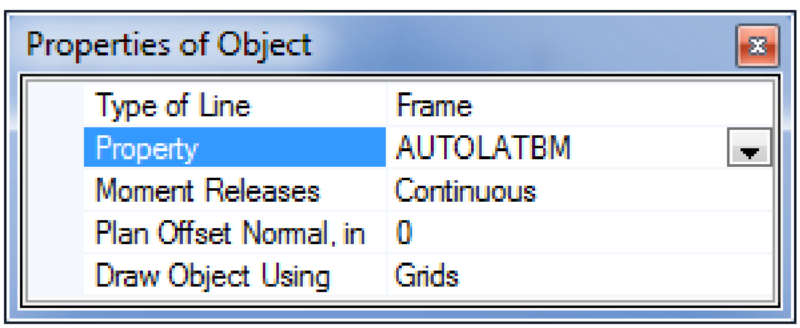


Figure 17
Properties of Object form for frame objects

B. Click once in the drop-down list opposite the Property item to activate it and then scroll down to select *AUTOLATBM* in the resulting list. Recall that AUTOLATBM is the auto select section list that was created in Step 1.

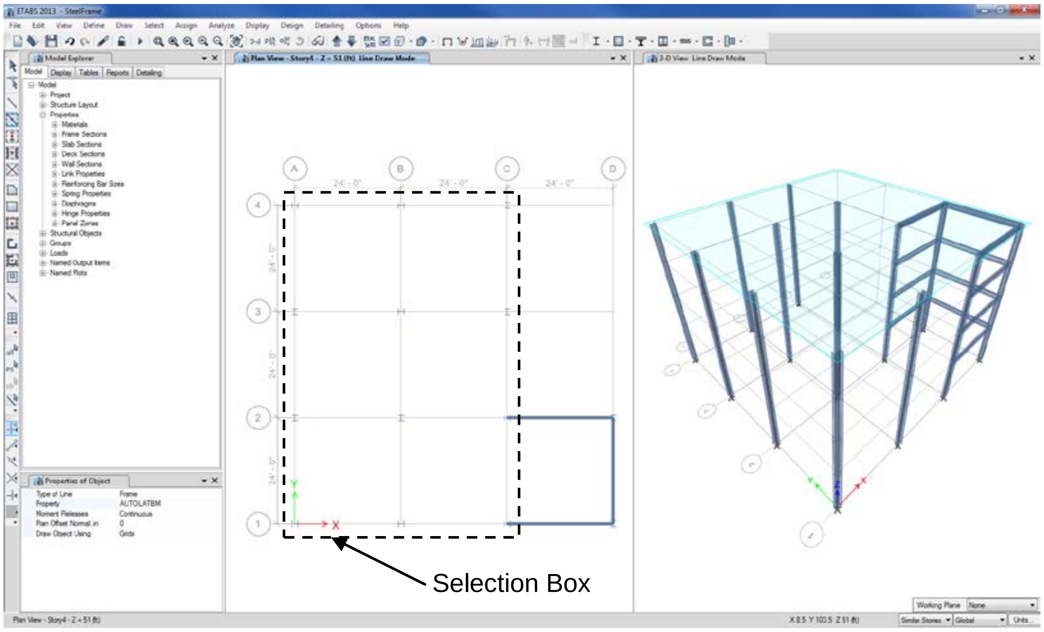


Figure 18
Drawing lateral force-resisting beam objects in a windowed region

C. Left click once in the Plan View on grid line D between grid lines 1 and 2. A beam is drawn along the selected grid line. Because the Similar Stories option is active, beams are created at all levels.

D. In a similar manner, left click once on grid line 1 between grid lines C and D and then left click once on grid line 2 between grid lines C and D to draw beams in two more locations.

E. Now draw the remaining lateral force-resisting beams in one action by windowing around the grid lines to add beams between the columns drawn earlier in Step 2, as shown in Figure 18. To window, click the left mouse button above and to the left of grid intersection A-4 and then, while holding the left mouse button down, drag the mouse until it is below and to the right of grid intersection C-1. A selection box will expand around the grid line intersections as the mouse is dragged across the model. Release the left mouse button to draw the beams.

F. Click the **Select Object** button, , to change the program from Draw mode to Select mode.

G. Left click once on the beam along grid line C between grid lines 2 and 3 to select it. Press the Delete key on your keyboard or click the **Edit menu** > **Delete** command to delete the selection because no beams should connect points C-3 and C-2 in the model.

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

Draw the Secondary (Infill) Beam Objects

Make sure that the Plan View is active. Now draw the secondary beams that span between girders using the following Action Items.

A. Click the **Quick Draw Secondary Beams** button, or the **Draw menu** > **Draw Beam/Column/Brace Objects** > **Quick Draw Secondary Beams** command. The Properties of Object form for beams shown in Figure 19 will display "docked" in the lower left-hand corner of the main window.

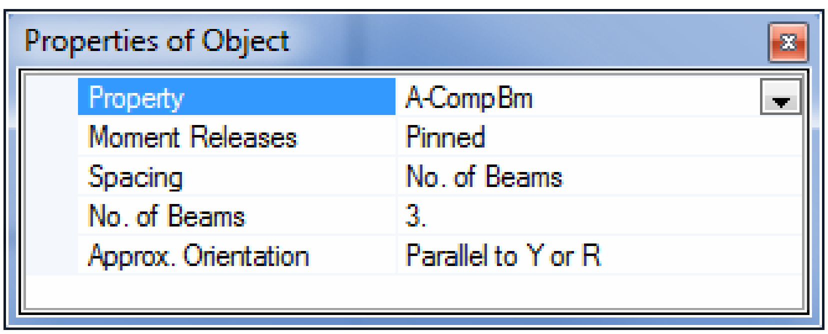


Figure 19 Properties of Object for beams

Make sure that the Property item is set to *A-CompBm*. If it is not, click once in the drop-down list opposite the Property item to activate it and then select *A-CompBm* from the resulting list. A-CompBm is a built-in auto select section list intended to be used for composite secondary beams. Review the sections included in the A-CompBm auto select list as follows: (1) click the **Define menu** >

Section Properties > **Frame Sections** command. (2) Highlight *A-CompBm* in the properties list. (3) Click the **Modify/Show Property** button; the sections in the list are displayed in the Auto Select List area of the form. (4) When finished, click the **Cancel** buttons to close both forms.

Make sure that the Approx. Orientation item in the Properties of Object form is set to *Parallel to Y or R*.

B. Left click once in the bay bounded by grid lines C, D, 1 and 2 to draw the first set of secondary beams.

C. Draw the remaining secondary beams in one action by windowing around the bays where secondary beams are to be added, as shown in Figure 20. To window, click the left mouse button above and to the left of grid intersection A-4 and then, while holding the left mouse button down, drag the mouse until it is below and to the right of grid intersection C-1. A selection box similar to that shown in Figure 20 will expand as the mouse is dragged across the model. Release the left mouse button to draw the secondary beam objects.

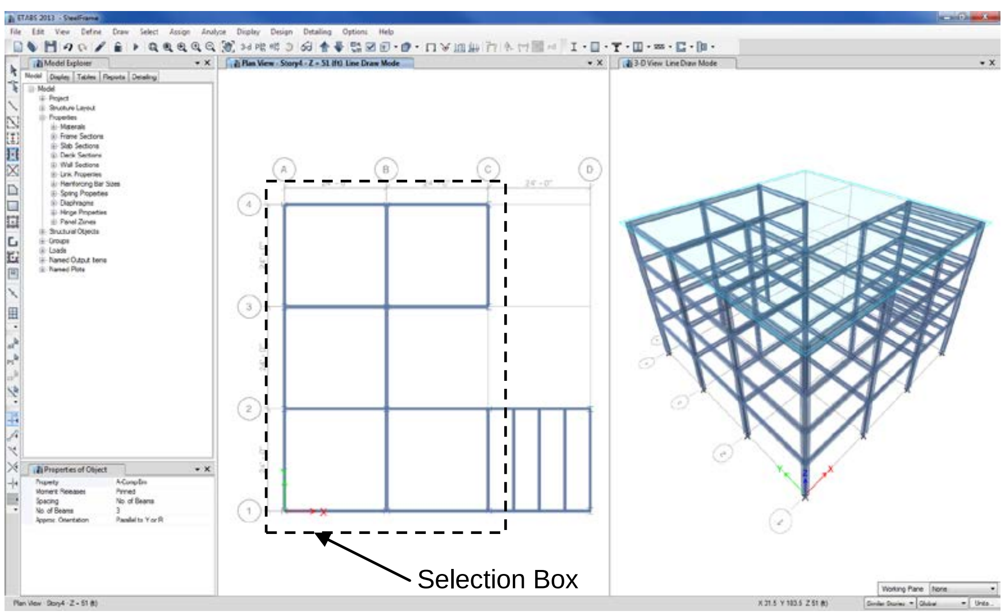


Figure 20
Drawing secondary beam objects in a windowed region

D. Click the **Select Object** button, , to change the program from Draw mode to Select mode.

E. Click the **Select using Intersecting Line** button, , or click the **Select menu > Select > Intersecting Line** command to put the program in intersecting line selection mode.

In intersecting line selection mode, left click the mouse once to start a line. Then move the mouse to another location, thus creating a selection line. When the left mouse button is double clicked, all objects that are crossed by the selection line are selected.

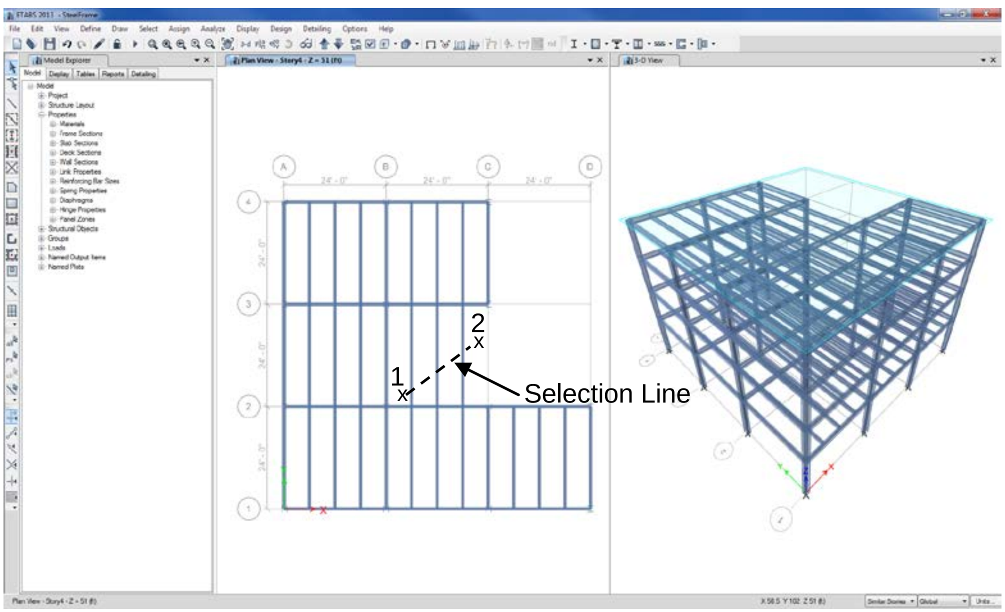


Figure 21
Selection using an intersecting line

Refer to Figure 21. Left click the mouse in the Plan View between grid lines 2 and 3 just to the right of grid line B at the point labeled 1 in the figure. Move the mouse pointer to the point labeled 2 in the figure - the selection line should be crossing the unwanted secondary beams in the bay bounded by grid lines 2, 3, B and C. Double click the left mouse button to select the beams.

F. Press the Delete key on your keyboard or click the **Edit menu** > **Delete** command to delete the selected beams from the model.

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

Step 3 Add Shell Objects

In this Step, floors are added to the model and exterior cladding is created to which wind load can be assigned in Step 8.

Draw the Floor Shell Objects

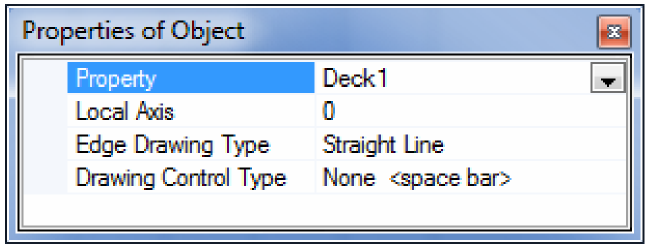
Make sure that the Plan View is active. Now draw a shell object to represent the floor using the following Action Items.

A. Click the **Set Display Options** button . When the Set View Options form displays, uncheck the *Extrude Frames* check box on the General tab and check the *Apply to All Windows* check box, as shown in Figure 22. Click the **OK** button.

Figure 22
Set View Options form

B. Click the **Draw Floor/Wall** button, , or select the **Draw menu** > **Draw Floor/Wall Objects** > **Draw Floor/Wall** command. The Properties of Object form for shells shown in Figure 23 will display "docked" in the lower left-hand corner of the main window.

Figure 23
Properties of Object form for shells



Make sure that the Property item in this box is set to *Deck1*. If it is not, click once in the drop-down list opposite the Property item to activate it and then select Deck1 in the resulting list. Deck1 is a built-in deck section property with membrane behavior. The deck properties are reviewed in a subsequent Action Item of this step.

C. Check that the **Snap to Grid Intersections & Points** command is active. This will assist in accurately drawing the shell object. This command is active when its associated button is depressed. Alternatively, use the **Draw menu** > **Snap Options** command to ensure that these snaps are active. By default, this command is active.

D. Click once at column A-1. Then, moving clockwise around the model, click once at these intersection points in this order to draw the outline of the building: A-4, C-4, C-3, B-3, B-2, D-2, and D-1. Press the Enter key on your keyboard to complete the deck object.

If you have made a mistake while drawing this object, click the **Select Object** button, , to change the program from Draw mode to Select mode. Then click the **Edit menu** > **Undo Shell Add** command. Repeat Action Items A, B and C.

Note in your model the two-headed arrow just above and to the left of column B-2 that indicates the direction of the deck span. The deck is spanning in the global X-direction, perpendicular to the secondary beams - this impacts the distribution of vertical loads to the supporting members. Note that the deck spans in the local 1-axis direction of the associated shell object.

E. Click the **Select Object** button, , to change the program from Draw mode to Select mode.

The model now appears as shown in Figure 24.

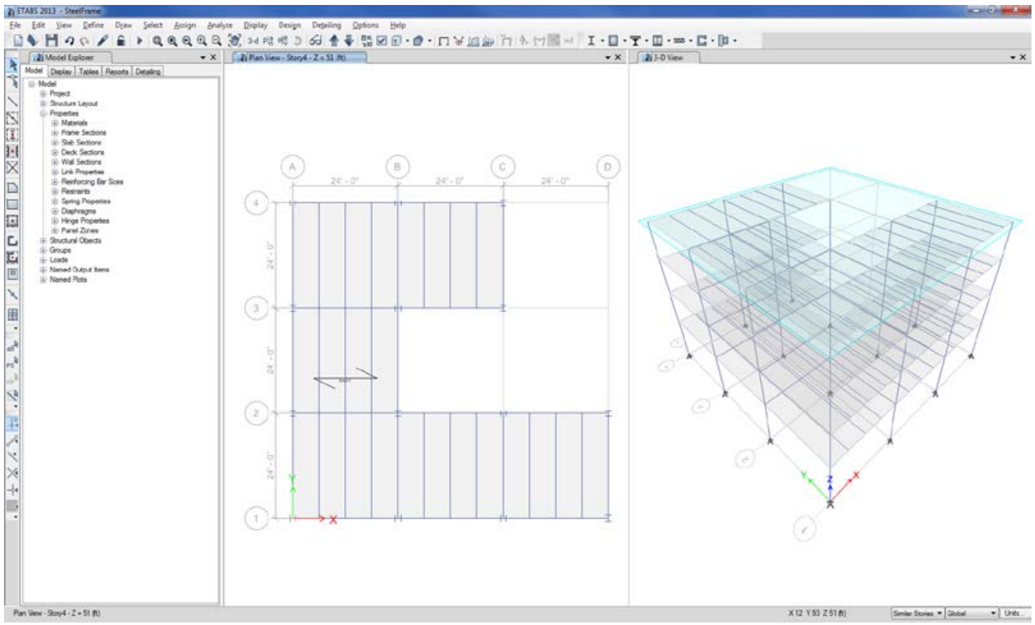


Figure 24
Model after the floor shell objects have been added

F. Review the Deck1 property that was assigned to the deck section. Click the **Define menu** > **Section Properties** > **Deck Sections** command to access the Deck Properties form.

1. Highlight the *Deck1* section and click the **Modify/Show Property** button. The Deck Property Data form shown in Figure 25 displays. Note that the Modeling Type shows as *Membrane*.

2. Set the Slab Depth, tc item to **3** to indicate that the slab depth above the metal deck is 3 inches.

3. Click the **OK** button and then click the **OK** button on the Deck Properties form to accept your changes.

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

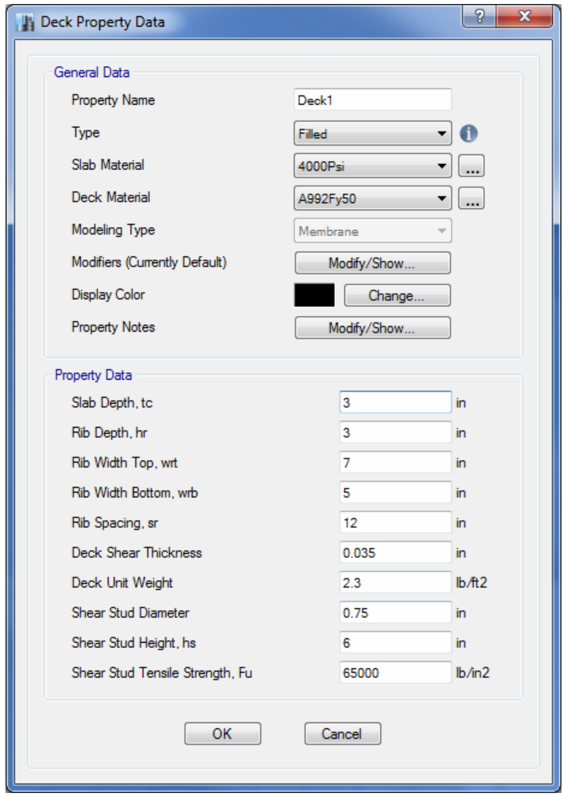


Figure 25
Deck Property Data form

Add Exterior Cladding for Wind Load Application

Exterior cladding consisting of "dummy" shell objects that have no mass or stiffness will be added to the model. The areas will be used in Step 8 to apply wind load to the building.

Draw the Cladding

Make sure that the Plan View is active. Now draw cladding around the entire perimeter of the building.

A. Click the **Draw menu** > **Auto Draw Cladding** command. The Cladding Options form shown in Figure 26 will display.

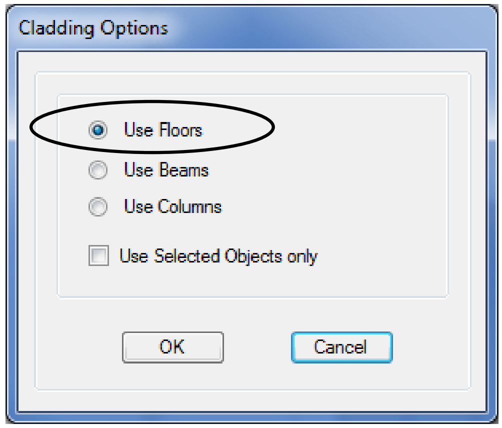


Figure 26
Cladding Options form

B. Select the *Use Floors* option and then click the **OK** button. Cladding is added around the perimeter of the structure forming a building envelope.

   In this case the building perimeter was defined by the outline of the floor objects.

C. Click on the 3-D View tab to make it active.

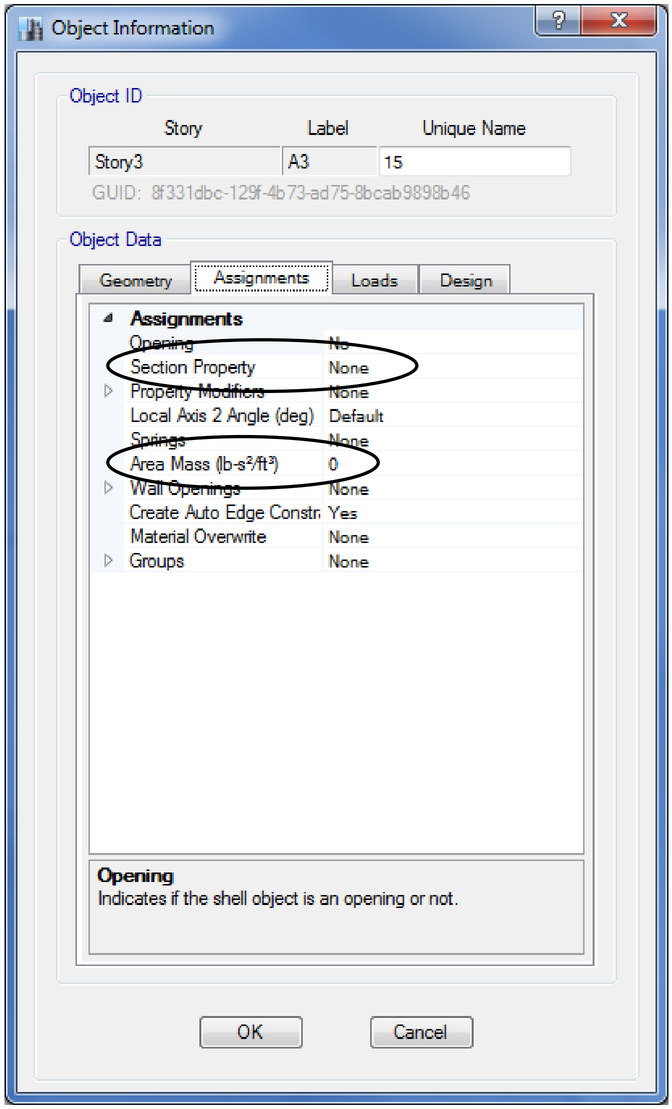
D. Click the **Set Elevation View** button and select *A* from the Set Elevation View form. Click the **OK** button to display the elevation with cladding.

E. Right-click on the cladding (not on a beam or column) in the elevation view to display the Object Information form shown in Figure 27.

   On the Object Information form, note that on the Assignments tab that the Section Property item shows *None*. This indicates that the cladding is comprised of "dummy" wall-type objects that have no stiffness.

   Also note that the Area Mass is *0*. This means that the cladding is adding no additional mass to the building. These dummy wall objects will be used solely for the purpose of applying wind loads later in the tutorial.

Figure 27
Object Information form for cladding



F. Click the **OK** button to close the Object Information form.

G. Make sure the right-hand Elevation View is active. Click on the **Set Default 3D View** button, 3-d, to change the Elevation View to a 3D View.

H. To adjust the transparency of objects, click the **Options menu > Graphics Colors > Display** command to display the Set Display Colors form.

   1. In the Set Transparency area, select a value from 0 to 1, 1 being completely transparent, from the drop-down lists for each object.

   2. Click the **OK** button to accept your changes.

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

I. Click on the Plan View tab to make it active.

Your model now appears as shown in Figure 28.

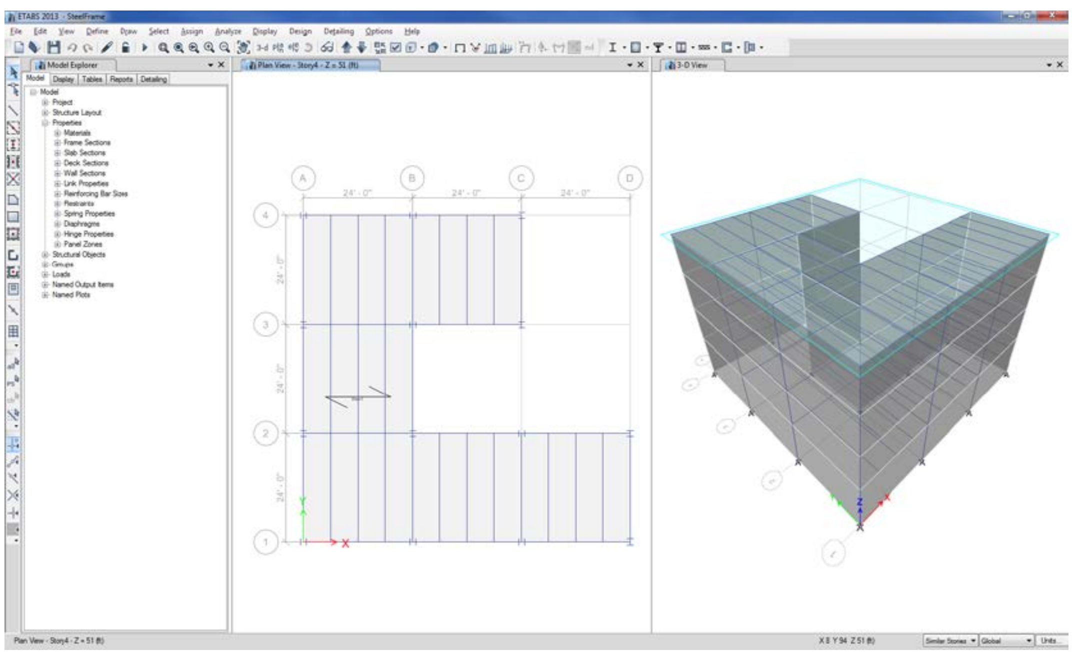


Figure 28
Model after perimeter cladding objects have been added

Step 4 Add a Wall Stack

In this Step, a wall stack is added to represent the elevator core. Wall stacks are predefined arrangements of walls that can be added to models with a single click. Make sure that the Plan View is active.

A. Click the **Draw menu > Draw Wall Stacks** command, or **the Draw Wall Stacks** button, , to access the New Wall Stack form shown in Figure 29.

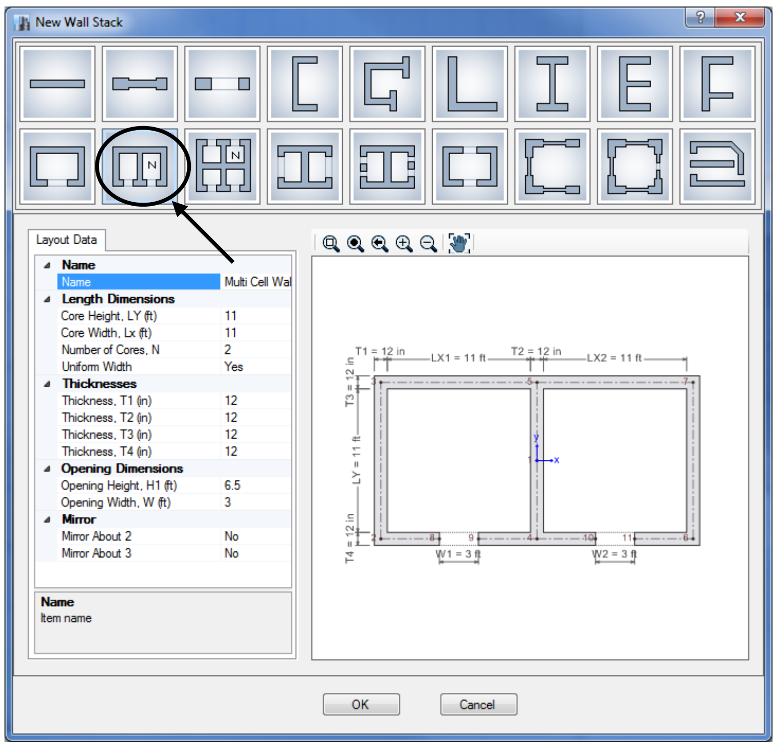
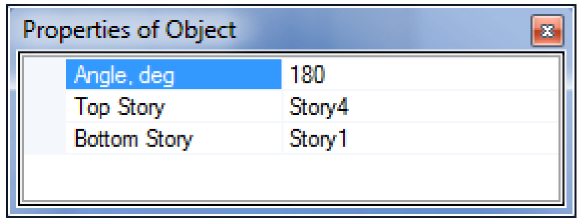


Figure 29
New Wall Stack form

B. Click on the **Multi Cell Wall** button to display a two cell core.

C. Review the information and data contained in the Layout Data tab on the New Wall Stack form, and then click the **OK** button. The Properties of Object form for Wall Stacks shown in Figure 30 will display "docked" in the lower left-hand corner of the main window.

Figure 30
Properties of Object form for Wall Stack objects



D. Click in the Angle edit box on the Properties of Object form, set the angle to **180**, and press the Enter key on your keyboard. This will rotate the wall stack object 180 degrees from the default position.

E. Left click once in the Plan View such that the top-right corner of the wall stack shown using dashed lines is located at the intersection of grid lines C and 1. This will not be where the cursor is located - the cursor is shown at the center of the wall stack.

Notice that the wall stack is also shown in the 3-D View, and that it spans the entire height of the building. The height of the wall stack can be controlled using the Top and Bottom Story drop-down lists in the Properties of Object form.

F. Click the **Select Object** button, , to change the program from Draw mode to Select mode.

G. Click the **Select menu** > **Select** > **Groups** command to display the Select by Groups form. On this form highlight *Wallstack1* and then click the **Select** button followed by the **Close** button to select the wall stack just drawn.

H. Click the **Edit menu** > **Move Joints/Frames/Shells** command to display the Move Joints/Frames/Shells form.

I. On the Move Joints/Frames/Shells form, type **-1.5** into the Delta Y edit box and click the **OK** button. This moves the wall stack 18 inches away from the building in order to isolate the core structurally.

J. Click the **File menu** > **Save** command to save your model.

Step 5 Define Static Load Patterns

The static loads used in this example consist of the dead, live, earthquake and wind loads acting on the building. An unlimited number of load patterns can be defined.

For this example building, assume that the dead load consists of the self weight of the building structure, plus 35 psf (pounds per square foot) additional dead load applied to the floors and 0.25 klf (kips per linear foot) additional dead load applied to the beams around the perimeter of the building. The 35 psf additional dead load applied to the floors accounts for items such as partitions, ceiling, mechanical ductwork, electrical items, plumbing, and so forth. The 0.25 klf additional dead load around the perimeter accounts for the cladding.

The live load is taken to be 100 psf at each story level. This live load is reducible for steel frame and composite beam design.

Note that realistically those loads would probably vary at some of the different floor levels. However, for the purposes of this example, we have chosen to apply the same load to each story level.

This example also applies an ASCE 7-10 static earthquake load to the building and an ASCE 7-10 wind load to the building. The forces that are applied to the building to account for the earthquake and wind load are automatically calculated by the program.

A. Click the **Define menu > Load Patterns** command to access the Define Load Patterns form shown in Figure 31. Note that there are two default load patterns defined. They are Dead, which is a dead load, and Live, which is a live load.

   Note that the Self Weight Multiplier is set to 1 for the Dead pattern. This indicates that this load pattern will automatically include 1.0 times the self weight of all members.

B. Click on Live to highlight the row, as shown in Figure 31. Select *Reducible Live* from the Type drop-down list. Click the **Modify Load** button to change the load type from live to reducible live. We will apply live load to the structure later.

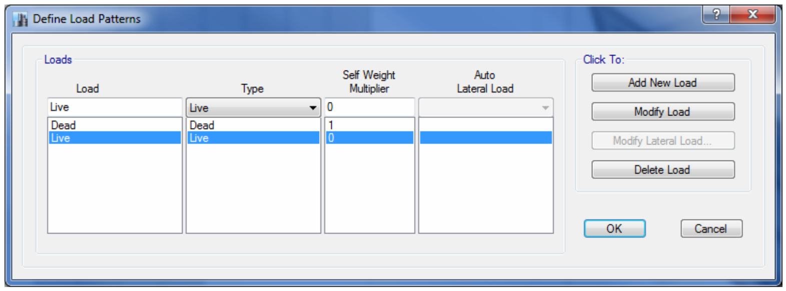


Figure 31
Define Load Patterns form

C. Double click in the edit box for the Load column. Type the name of the new load; in this case, type **Sdead**. Select a Type of load from the Type drop-down list; in this case, select *Super Dead*. Make sure that the Self Weight Multiplier is set to zero. Self weight should be included in only one load pattern; otherwise, self weight might be double counted in the analysis. In this example, self weight has been assigned to the Dead load pattern. Click the **Add New Load** button to add the Sdead load to the Load list.

D. Repeat Action Item C to add a Super Dead-type load named **Cladding**. We will apply superimposed dead load to the structure later.

E. To define the ASCE 7-10 earthquake load, double click in the edit box for the Load column again and type **Eqy**. Select *Seismic* for the Type. Make sure the Self Weight Multiplier is zero. Use the Auto Lateral Load drop-down list to select *ASCE 7-10*; with this option selected, ETABS will automatically apply static earthquake load based on the ASCE 7-10 code requirements. Click the **Add New Load** button.

F. With the Eqy load highlighted, click the **Modify Lateral Load** button. This will access the ASCE 7-10 Seismic Loading form (the ASCE 7-10 form displays because the Auto Lateral Load type was set to ASCE 7-10 in item E). On this form, uncheck all but the *Y Dir* option at the top of the form, as shown in Figure 32. Click the **OK** button. The Define Load Patterns form redisplays.

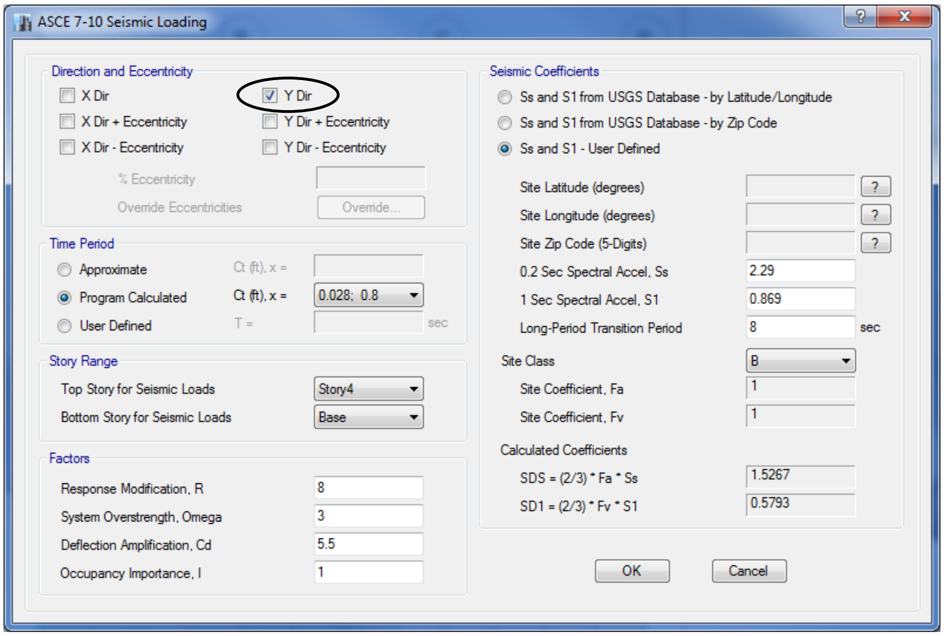


Figure 32
ASCE 7-10 Seismic Load Pattern form

G. To define the ASCE 7-10 wind load, double click in the edit box for the Load column again and type **Windx**. Select *Wind* as the Type. Select *ASCE 7-10* from the Auto Lateral Load drop-down list. Click the **Add New Load** button.

H. With the Windx load highlighted, click the **Modify Lateral Load** button. This will bring up the Wind Load Pattern - ASCE 7-10 form shown in Figure 33 (the ASCE 7-10 form displays because the Auto Lateral Load type was set to ASCE 7-10 in item G). Select the *Exposure from Frame and Shell Objects* option. Notice that the form changes, and then check the *Include Shell Objects* option.

The Exposure from Shell Objects option means that the wind load will be applied only in the direction defined by the user-specified wind pressure coefficients explicitly assigned (Step 8) to the dummy vertical shell objects that were drawn earlier. By comparison, selection of the Exposure from Extents of Rigid Diaphragms option would result in the program automatically applying all possible permutations of the ASCE 7-10 wind load to the diaphragms.

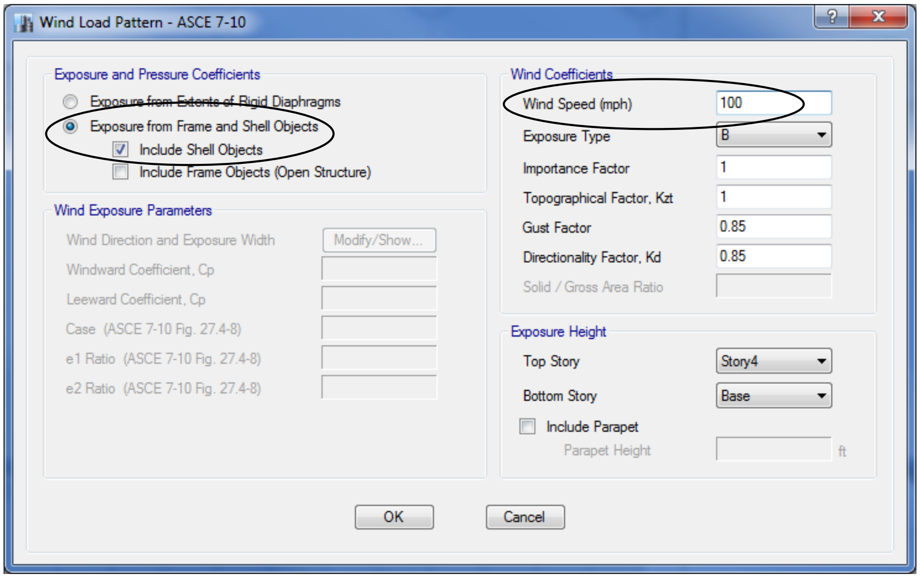


Figure 33
ASCE 7-10 Wind Load Pattern form

Type **100** into the edit box for Wind Speed, as shown in Figure 33, and then click the **OK** button. The Define Load Patterns form redisplays.

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

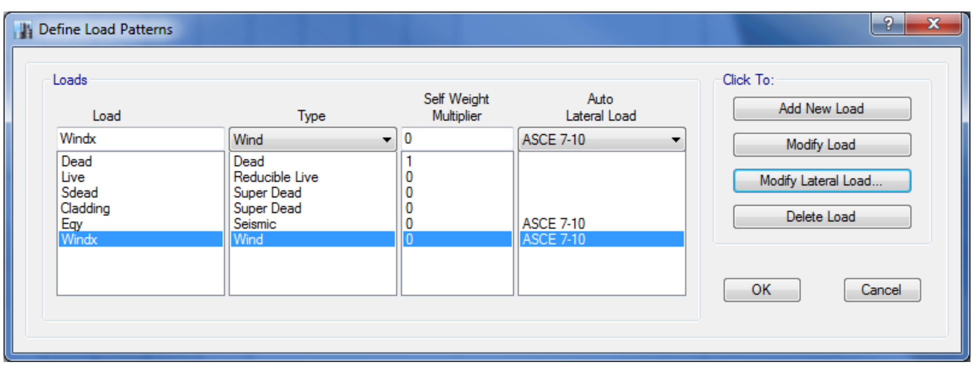


Figure 34
The Define Load Patterns form after all load patterns have been defined.

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

Step 6 Assign Gravity Loads

In this Step, the superimposed dead and live gravity loads will be applied to the model. Make sure that the Similar Stories feature is enabled and that the Plan View is active.

A. Verify that lb/ft2 are the units selected for Force/Area by holding the mouse cursor over the **Units** button in the bottom right-hand corner.

B. Click anywhere on the deck (but not on a beam) to select the deck. A dashed line should appear around the perimeter of the deck. This dashed line indicates that the deck has been selected. If you make a mistake in selecting, click the **Clear Selection** button, , and try again.

The status bar in the lower left-hand corner of the Main ETABS window should indicate that four shell objects have been selected because the Similar Stories feature is active.

C. Click the **Assign** > **Shell Loads** > **Uniform** command. This displays the Shell Load Assignment - Uniform form. Select *Sdead* from the Load Pattern Name drop-down list, as shown in Figure 35.

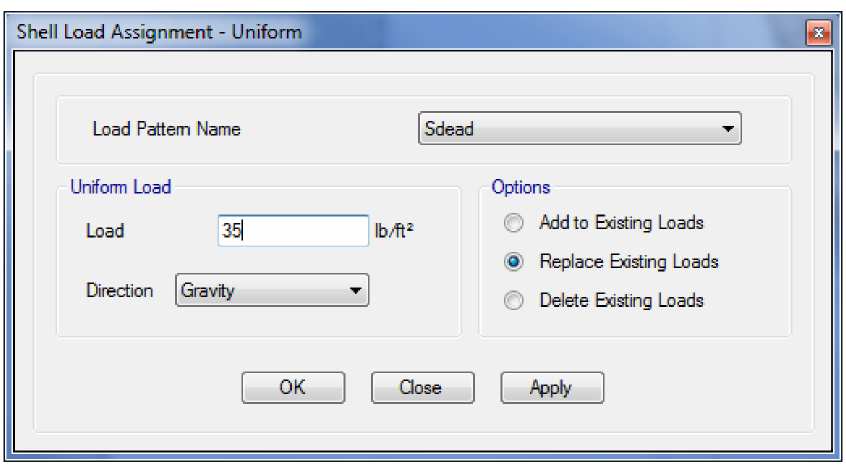


Figure 35 Shell Load Assignment - Uniform form

Note that the Direction specified for the load is Gravity. The gravity load direction is downward; that is, in the negative Global Z direction.

1. Type **35** in the Load edit box. Be sure that the units are shown as lb/ft2.

2. Click the **Apply** button on the Shell Load Assignment form to apply the superimposed dead load.

D. With the Shell Load Assignment - Uniform form still displayed, click anywhere on the deck (but not on a beam) to select the deck.

E. Select *Live* from the Load Pattern Name drop-down box.

1. Type **100** in the Load edit box. The Shell Load Assignment - Uniform form should now appear as shown in Figure 36.

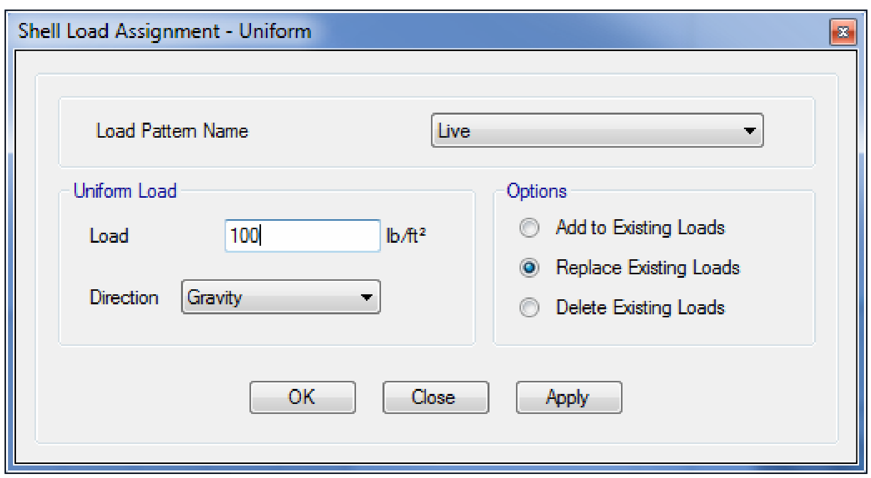


Figure 36 Shell Load Assignment - Uniform form

2. Click the **Apply** button on the Shell Load Assignment - Uniform form to accept the live load.

3. Click the **Close** button to exit the Shell Load Assignment form.

F. Make the **Snap to Grid Intersections and Points** command not active. This will make it easier to select the perimeter beams. This command is active when its associated button is depressed. Thus, make sure the button is not depressed. You can also toggle the snap feature using the **Draw menu > Snap Options** command.

G. Select the perimeter beam along grid line A between grid lines 1 and 2 by left clicking on it once in Plan View. Notice that the status bar in the lower left-hand corner of the main ETABS window indicates that four frames have been selected because the Similar Stories feature is active. Also note that the selected lines appear dashed.

H. Select the other thirteen perimeter beams in a similar manner. When you have selected all perimeter beams, the status bar should indicate that 56 frames have been selected (14 beams times 4 stories = 56 beams). Note that the Cladding load is being applied to the perimeter beams and not to the deck.

I. Click the **Assign menu > Frame Loads > Distributed** command. This displays the Frame Load Assignment - Distributed form shown in Figure 37. Select *Cladding* from the Load Pattern Name drop-down list.

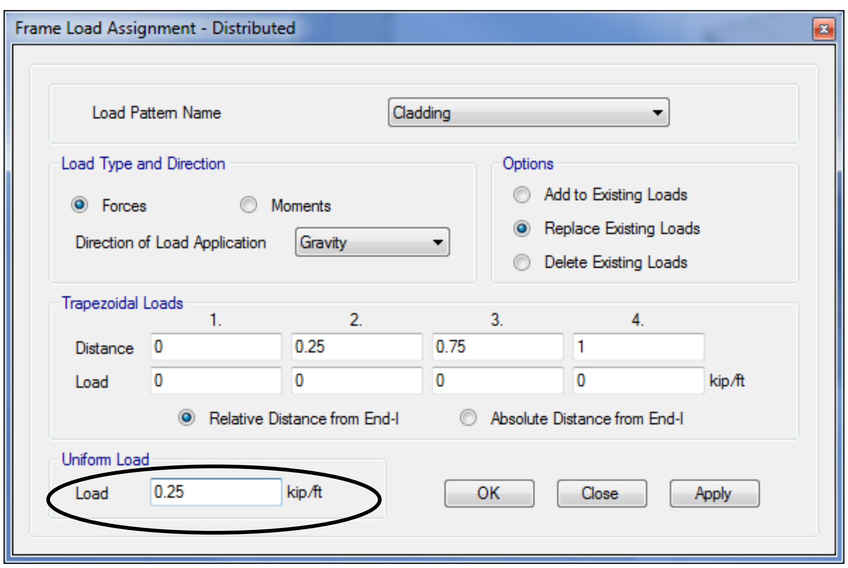


Figure 37
Frame Load Assignment - Distributed form

1. Verify that the units are set to kip/ft and then enter **0.25** in the Load edit box that is located in the Uniform Load area of the form.

2. Click the **Apply** button on the Frame Load Assignment - Distributed form to apply the uniform superimposed dead load that is applied to the perimeter beams to represent the cladding.

3. Click the **Close** button to exit the Frame Load Assignment form.

Note that the Frame Load Assignment - Distributed form also has a Delete Existing Loads option. To delete a load assignment, select the beam(s) and use the **Assign menu > Frame Loads > Distributed** command to access the form. In the Load Pattern Name drop-down list, locate the load to be removed, select the Delete Existing Loads option and click the **OK** button.

J. Make sure the Plan View is active. Click on the **Set Default 3D View** button, 3-d, to change the Plan View to a 3D View. You should now be able to graphically see the load applied to the perimeter beams, as illustrated in Figure 38.

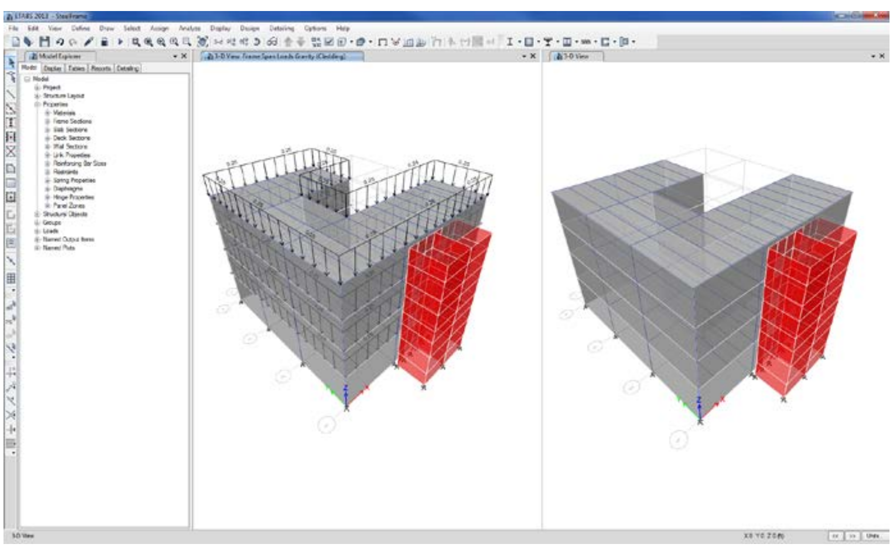


Figure 38
Frame distributed loads applied to the perimeter beams

K. Click the **Assign menu** > **Clear Display of Assigns** command to clear the display of the assigned loads.

L. Make sure the left-hand 3D view is active. Click the **Set Plan View** button and select *Story4* from the Select Plan View form. Click the **OK** button.

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

Step 7 Define a Developed Elevation

In this Step, a Developed Elevation View of the right-hand side of the building will be defined so that the wind load can be assigned to it in Step 8.

A. Click the **Draw menu** > **Draw Developed Elevation Definition** command. This displays the Developed Elevation Name form shown in Figure 39.

Figure 39 Developed Elevation Name form

1. Type **RIGHT** in the New Developed Elevation Name edit box. This will be the name of the Developed Elevation.

2. Click the **OK** button. Note that the display title tab for the plan view indicates that the program is in the Devel Elev Draw Mode. The model appears as shown in Figure 40.

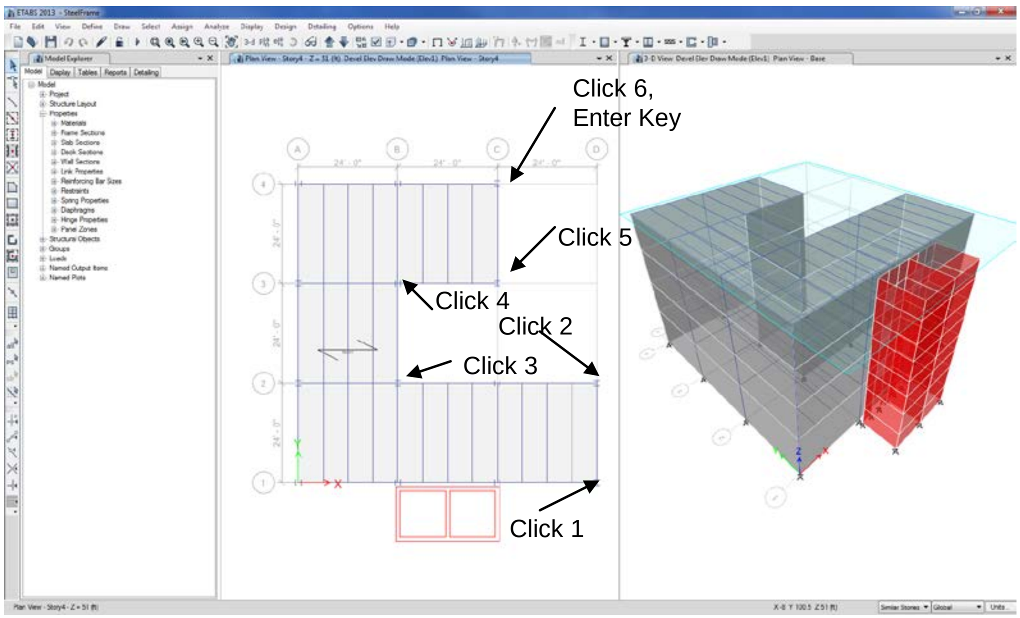


Figure 40
Developed elevation draw mode

B. Make the **Snap to Grid Intersections and Points** command active. This will assist in accurately drawing the developed elevation definition. This command is active when its associated button  is depressed. Alternatively, use the **Draw menu** > **Snap Options** command to ensure that this command is active.

C. Working in the left-hand Plan View (Devel Elev Draw Mode (RIGHT)), left click once at Grid D-1. Then moving counterclockwise around the building, left click once at D-2, B-2, B-3, C-3 and C-4 in that order. The sequence of clicks is illustrated in Figure 40. It is important to follow this exact sequence.

D. When all of the joints have been clicked, press the Enter key on your keyboard to finish drawing the developed elevation definition. The Plan View changes to an Elevation View showing the developed elevation as shown in Figure 41.

The developed elevation is an "unfolded" view of the newly defined elevation.

As many developed elevations as desired can be defined. Note however, that a developed elevation can not cross itself and it can not close on itself. Either of those situations would require that the same point occur in two different locations within the developed elevations, which is not allowed.

Note that the developed elevation is outlined in the 3-D View.

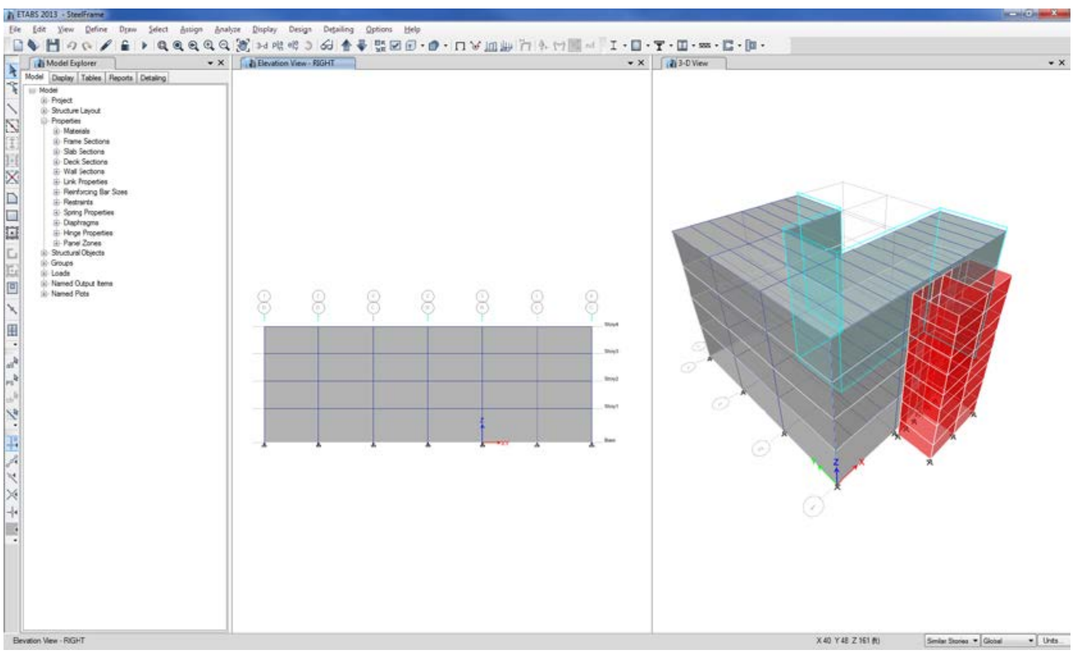


Figure 41
Developed Elevation View

After a developed elevation has been defined, it can be viewed, objects can be drawn on it, assignments can be made to objects in it, and so forth, similar to any other Elevation View. The RIGHT Elevation View will be used in the next step.

E. Make sure the Developed Elevation View (i.e., Elevation View RIGHT) is active. Click the **Set Plan View** button and select *Story4* from the Select Plan View form. Click the **OK** button.

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

Step 8 Assign Wind Loads

In this Step, wind loads are applied in the X direction to the exterior cladding along grid line A and the developed elevation created in Step 7. Typically, wind pressure coefficients are applied to the vertical surface of a shell object. In such cases, and in this example, a positive wind pressure coefficient applies wind load in the positive local 3-axis direction of the shell object. A negative wind pressure coefficient applies wind load in the negative local 3-axis direction of the shell object.

A. Click in the 3D View tab to make that window active.

B. Click the **View menu > Set Display Options** command or click the **Set Display Options** button, , to access the Set View Options form shown in Figure 42.

   1. Select the Object Assignments tab and check the *Local Axes* check box in the Shell Assignments area to turn on the shell local axes and then click the **OK** button to exit the form. Red, green and blue arrows display defining the shell object local axes. Recall that Red = 1 axis, Green = 2 axis and Blue = 3 axis.

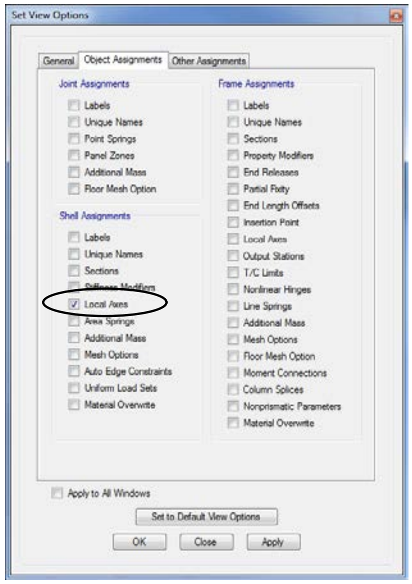


Figure 42
Set View Options form

The building appears as shown in Figure 43. Notice that for the cladding shell objects along grid line A, the blue arrows representing the local 3-axes point to the left in the negative global X direction. Note the global axes that are located at the origin of the model.

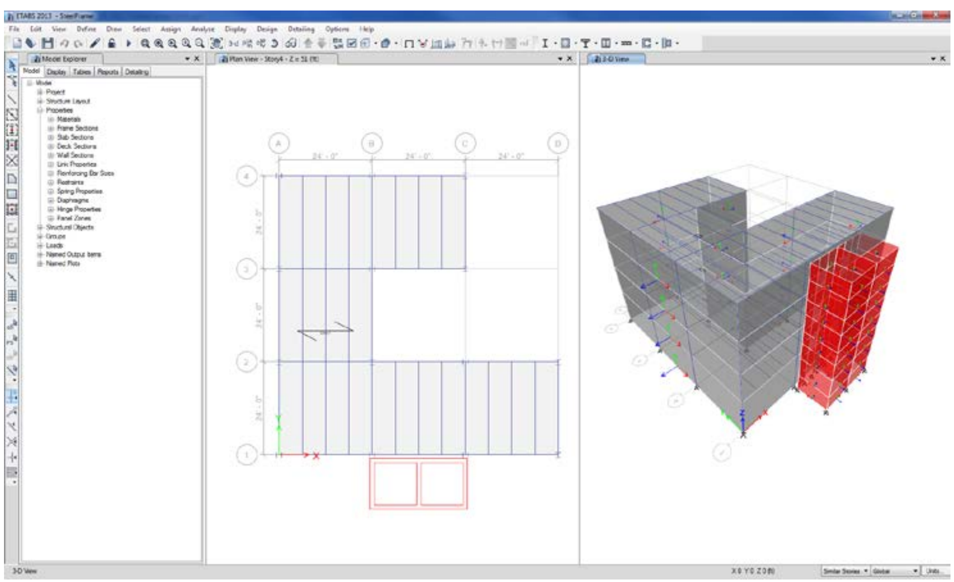


Figure 43
Shell object local axes

C. Click the **Rotate 3D View** button, , and then left click in the 3D View and hold down the left mouse button; then drag the mouse to the left. Notice how the view is being rotated.

   Rotate the view such that you can see the other cladding shell objects located on grid lines B, C and D. Confirm that the local 3 axes for those elements are pointing in the positive global X direction.

D. When you have confirmed the direction of the vertical cladding shell objects local 3 axes, click the **View menu > Set Display Options** command or click the **Set Display Options** button, , to access the Set View Options form. Uncheck the *Local Axes* check box in the Shell Assignments area on the Object Assignments tab to turn off the shell local axes display and then click the **OK** button to exit the form.

E. Make sure the 3D View is active and then click on the **Set Default 3D View** button, 3-d, to reset to the default 3D view.

F. With the 3D View active, click on the **Set Elevation View** button, , and select *A* to reset the view to an elevation of grid line A. Click the **OK** button to close the form.

G. Click the left mouse button and drag the mouse to draw a "rubber band" selection box window around all of the panels in this elevation view, as shown in Figure 44.

H. Click the **Assign menu > Shell Loads > Wind Pressure Coefficient** command, which accesses the Shell Load Assignment - Wind Pressure Coefficient form shown in Figure 45.

   1. Select *Windx* from the Wind Load Pattern Name drop-down list. Set the Coefficient, Cp to **-0.8** (make sure to use a negative sign), and choose the *Windward (varies)* option.

      Selecting the Windward option means that the wind load applied to these dummy panels will vary over the height of the building in accordance with the building code specified when the wind load was defined, in this case, ASCE 7-10.

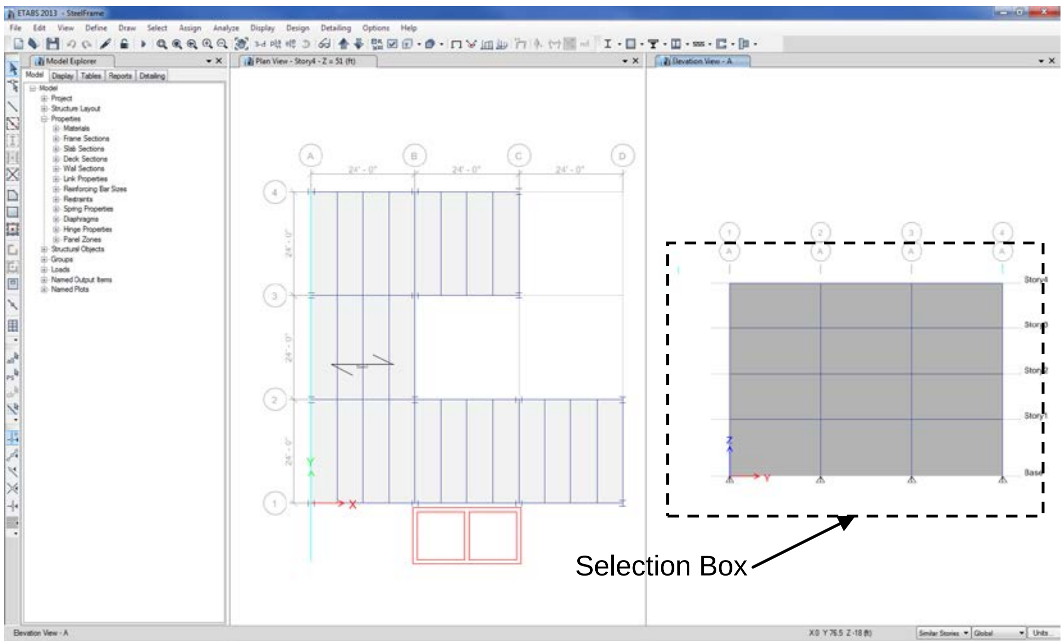


Figure 44
Selecting vertical shell objects in an elevation view

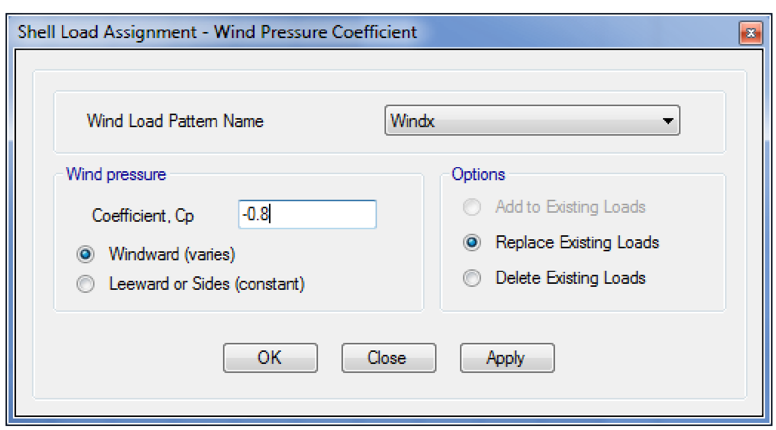


Figure 45
Shell Load Assignment - Wind Pressure Coefficient form

2. Click the **Apply** button to apply this load. Note that with the negative Cp, the load will act in the positive global X direction.

3. Click the **Close** button to exit the Wind Pressure Coefficients form.

I. With the Elevation View active, click the **Set Elevation View** button and check the *Include User Elevations* option. Highlight *RIGHT* in the Elevations area and click the **OK** button. In the Developed Elevation View, click the left mouse button once on the cladding between 1D and 2D, once between 2B and 3B, and once between 3C and 4C to select the shell objects with an X face, as shown in Figure 46. The status bar should indicate that 12 shells have been selected.

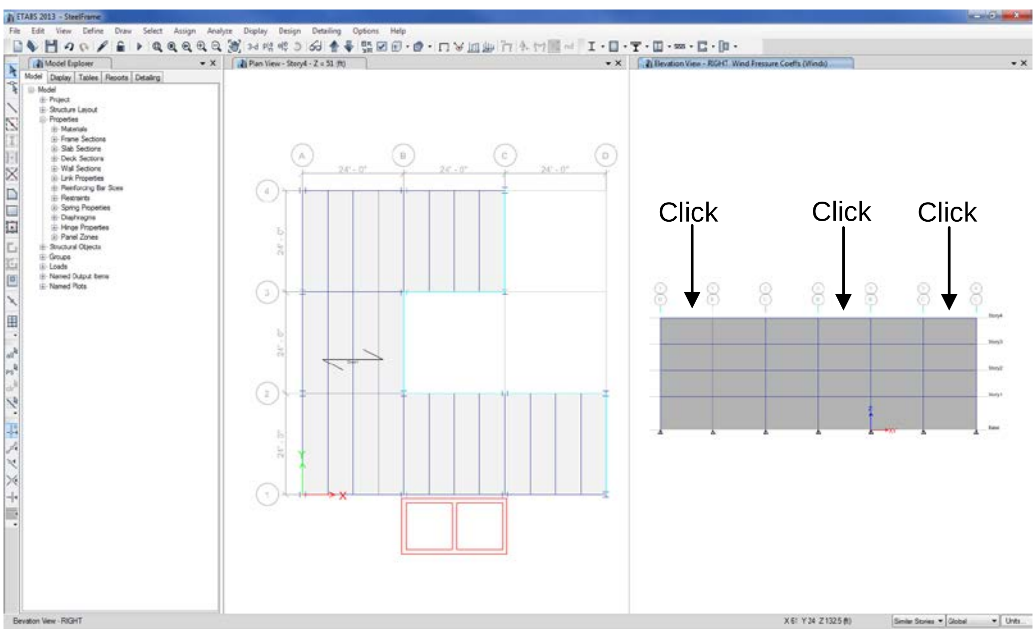


Figure 46
Selecting vertical shell objects in a developed elevation view

J. Click the **Assign menu > Shell Loads > Wind Pressure Coefficient** command, to bring up the Shell Load Assignment - Wind Pressure Coefficient form. Set the Coefficient, Cp to **0.5**, and select the *Lee-*

ward or Sides (constant) option. Click the **OK** button to apply this load and close the form. Note that with these shell objects, Cp must be positive for the load to act in the positive global X direction.

Selecting the Leeward or Sides option means that the wind load applied to these dummy panels will be constant over the height of the building in accordance with the building code specified when the wind load was defined, in this case, ASCE 7-10. The magnitude of the wind load is based on the elevation of the top of the building.

K. Click the **Assign menu** > **Clear Display of Assigns** command to clear the display of the wind pressure coefficient assignments.

L. Make sure the Elevation View is active and then click on the **Set Default 3D View** button, 3-d, to reset the view to the default 3D view.

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

Step 9 Review Tabular Display of Input Data

In this Step, a tabular display of the gravity loads that were input in Step 6 will be reviewed. Make sure that the Model Explorer window is visible; if not, click the **Options menu** > **Show Model Explorer** command.

A. Click the Tables tab in the Model Explorer to display the tables tree. Click on the **Model** node located under the Tables branch to expand the tree. The Tables tab should now look similar to that shown in Figure 47.

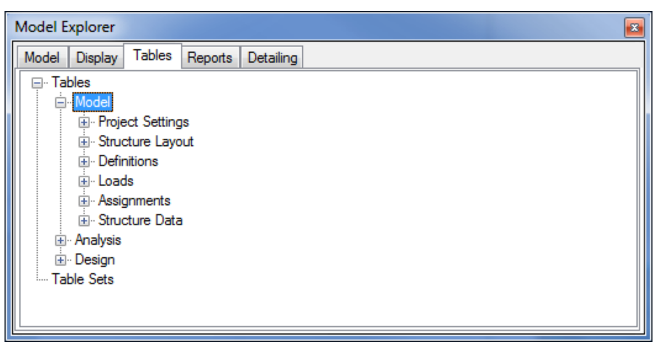


Figure 47
Model Explorer with Tables tab selected

B. Click on the **Loads** node and then on the **Applied Loads** node to expose the Shell loads - Uniform leaf. The Tables tab should now look similar to that shown in Figure 48.

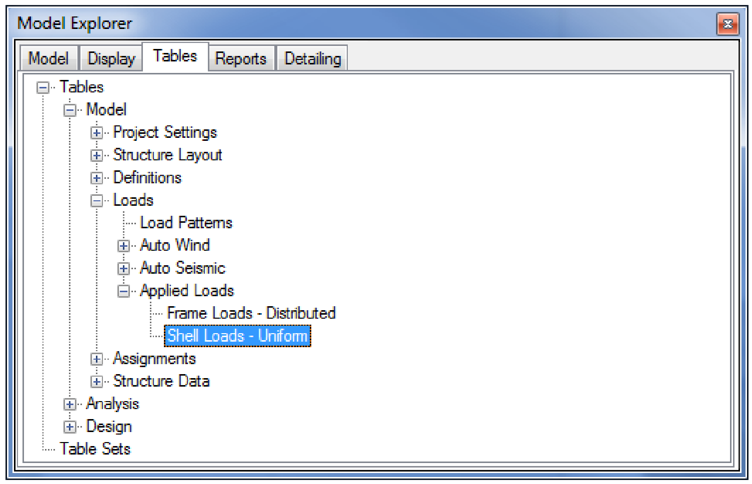
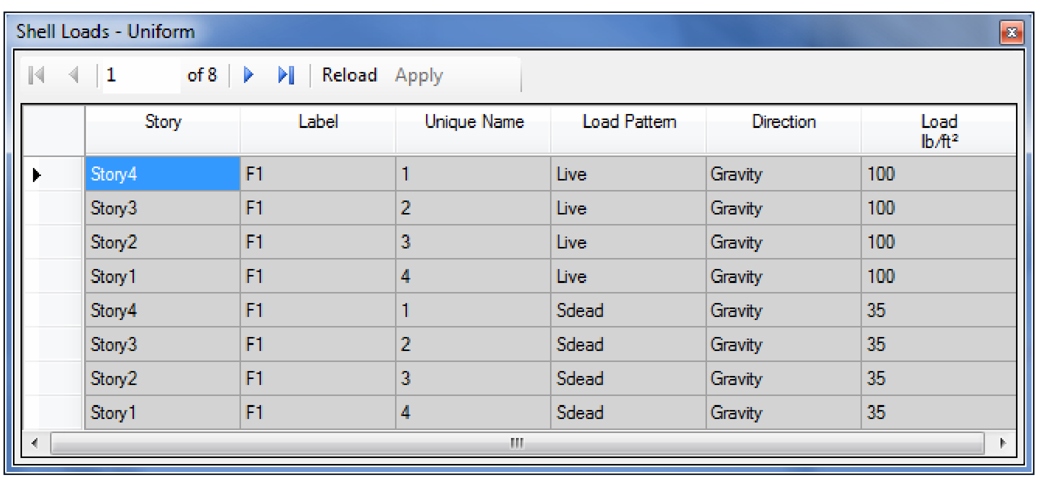


Figure 48
Model Explorer with Shell Loads - Uniform leaf showing

1. Right click on the Shell Loads - Uniform leaf and from the context sensitive menu select **Show Table**. The table shown in Figure 49 is displayed along the bottom of the program window.


Shell Loads - Uniform

1 of 8 Reload Apply

Story	Label	Unique Name	Load Pattern	Direction	Load lb/ft²
Story4	F1	1	Live	Gravity	100
Story3	F1	2	Live	Gravity	100
Story2	F1	3	Live	Gravity	100
Story1	F1	4	Live	Gravity	100
Story4	F1	1	Sdead	Gravity	35
Story3	F1	2	Sdead	Gravity	35
Story2	F1	3	Sdead	Gravity	35
Story1	F1	4	Sdead	Gravity	35

Figure 49
Table for Uniform Shell Loads

Each row in the table corresponds to a floor object at a particular story level. Notice that the sixth column in the table, labeled Load lb/ft2, displays the uniform surface loads that were input for each of the deck shell objects. The fourth column displays the load pattern, in this case either Live or Sdead, associated with the loads.

2. Right click on Frame Loads - Distributed leaf in the Model Explorer and from the context sensitive menu select **Show Table**. The table for the applied perimeter girder loads is displayed.

3. Hold the left mouse button down on the tab of a table to move it elsewhere in the display, or to dock it at another location using the docking arrows.

4. Click the **[X]** button on the title bar of the tables window to close the Frame Loads - Distributed table. The model now appears as shown in Figure 50.

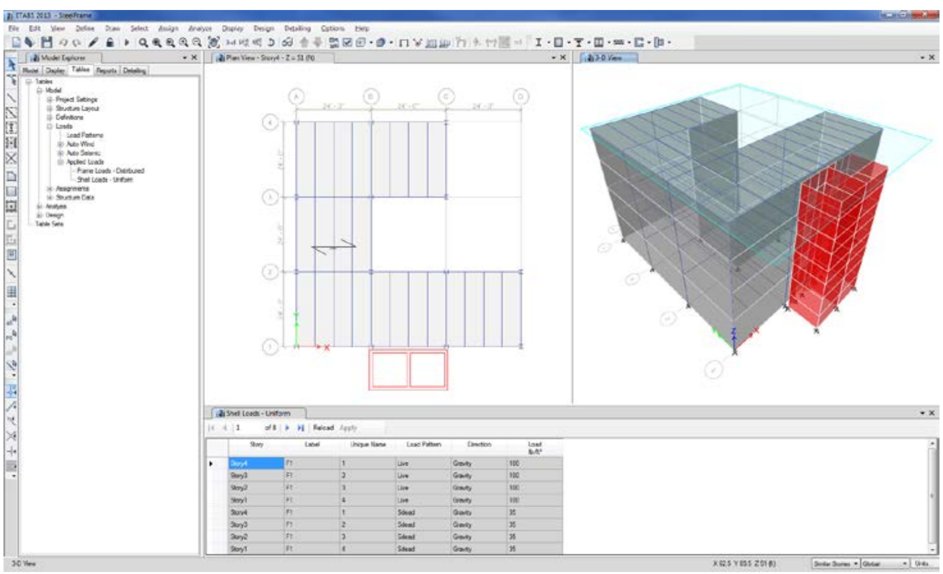


Figure 50
Model with table displayed

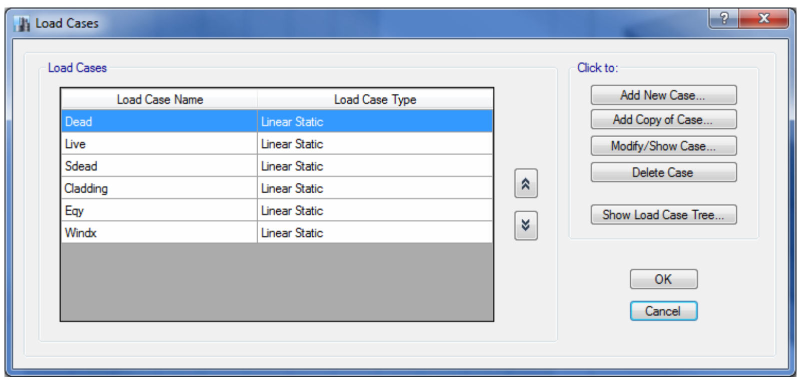
5. Click the **[X]** button on the title bar of the Shell Loads - Uniform table to close the table.

Step 10 Run the Analysis

In this Step, the analysis will be run.

A. Click the **Define menu > Load Cases** command to access the Load Cases form as shown in Figure 51. ETABS automatically generates load cases for each of the previously defined load patterns. Review and verify that the load cases are appropriate, and then click the **OK** button to exit the form.

Figure 51
Load Cases form



B. Click the **Analyze menu > Set Load Cases to Run** command to access the Set Load Cases to Run form. Verify on this form that the Action for each case is set to Run, and then click the **OK** button.

C. Click the **Analyze menu > Run Analysis** command or the Run Analysis button, .

The program will create the analysis model from your object-based ETABS. After the analysis has been completed, the program performs a few bookkeeping actions that are evident on the status bar in the bottom left-hand corner of the ETABS window.

When the entire analysis process has been completed, the model automatically displays a deformed shape view of the model, and the model is locked. The model is locked when the Lock/Unlock Model button, , appears locked. Locking the model prevents any changes to the model that would invalidate the analysis results.

Step 11 Graphically Review the Analysis Results

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

A. Make sure the 3D View is active. Then click on the **Set Elevation View** button, , and select *1* and click the **OK** button to reset the view to an Elevation View of grid line 1.

B. Click the **Display Frame... Forces** button, , or the **Display menu > Force/Stress Diagrams > Frame... Forces** command to access the Member Force Diagram form shown in Figure 52.

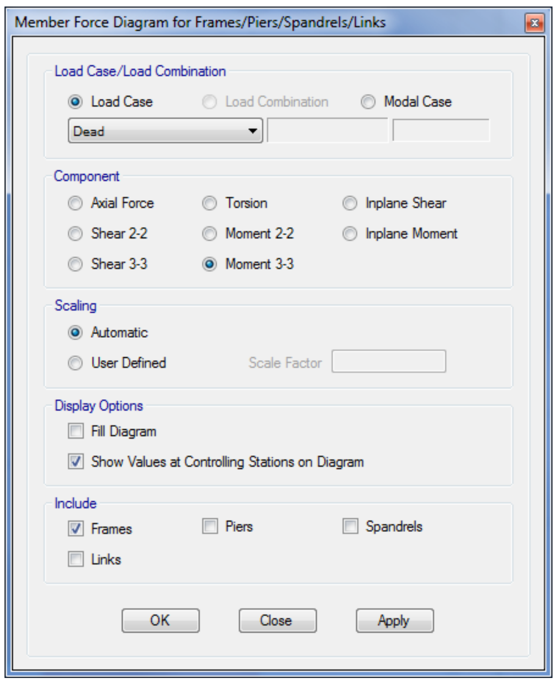


Figure 52
Member Force Diagram form

1. Select *Dead* from the Load Case drop-down list.

2. Select the *Moment 3-3* component.

3. Uncheck the *Fill Diagram* if it is checked.

4. Check the *Show Values at Controlling Stations on Diagram* check box.

5. Click the **OK** button to generate the moment diagram output shown in Figure 53.

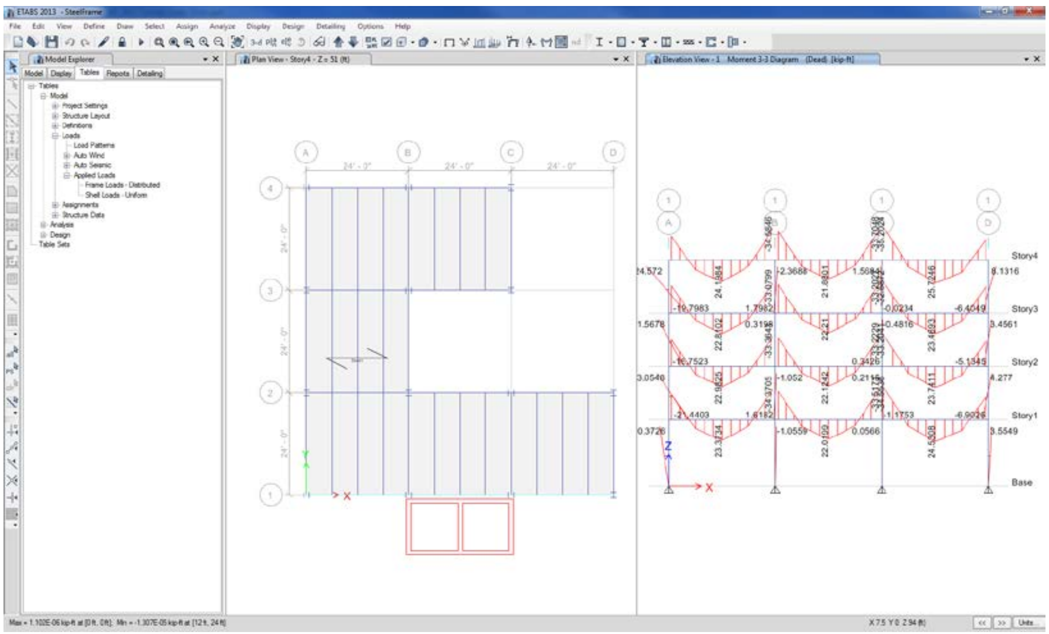


Figure 53
M33 moment diagram in an elevation view

Note that these moment diagrams are plotted with the moment on the tension side of the member. Change this, if desired, using the **Options menu > Moment Diagrams on Tension Side** command toggle.

C. Right click on the top level beam between grid lines A and B to access the Diagram for Beam form shown in Figure 54.

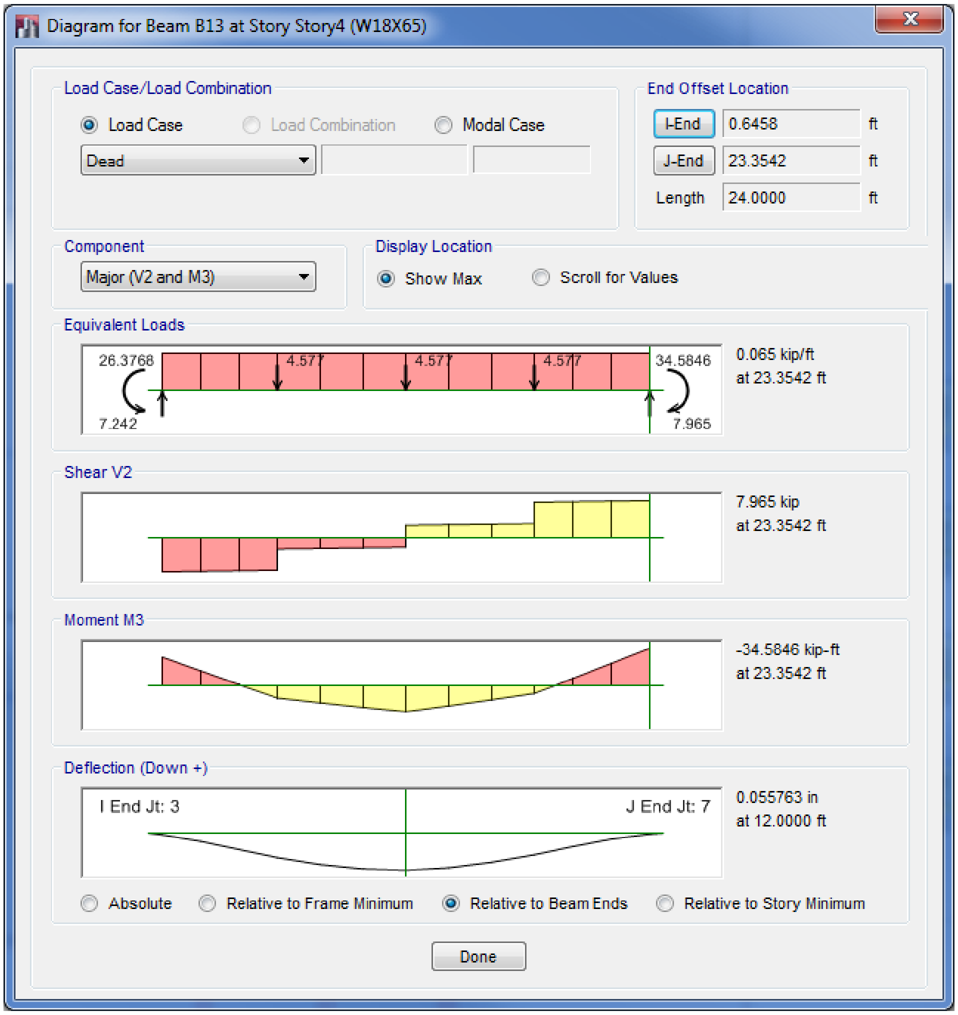


Figure 54
Force details obtained by right-clicking a beam shown in the elevation view in Figure 53

Note that the applied load, shear, moment and deflection are shown for the beam, and the maximum values are identified on the Diagram for Beam form.

1. Click the *Scroll for Values* option in the Display Location area and a scroll line appears in each diagram. Drag the scroll line with your mouse to see values at different locations along the beam.

2. Type **6.5** into the Display Location edit box and press the Enter key. The load, shear, moment and deflection values are displayed at this exact location along the beam.

3. Click the Load Case drop-down list and select *Cladding* from the list to display the forces acting on this beam from the superimposed dead load named Cladding. The Equivalent Loads should display a value of 0.250 kip/ft, which is the cladding load that was applied in Step 6.

4. Click the **Done** button to close the form.

D. Make sure the Elevation View is active and then click the **Display menu > Undeformed Shape** command or the **Show Undeformed Shape** button, , to clear the display of the moment diagrams in the Elevation View.

E. Make sure the Elevation View is active and then click on the **Set Default 3D View** button, 3-d, to reset the view to the default 3D view as shown in Figure 55.

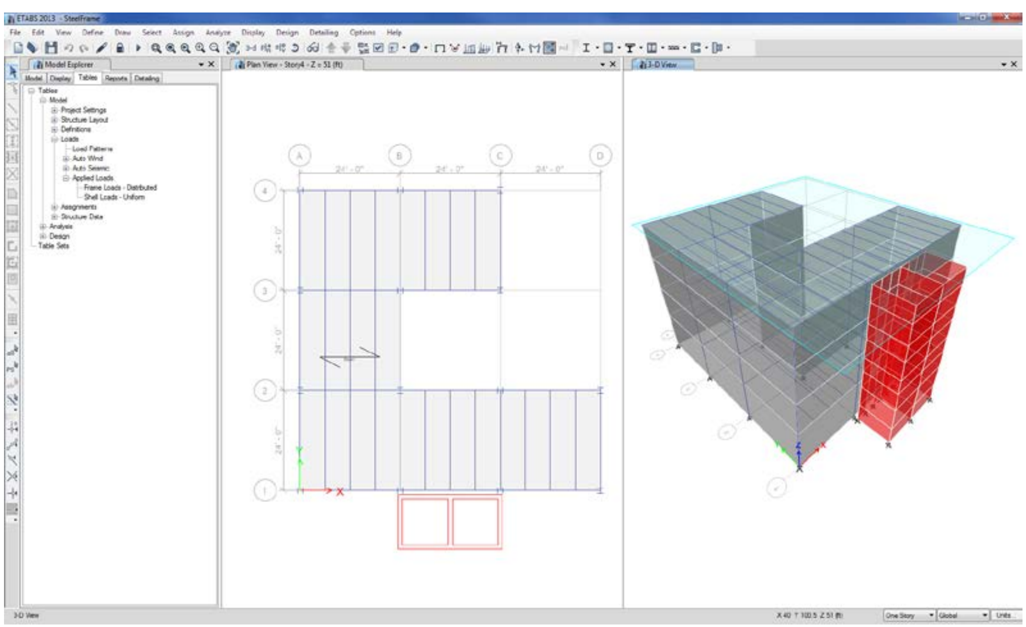


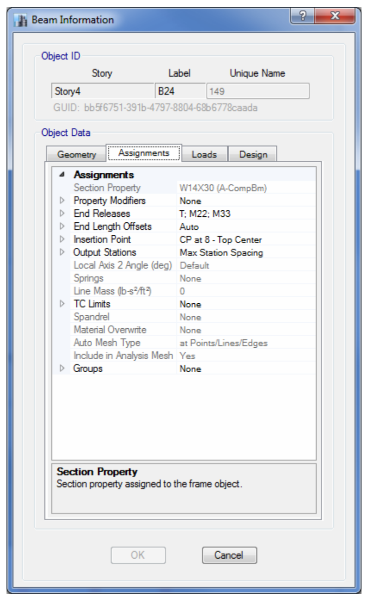
Figure 55
Undeformed Shape

Step 12 Design the Composite Beams

In this Step, the composite beams will be designed. Note that the analysis (Step 10) should be run before performing the following Action Items.

A. In the Plan View, right click on one of the secondary (infill) beams in the bay bounded by grid lines 1, 2, A and B. The Beam Information form shown in Figure 56 appears.

Figure 56
Beam Information form



Note that the Design tab reports that the Design Procedure is Composite Beam. The program assigned this default design procedure to this frame object because (1) it lies in a horizontal plane, (2) the ends of the beam are pinned (that is, moment is released at each end of the beam), and (3) it is assigned a steel section that is either I-shaped or a channel.

To change the design procedure for a beam, select the beam and use the **Design menu > Overwrite Frame Design Procedure** command.

Review the information available on all four tabs of the Beam Information form and then click the **Cancel** button to close the form.

B. Click the **Design menu > Composite Beam Design > View/Revise Preferences** command. The Composite Beam Design Preferences form shown in Figure 57 displays.

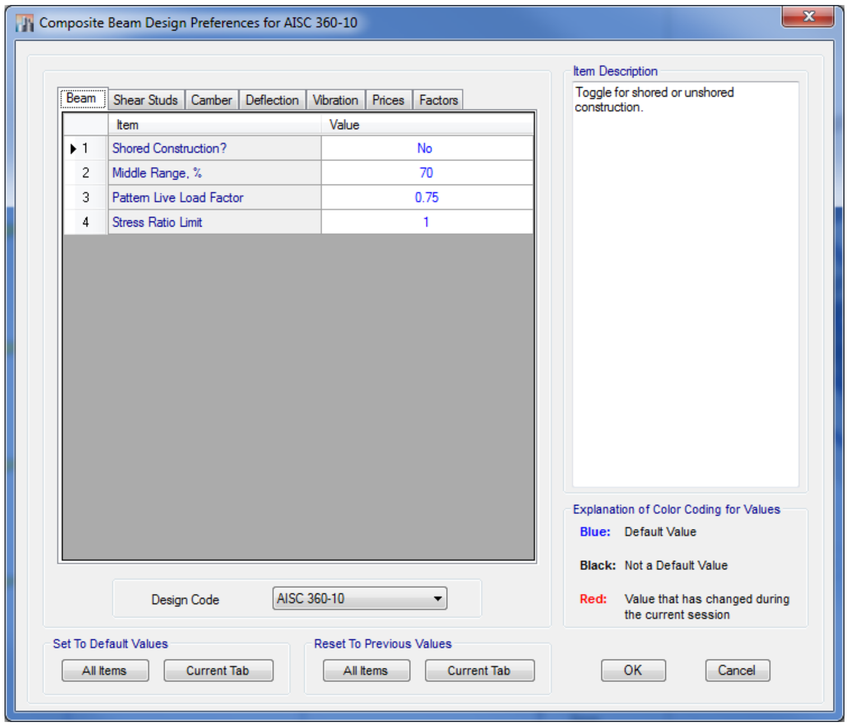


Figure 57
Preferences form for composite beam design

1. Click the Design Code drop-down list near the bottom of the form to review the available design codes. Select the *AISC-360-10* code.

2. Review the information available on all seven tabs in the Composite Beam Design Preferences form and then click the **OK** button to accept any changes made to the form.

C. Click on the title tab of the 3D View to make it active.

D. Click the **Set Display Options** button  When the Set View Options form displays, uncheck the *Object Fill* check box as shown in Figure 58. This will remove the display of the fill in the shell objects.

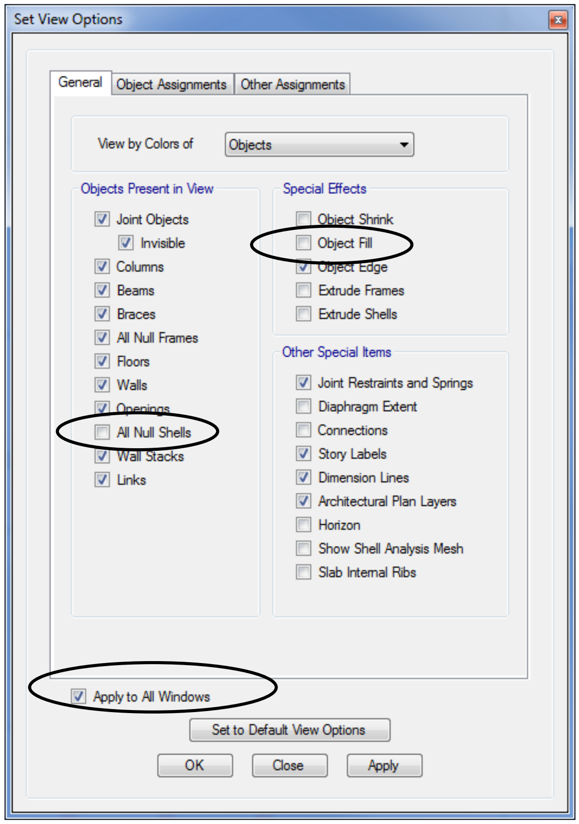


Figure 58
Set View Options form

1. In the Objects Present in View area of the form, uncheck the *All Null Shells* check box.

2. Check the *Apply to All Windows* check box and click the **OK** button to accept the changes.

E. With the 3D View active, click the **Design menu > Composite Beam Design > Start Design/Check** command to start the design process. The program designs the composite beams, selecting the optimum beam size from the A-CompBm auto select section list that was assigned to them when they were drawn in Step 2.

When the design is complete, the selected sizes are displayed on the model. The model appears as shown in Figure 59.

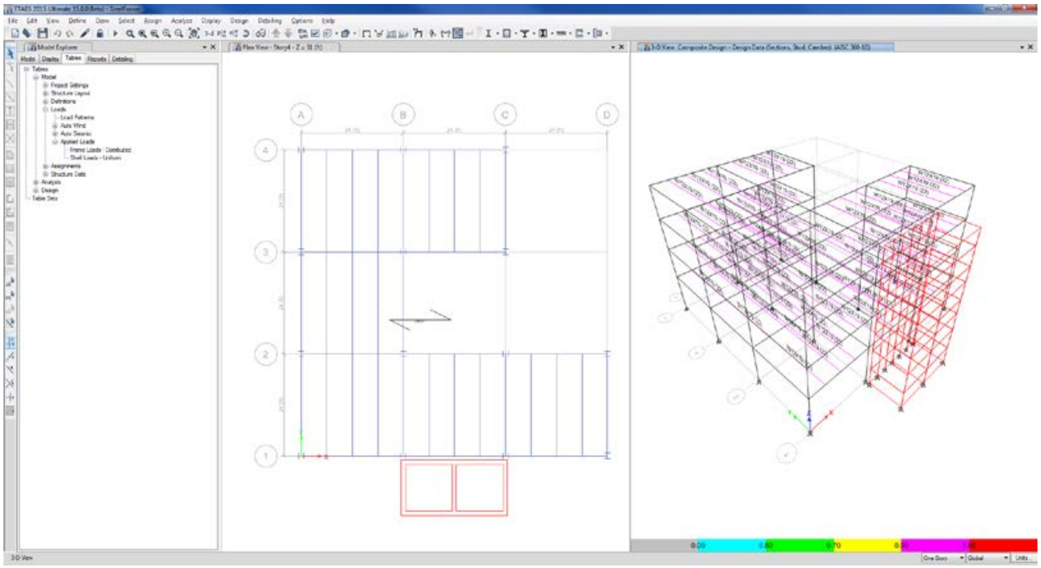


Figure 59
Model after the initial composite design

F. Click the **Design menu > Composite Beam Design > Verify Analysis vs Design Section** command. A message similar to the one shown in Figure 60 appears.

In the initial analysis (Step 10), the program used the median section by weight from the A-CompBm auto select section list. During design (this Step), the program selected a W12X19 design section, which differs from the analysis section used. The message in Figure 60 indicates that the analysis and design sections are different. Click the **No** button to close the form.

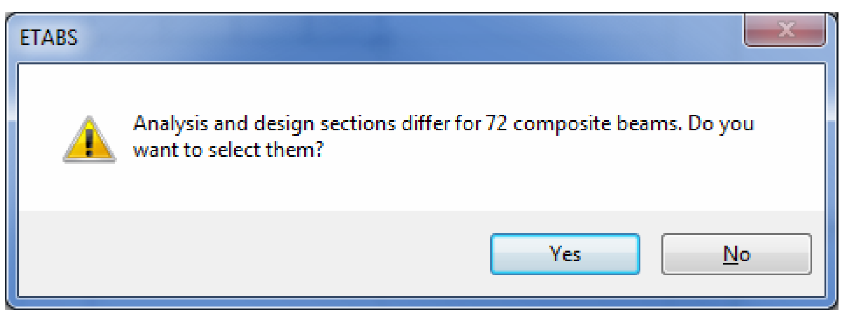


Figure 60
Analysis vs Design Section warning message for an incomplete design

The goal is to repeat the analysis (Step 10) and design (Step 12) process until the analysis and design sections are all the same. Note that when the building is reanalyzed (i.e., Step 10 is repeated), ETABS will use the current design sections (i.e., those selected in Step 12) as new analysis sections for the next analysis run. Thus, in the next analysis of this example, the composite beams will be analyzed using W12X19 analysis sections.

G. Right click on one of the composite beams in the 3D View shown in Figure 59. The Interactive Composite Beam Design and Review form shown in Figure 61 displays.

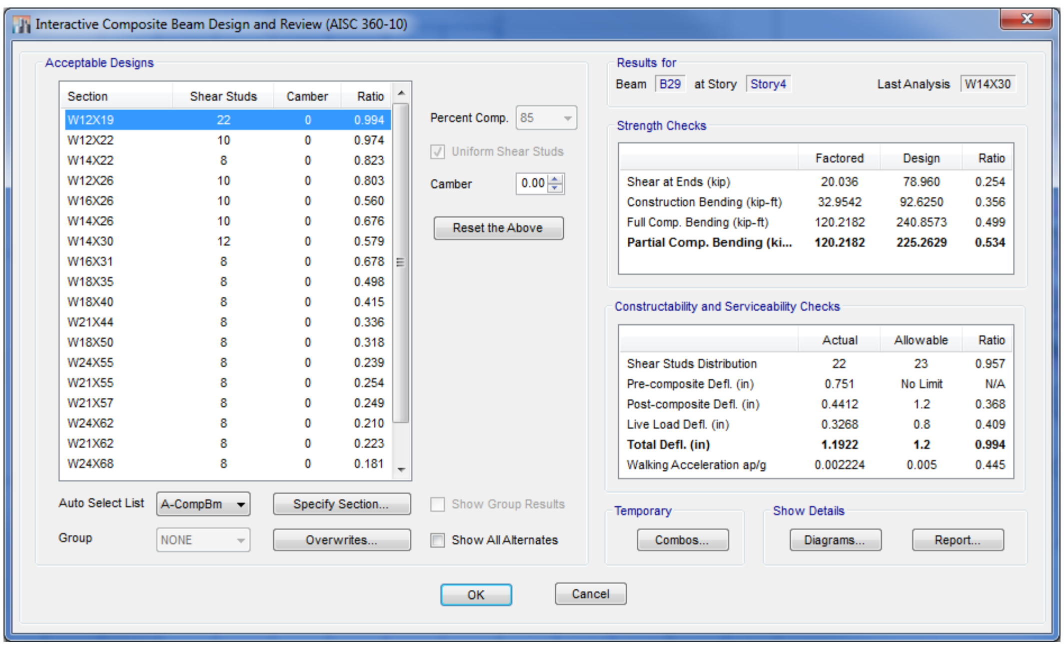
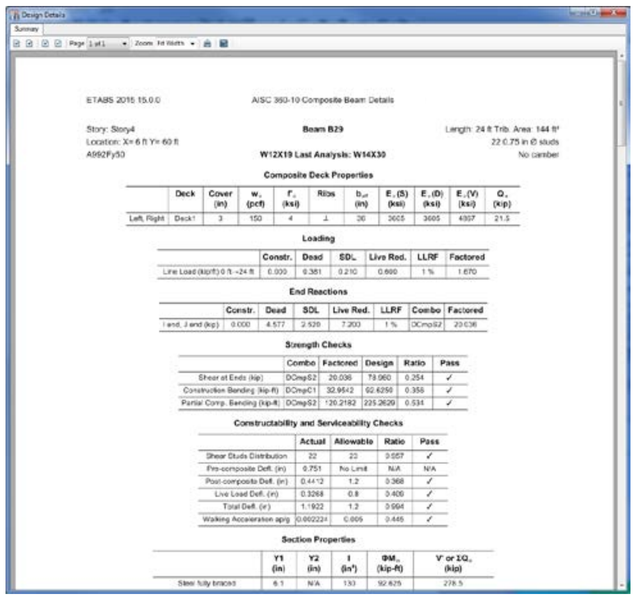


Figure 61
Interactive Composite Beam Design and Review form

Note that the current design section is highlighted as W12X19 and the last analysis section is reported as W14X30.

The Acceptable Designs list shows the beams in the A-CompBm auto select section list and their respective design ratios.

1. Click the **Report** button on the Interactive Composite Beam Design and Review form. The Composite Beam Design Report shown in Figure 62 displays. This report shows comprehensive design information about the beam. Review the information in this report. Then click the **[X]** in the upper right-hand corner of the form to close it.



ETABS 2015 15.0.0 AISC 360-10 Composite Beam Details

Story: Story4
Location: X= 6 ft Y= 60 ft
A992Fy50

Beam B29

W12X19 Last Analysis: W14X30

Length: 24 ft Trib. Area: 144 ft²
22 0.75 in Ø studs
No camber

Composite Deck Properties

	Deck	Cover (in)	w_c (pcf)	f'_c (ksi)	Ribs	b_{eff} (in)	E_c(S) (ksi)	E_c(D) (ksi)	E_c(V) (ksi)	Q_n (kip)
Left, Right	Deck1	3	150	4	⊥	36	3605	3605	4507	21.5

Loading

	Constr.	Dead	SDL	Live Red.	LLRF	Factored
Line Load (kip/ft) 0 ft - 24 ft	0.000	0.381	0.210	0.600	1 %	1.670

End Reactions

	Constr.	Dead	SDL	Live Red.	LLRF	Combo	Factored
I end, J end (kip)	0.000	4.577	2.520	7.200	1 %	DCmpS2	20.036

Strength Checks

	Combo	Factored	Design	Ratio	Pass
Shear at Ends (kip)	DCmpS2	20.036	78.960	0.254	✓
Construction Bending (kip-ft)	DCmpC1	32.9542	92.6250	0.356	✓
Partial Comp. Bending (kip-ft)	DCmpS2	120.2182	225.2629	0.534	✓

Constructability and Serviceability Checks

	Actual	Allowable	Ratio	Pass
Shear Studs Distribution	22	23	0.957	✓
Pre-composite Defl. (in)	0.751	No Limit	N/A	N/A
Post-composite Defl. (in)	0.4412	1.2	0.368	✓
Live Load Defl. (in)	0.3268	0.8	0.409	✓
Total Defl. (in)	1.1922	1.2	0.994	✓
Walking Acceleration ap/g	0.002224	0.005	0.445	✓

Section Properties

	Y1 (in)	Y2 (in)	I (in⁴)	ΦM_n (kip-ft)	V' or ΣQ_n (kip)
Steel fully braced	6.1	N/A	130	92.625	278.5

Figure 62 Composite Beam Design Report

2. Click the **Cancel** button to close the Interactive Composite Beam Design and Review form.

H. To rerun the analysis with the new analysis sections for the composite beams, click the **Analyze menu > Run Analysis** command or the **Run Analysis** button, ▶.

I. When the analysis is complete, click the **Design menu > Composite Beam Design > Start Design/Check** command to start the composite beam design process.

J. Click the **Design menu > Composite Beam Design > Verify Analysis vs Design Section** command. The message shown in Figure 63 should display, indicating that the analysis and design sections are the same for all composite beams. If you do not get this message, repeat Action Items H, I and J until you do get it, before proceeding to the next Action Item. Click the **OK** button.

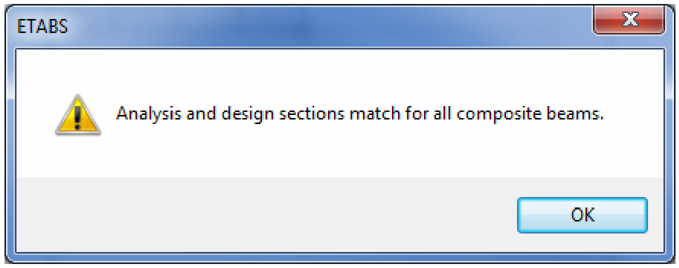


Figure 63
Analysis vs Design Section warning message for a complete design

K. Click the **Design menu > Composite Beam Design > Verify All Members Passed** command. The message shown in Figure 64 should appear, indicating that all composite beams passed the design check. Click the **OK** button to close the form.

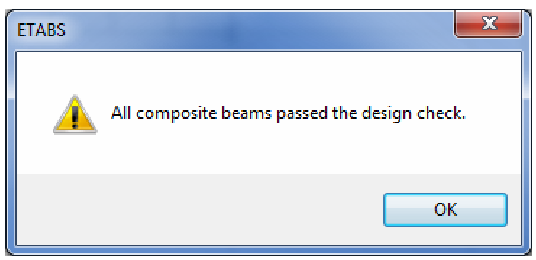


Figure 64
Verify All Members Passed warning message for a complete design

L. Click the **Select All** button, all, or click the **Select menu > Select > All** command, or press the Ctrl and A keys simultaneously on your keyboard to select all objects in the model.

M. Click the **Design menu > Composite Beam Design > Make Auto Select Section Null** command and click the **OK** button on the resulting message. This removes the auto select section list assignments from the composite beam members and replaces them with their current design sections.

N. Click the **Assign menu > Clear Display of Assigns** command. Also click the **Clear Selection** button, clr, to clear the selection.

O. Click the **File menu > Save** command, or the **Save** button, , to save your model. The composite beam design is now complete.

Step 13 Design the Steel Frame

In this Step, steel frame design is completed. Note that the analysis (Step 10) should be run before performing the following Action Items.

A. Click the **Select menu > Select > Properties > Frame Sections** command. The Select by Frame Property form shown in Figure 65 displays.

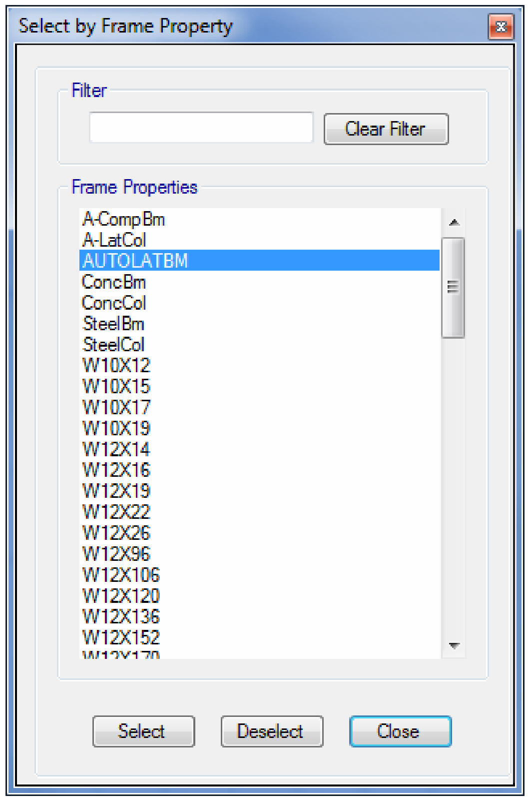


Figure 65
Select by Frame Property form

1. Highlight *AUTOLATBM* in the Frame Properties list and click the **Select** button. This selects all of the beams assigned the AUTOLATBM auto select section list.

2. Click the **Close** button to close the form.

B. Click the **Design menu > Steel Frame Design > Lateral Bracing** command. The Lateral Bracing form displays.

C. Select the *User Specified* option and click the **Specify Point Bracing** button. The Point Braces form shown in Figure 66 displays. This form is used to define the points where bracing occurs along the beams.

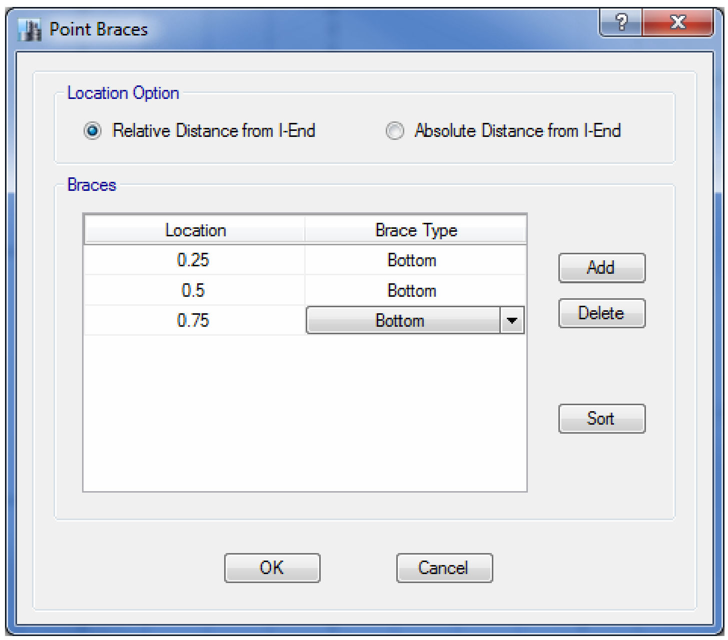


Figure 66
Point Braces form

1. Verify that the *Relative Distance from I-End* option is selected - this allows ratios to be used for specifying the bracing locations.

2. Click the **Add** button and type **0.25** in the Location edit box and select *Bottom* from the Brace Type drop-down list. For this model, bracing is being defined only for the bottom of the beam, as it is assumed that the deck braces the top.

3. Repeat step 2 twice more, entering **0.5** and **0.75** for the locations. We are bracing the beams at the quarter points.

4. Click the **OK** button to close the Point Braces form.

D. Click the **OK** button to close the Lateral Bracing form.

E. Click the **Design menu > Steel Frame Design > View/Revise Preferences** command. Select *AISC360-10* from the Design Code drop-down list on the Steel Frame Design Preferences form. Click the **OK** button to close the form.

F. In the Plan View, right click on the beam along grid line A between grid lines 1 and 2. The Beam Information form shown in Figure 67 displays. Review the information. Note that the design procedure for this beam is Steel Frame. Click the **Cancel** button to close the form.

G. Click on the title tab of the 3D View to make it active. This allows the design results to appear in the 3D View.

H. Click the **Design menu > Steel Frame Design > Start Design/Check** command or click the **Steel Frame Design** button, , to start the steel frame design process. The columns and the lateral beams that span between columns are designed.

I. When the initial design is complete, a form similar to that shown in Figure 68 displays.

   Similar to composite beam design (described in Step 12), in the initial analysis, the program used the median section by weight from the AUTOLATBM and A-LatCol auto select section lists for the analysis. The design sections chosen differ from the analysis sections used. The message in Figure 68 indicates that the analysis and design sections are different.

   1. Click the **No** button two times to close the different message forms.

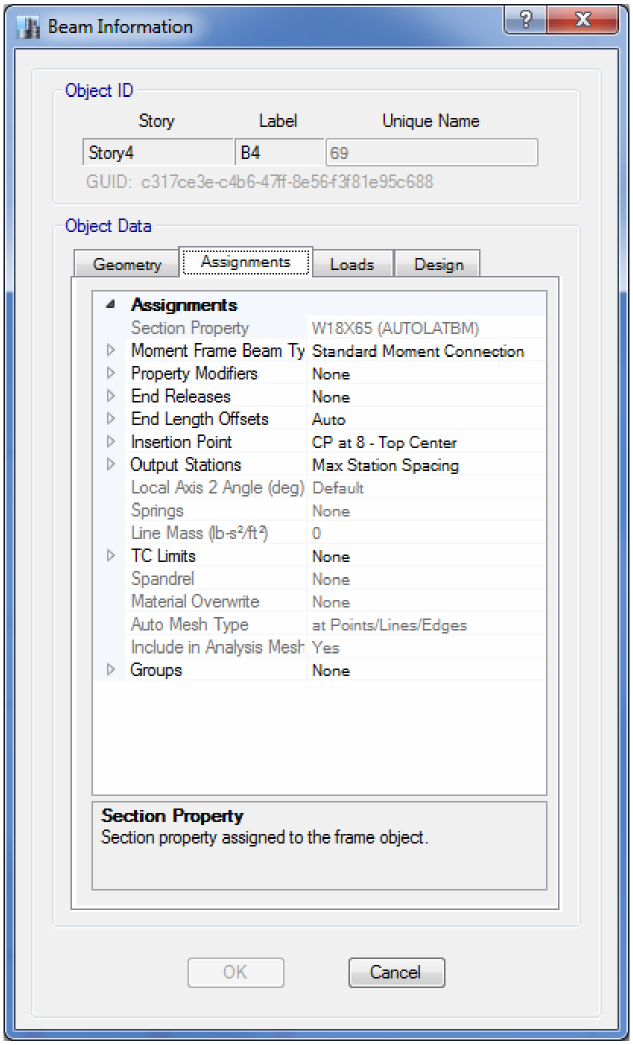


Figure 67
Beam Information form

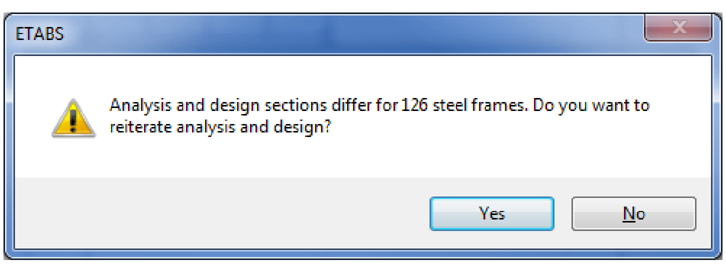


Figure 68
Analysis vs Design Section warning message for an incomplete design

J. Click on the title tab of the Plan View to activate the view.

K. Click the **Design menu > Steel Frame Design > Display Design Info** command. The Display Design Results form displays.

1. Make sure that the *Design Output* option is selected and that *P-M Ratio Colors & Values* is selected in the Design Output drop-down list. Then click the **OK** button.

Results are displayed in the Plan View and the model appears as shown in Figure 69.

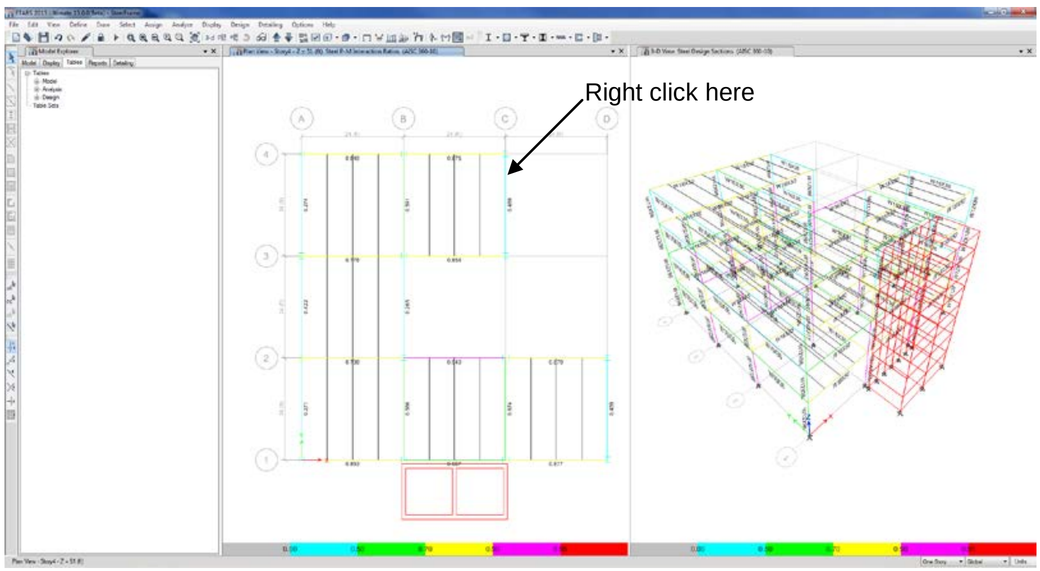
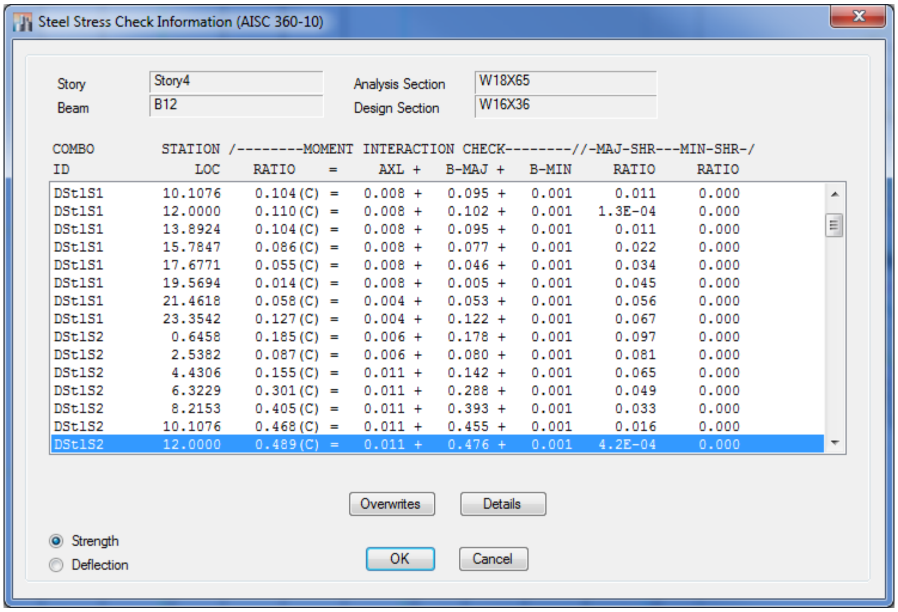


Figure 69
Model after the initial steel frame design



Steel Stress Check Information (AISC 360-10)

Story: Story4
Beam: B12
Analysis Section: W18X65
Design Section: W16X36

COMBO ID	STATION LOC	RATIO	=	AXL	+	B-MAJ	+	B-MIN	MAJ-SHR RATIO	MIN-SHR RATIO
DStlS1	10.1076	0.104(C)	=	0.008	+	0.095	+	0.001	0.011	0.000
DStlS1	12.0000	0.110(C)	=	0.008	+	0.102	+	0.001	1.3E-04	0.000
DStlS1	13.8924	0.104(C)	=	0.008	+	0.095	+	0.001	0.011	0.000
DStlS1	15.7847	0.086(C)	=	0.008	+	0.077	+	0.001	0.022	0.000
DStlS1	17.6771	0.055(C)	=	0.008	+	0.046	+	0.001	0.034	0.000
DStlS1	19.5694	0.014(C)	=	0.008	+	0.005	+	0.001	0.045	0.000
DStlS1	21.4618	0.058(C)	=	0.004	+	0.053	+	0.001	0.056	0.000
DStlS1	23.3542	0.127(C)	=	0.004	+	0.122	+	0.001	0.067	0.000
DStlS2	0.6458	0.185(C)	=	0.006	+	0.178	+	0.001	0.097	0.000
DStlS2	2.5382	0.087(C)	=	0.006	+	0.080	+	0.001	0.081	0.000
DStlS2	4.4306	0.155(C)	=	0.011	+	0.142	+	0.001	0.065	0.000
DStlS2	6.3229	0.301(C)	=	0.011	+	0.288	+	0.001	0.049	0.000
DStlS2	8.2153	0.405(C)	=	0.011	+	0.393	+	0.001	0.033	0.000
DStlS2	10.1076	0.468(C)	=	0.011	+	0.455	+	0.001	0.016	0.000
DStlS2	12.0000	0.489(C)	=	0.011	+	0.476	+	0.001	4.2E-04	0.000

Overwrites | Details
Strength
Deflection
OK | Cancel

Figure 70
Steel Stress Check Information form

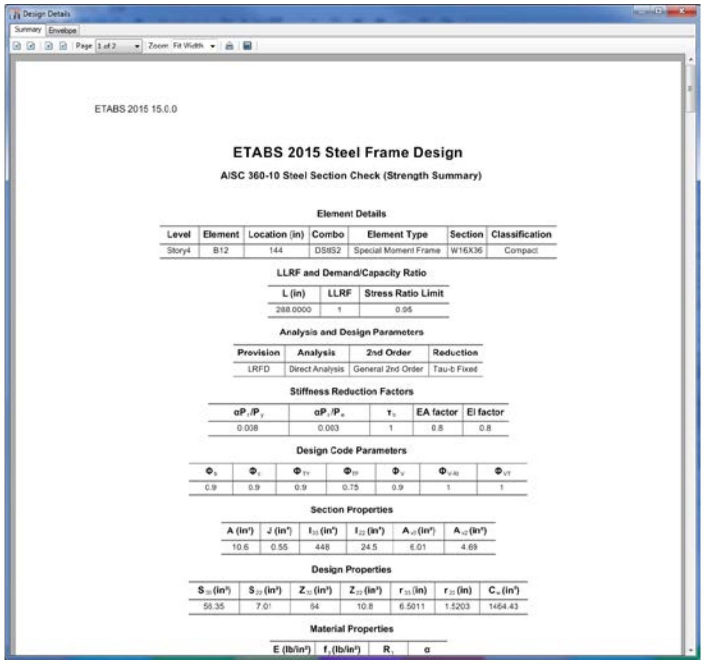
L. In the Plan View, right click on the beam along grid line C between grid lines 3 and 4 as indicated in Figure 69. The Steel Stress Check Information form shown in Figure 70 displays. Note that the reported analysis and design sections are different.

The main body of the form lists the design stress ratios obtained at various stations along the beam for each combination. Note that the program automatically created code-specific design combinations for this steel frame design. (It also did this for the composite design.)

Click the **Details** button on the Steel Stress Check Information form. The Report Viewer shown in Figure 71 displays with detailed design information about the selected member. Note that you can print this information using the Print Report button on the menu bar.

Click the **[X]** in the upper right-hand corner of the Report Viewer to close it.

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



Design Details

Summary | Envelope

Page 1 of 2 | Zoom Fit Width

ETABS 2015 15.0.0

ETABS 2015 Steel Frame Design

AISC 360-10 Steel Section Check (Strength Summary)

Element Details

Level	Element	Location (in)	Combo	Element Type	Section	Classification
Story4	B12	144	DStlS2	Special Moment Frame	W16X36	Compact

LLRF and Demand/Capacity Ratio

L (in)	LLRF	Stress Ratio Limit
288.0000	1	0.95

Analysis and Design Parameters

Provision	Analysis	2nd Order	Reduction
LRFD	Direct Analysis	General 2nd Order	Tau-b Fixed

Stiffness Reduction Factors

$\alpha P_r/P_y$	$\alpha P_r/P_e$	τ_b	EA factor	EI factor
0.008	0.003	1	0.8	0.8

Design Code Parameters

Φ_b	Φ_c	Φ_{TY}	Φ_{TF}	Φ_V	Φ_{V-RI}	Φ_{VT}
0.9	0.9	0.9	0.75	0.9	1	1

Section Properties

A (in²)	J (in⁴)	I_{33} (in⁴)	I_{22} (in⁴)	A_{v3} (in²)	A_{v2} (in²)
10.6	0.55	448	24.5	6.01	4.69

Design Properties

S_{33} (in³)	S_{22} (in³)	Z_{33} (in³)	Z_{22} (in³)	r_{33} (in)	r_{22} (in)	C_w (in⁶)
58.35	7.01	64	10.8	6.5011	1.5203	1464.43

Material Properties

E (lb/in²)	f_y (lb/in²)	R_y	α

Figure 71
Report Viewer

M. Click the **Design menu** > **Steel Frame Design** > **Select Design Combinations** command. The Design Load Combinations Selection form shown in Figure 72 displays.

The Design Combinations list identifies the ten default steel frame strength design combinations created by the program. Click on *DStlS6* to highlight it and then click the **Show** button. The Load Combination Data form shown in Figure 73 displays, showing how the program defined design combination DStlS6.

1. Click the **OK** button in the Load Combination Data form to close it. If desired, review other design combination definitions and then click the **OK** button to close the Data form.

2. Click the **Cancel** button in the Design Load Combinations Selection form to close it without accepting any changes that may have inadvertently been made.

N. Click on the title tab of the Plan View to activate the view.

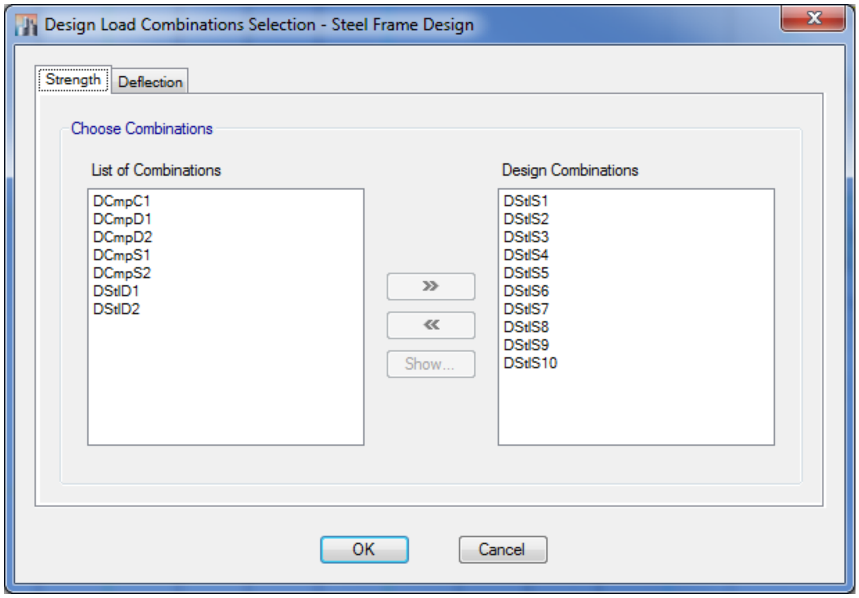


Figure 72
Design Load Combinations Selection form

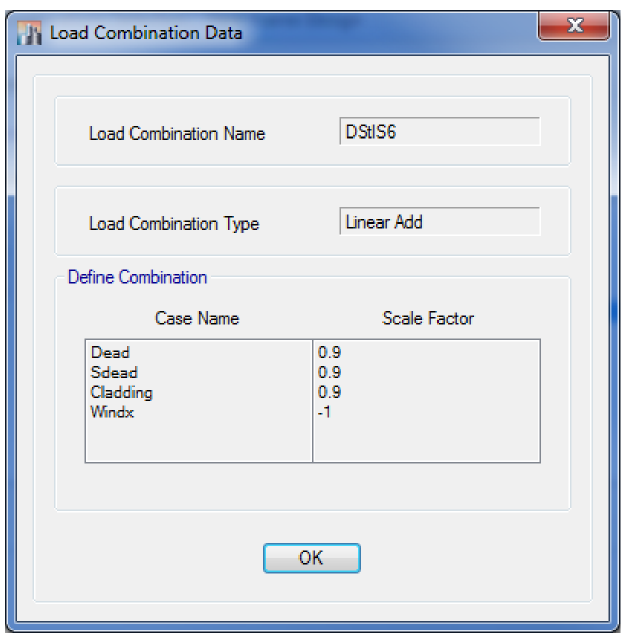


Figure 73
Load Combination Data form

O. Click the **Display menu > Undeformed Shape** command or the **Show Undeformed Shape** button, , to clear the display of the stress ratios.

P. Click on the title tab of the 3D View to activate the 3D view.

Q. To rerun the analysis with the new analysis sections for the steel beams, click the **Analyze menu > Run Analysis** command or the **Run Analysis** button, .

R. When the analysis is complete, a deformed shape will display. Click the **Design menu > Steel Frame Design > Start Design/Check of Structure** command or click the **Steel Frame Design** button, , to start the steel frame design process.

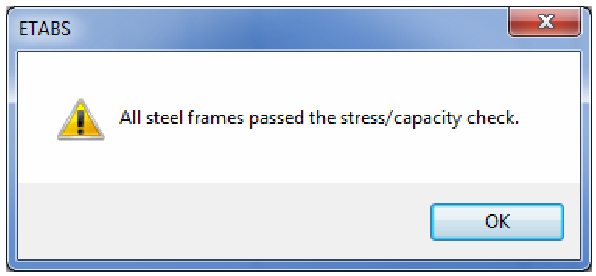
   When the design is complete, a message will display indicating how many design sections are different from the analysis sections. Click the **Yes** button to reiterate the analysis and design and repeat this process until the analysis and design sections are the same, which is indicated when no message displays at the end of the design (and the windows "wait cursor" has disappeared). This may take several iterations for this example.

S. When the analysis and design sections are the same, click the **Select All** button, , or click the **Select menu > Select > All** command, or press the Ctrl and A keys simultaneously on your keyboard to select all objects in the model.

T. Click the **Design menu > Steel Frame Design > Make Auto Select Section Null** command and click **OK** for the resulting message. This removes the auto select section list assignments from the steel frame members and replaces them with their current design sections.

U. Click the **Design menu > Steel Frame Design > Verify All Members Passed** command. A form similar to that shown in Figure 74 should appear indicating that all members passed.

Figure 74
Verify All Members Passed warning message for a complete design



Note that members not passing at this stage is an indication of inadequate sections in the auto select list. The program would have used the largest section in the auto select list for both analysis and design, finding it inadequate. In that case, either add more sections to the auto select list or assign a larger section to the members that did not pass and continue with the design process. Click the **OK** button to close the form.

V. Click the **Clear Section** button, , to clear the selection. Click the **File menu** > **Save** command, or the **Save** button, , to save your model. The steel frame design tutorial is now complete.