# **COMPUTERS & STRUCTURES, INC.**

## STRUCTURAL AND EARTHQUAKE ENGINEERING SOFTWARE

Integrated Building Design Software

# Introductory Tutorial - Parts I & II

REALLY, THERE

(R)

u Pesi

TECHNOLOGIC FOR A BETTA



# Introductory Tutorial Parts I & II

# **ETABS® 2016**

## **Integrated Building Design Software**

ISO ETA122815M3 Rev. 0 Proudly developed in the United States of America

December 2015

## Copyright

Copyright © Computers & Structures, Inc., 1978-2015 All rights reserved.

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

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

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

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

Computers & Structures, Inc. <u>http://www.csiamerica.com/</u>

<u>info@csiamerica.com</u> (for general information) <u>support@csiamerica.com</u> (for technical support)

## DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND TESTING OF THIS SOFTWARE. HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THIS PRODUCT.

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

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

## Contents

# **Introductory Tutorial**

#### Part I - Steel Building Example

The Proj	ect	2
Step 1	Begin a New Model Define an Auto Select Section	2
	LISt	1
Step 2	Add Frame Objects	12
	Set Up to Add Objects to Multiple	
	Stories Simultaneously	12
	Draw Column Objects	13
	Save the Model	18
	Draw the Lateral Force-Resisting	
	Beam Objects	18
	Draw the Secondary (Infill) Beam	
	Objects	20
Step 3	Add Shell Objects	23
	Draw the Floor Shell Objects	23
	Add Exterior Cladding for Wind	
	Load Application	26
	Draw the Cladding	26

Step 4	Add a Wall Stack	30
Step 5	Define Static Load Patterns	32
Step 6	Assign Gravity Loads	36
Step 7	Define a Developed Elevation	40
Step 8	Assign Wind Loads	43
Step 9	Review Tabular Display of Input Data	48
Step 10	Run the Analysis	51
Step 11	Graphically Review the Analysis Results	52
Step 12	Design the Composite Beams	56
Step 13	Design the Steel Frame	63

### Part II - Concrete Building Example

The Proje	ect	74
Step 1	Begin a New Model	74
Step 2	Add Floor Openings Set Up to Add Objects to Multiple	81
	Stories Simultaneously	81
	Draw Shell Objects	81
	Save the Model	84
Step 3	Add Walls	84
Step 4	Define Static Load Patterns	86
Step 5	Review Diaphragms	89
Step 6	Review the Load Cases	92
Step 7	Run the Analysis	94
Step 8	Display the Results	96
Step 9	Design the Concrete Frames	99
Step 10	Design the Shear Walls	104

## 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.



### 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	0
Model Initialization form	Osc Schurgs Hond a moder ha      Use Built-in Settings With:     Display Units     Steel Section Database     Steel Design Code     Concrete Design Code     OK	U.S. Customary

- 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 1. 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.

New Model Quick Templates			? ×
Grid Dimensions (Plan)		Story Dimensions	
O Uniform Grid Spacing		Simple Story Data	
Number of Grid Lines in X Direction	4	Number of Stories	4
Number of Grid Lines in Y Direction	4	Typical Story Height	12 ft
Spacing of Grids in X Direction	24 1	t Bottom Story Height	þ5 ft
Spacing of Grids in Y Direction	24 1	t	
Specify Grid Labeling Options	Grid Labels		
Custom Grid Spacing		Custom Story Data	
Specify Data for Grid Lines	Edit Grid Data	Specify Custom Story Data	Edit Story Data
Add Structural Objects			
Blank     Grid Only     Steel Deck	Image: Base of the second s	Flat Slab with Perimeter Beams	Waffle Slab
	ОК	Cancel	

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

Objects Present in View         I Joint Objects         I Invisible         Columns         Beams         Braces         I Al Null Frames         I Al Null Frames         Openings         I Al Null Stacks         I Al Null Stacks         I Links	Special Effects Object Shirink Object Fill Object Fill Object Edge Extrude Frames Extrude Shells Other Special Items Joint Restraints and Springs Diaphragm Extent Connections Story Labels Dimension Lines Acchetectural Plan Layers Horizon Stab Internal Ribs
<ul> <li>Peams</li> <li>Braces</li> <li>Al Null Frames</li> <li>Roors</li> <li>Walls</li> <li>Openings</li> <li>All Null Shells</li> <li>Wall Stacks</li> <li>Uniks</li> </ul>	Exrude Frames Extrude Shells Other Special Items Joint Restraints a Diaphragm Exter Connections J Story Labels Dimension Lines Ambracetural Plan Honzon Shorn Shell Analy Slab Internal Ribi



4

- 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**  $\checkmark$  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.

	Quick Cartesian Grids	? 🗙
<b>Figure 6</b> Quick Cartesian Grids form	Grid Dimensions (Plan) Number of Grid Lines in X Direction Number of Grid Lines in Y Direction Spacing of Grids in X Direction Spacing of Grids in Y Direction X Grid Labeling First Grid Label Labeling Direction	4 4 24 1 24 1 1 1 1 1 1 1 1 1 1 1 1 1
	Y Gird Labeling First Gird Label Labeling Direction	1 Bottom to Top

- 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.



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.

Frame Property Shape Type	Const The Property	? <mark>×</mark>
Shape Type	Section Shape	I/Wide Flange 🔹
Frequently Used Shape Type	25	
Concrete		Steel
		I/Wide Flange Section
Special		Steel Composite
Section Designer	rismatic	
	ОК	Cancel

#### 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 I 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.



- 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 inder 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.

ſ	III Frame Section Property Data		? ×
<b>Figure 10</b> Frame Section Property Data form	General Data Property Name AL Auto Select Design Type Se Notes Shape Section Shape Au Section Property Source Source: User Defined Choose Sections in Auto Select List Available Sections Type Al Fitter SteelEim SteelCol SteelCol SteelCol SteelCol	JOLATEM           eel           Modfy/Show Notes           Addo Select              Addo Select           Addo Select           C           Addo Select           Show           W156277           W15429           Show           W15429           W15429 <th>Liet</th>	Liet
	Starting Section Starting Section in Auto Select List	Median Section by Area	Modify
		OK Cancel	

- 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 Modi-fy/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.

		Story	Height ft	Elevation ft	Master Story	Similar To	Splice Story	Splice Height ft
	•	Story4	12	51	Yes	None	No	0
		Story3	12	39	No	Story4	No	0
		Story2	12	27	No	Story4	No	0
		Story1	15	15	No	Story4	No	0
		Base		0				
1								
	Note: Rig	ht Click on Grid for Opt	ions					

### **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.

	Pro	perties of Object		×
2		Property	A-LatCol	-
es of		Moment Releases	Continuous	
orm		Angle, deg	90	
nns		Plan Offset X, in	0	
		Plan Offset Y, in	0	
		Cardinal Point	5 (Middle Center)	
		Draw Object Using	Grids	

#### 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**.





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, **A**, 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.

77/05	ID	CTODY	TOWER
TYPE	ID	STORY	TOWER
Joint	4	Story4	T1
Column	C4	Story4	T1
	ОК	Can	cel

- 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.

	Frame Assignment - Local Axes
<b>Figure 15</b> Frame Assignment - Local Axis form	Define Orientation             General Orientation - Applies to All Frame Objects             Angle              Rotate by Angle             Ø Orient with Grid System - Applies to Vertical Frame Objects Only         Grid System             Grid System             Grame object major direction is X             Ø Frame object major direction is X             Grame object major direction is Radial             Grame object major direction is Tangential



Figure 14 Selection List 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 **I**. 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, S 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.

Pro	perties of Object		x
	Type of Line	Frame	٦
	Property	AUTOLATBM	,
	Moment Releases	Continuous	
	Plan Offset Normal, in	0	
	Draw Object Using	Grids	

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 17** Properties of Object form for frame objects





- 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, **A**, 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.

Pro	operties of Object		
	Property	A-CompBm	-
	Moment Releases	Pinned	
	Spacing	No. of Beams	
	No. of Beams	3.	
	Approx. Orientation	Parallel to Y or R	

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 >

Figure 19 Properties of Object for beams

**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, **A**, to change the program from Draw mode to Select mode.
- E. Click the **Select using Intersecting Line** button, **N**, 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.





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, **b**, 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, <u>un</u>check 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.

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 22 Set View Options

form

Figure 23 Properties of Object form for shells

Properties of Object		<b>×</b>
Property	Deck1	4
Local Axis	0	
Edge Drawing Type	Straight Line	
Drawing Control Type	None <space bar=""></space>	

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 Se-

**lect Object** button,  $\land$ , 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.



- 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.

	General Data		
	Property Name	Deck1	
	Туре	Filed	- ()
	Slab Material	4000Psi	▼
	Deck Material	A992Fy50	▼
Jure 20	Modeling Type	Membrane	<b>_</b>
a form	Modifiers (Currently Default)	Modify/Show	
a Iom	Display Color	Change	
	Property Notes	Modify/Show	
	Property Data		
	Slab Depth, tc	3	in
	Rib Depth, hr	3	in
	Rib Width Top, wrt	7	in
	Rib Width Bottom, wrb	5	in
	Rib Spacing, sr	12	in
	Deck Shear Thickness	0.035	in
	Deck Unit Weight	2.3	lb/ft2
	Shear Stud Diameter	0.75	in
	Shear Stud Height, hs	6	in
	Shear Stud Tensile Strength, Fu	65000	lb/in2

### 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.

<b>Figure 26</b> Cladding Options form	Cladding Options  Use Roors Use Beams Use Columns Use Selected Objects only
	OK Cancel

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  $el_{V}^{e}$  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.



- 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, **I**, to save your model.
- I. Click on the Plan View tab to make it active.

Your model now appears as shown in 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.





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

Angle, deg	180	
Top Story	Story4	
Bottom Story	Story1	

- 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.

- 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 **Clad-ding**. 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.

### Introductory Tutorial

Direction and Eccentricity	_	Seismic Coefficients		
🗖 X Dir	Y Dir	Ss and S1 from USGS Database - by Latitude/Longitude		
X Dir + Eccentricity	Y Dir + Eccentricity	Ss and S1 from USGS Database - by Zip Code		
X Dir - Eccentricity	Y Dir - Eccentricity	Ss and S1 - User Defined		
% Eccentricity		Site Latitude (degrees)	?	
Override Eccentricities	Override	Site Longitude (degrees)	?	
ime Period		Site Zip Code (5-Digits)	?	
Approximate Ct (ft), x	=	0.2 Sec Spectral Accel, Ss	2.29	
Program Calculated Ct (ft), x	= 0.028; 0.8 -	1 Sec Spectral Accel, S1	0.869	
O User Defined ⊤ =	sec	Long-Period Transition Period	8 sec	
itory Range		Site Class	B 🔻	
Top Story for Seismic Loads	Story4 👻	Site Coefficient, Fa	1	
Bottom Story for Seismic Loads	Base 💌	Site Coefficient, Fv	1	
actors		Calculated Coefficients		
Paranana Madification P	8	SDS = (2/3) * Fa * Ss	1.5267	
System Overstrength Omena	3	SD1 = (2/3) * Fv * S1	0.5793	
Deflection Amplification Cd	5.5			
Deliection Anplincation, Cu	3.5			

#### 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.

Exposure and Pressure Coefficients	Wind Coefficients	
Exposure from Extents of Bigid Diaphragms	Wind Speed (mph)	100
Exposure from Frame and Shell Objects	Exposure Type	B 🗸
Include Frame Objects (Open Structure)	Importance Factor	1
Wind Supervise Promotion	Topographical Factor, Kzt	1
Wind Direction and Exposure Width Modify/Show	Gust Factor	0.85
Windward Coefficient Co	Directionality Factor, Kd	0.85
	Solid / Gross Area Ratio	
Case (ASCE 7-10 Fig. 27.4-8)	Exposure Height	
e1 Ratio (ASCE 7-10 Fig. 27.4-8)	Top Story	Story4 👻
e2 Ratio (ASCE 7-10 Fig. 27.4-8)	Bottom Story	Base 🔻
	Parapet Height	ft
ОК	Cancel	

#### 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.

Loads	Type	Self Weight Multiplier	Auto Lateral Load	Click To: Add New Load
Windx Dead	Wind Dead Beducible Live	▼ 0 1 0	ASCE 7-10	Modify Load
Sdead Cladding Eqy	Super Dead Super Dead Seismic	0000	ASCE 7-10	Delete Load
Windx	Wind	0	ASCE 7-10	

#### 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, 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, dr<sup>1</sup>, 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.

<b>Figure 35</b> Shell Load Assignment - Uniform form	Shell Load Assignment - Uniform Load Pattern Name Uniform Load Load 35 b/ft <sup>2</sup> Direction Gravity	Sdead  Options  Add to Existing Loads  Replace Existing Loads  Delete Existing Loads
form	Direction Gravity	Delete Existing Loads  Delete Existing Loads  Delete Existing Loads  Delete Existing Loads

#### 36 Step 6 Assign Gravity Loads

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.

	Shell Load Assignment - Uniform	
Figure 36 Shell Load As- signment - Uniform form	Uniform Load     Options       Load     100     lb/ft²       Direction     Gravity           Options         Options         Image: Direction     Image: Direction Content of the provided market o	
	OK Close Apply	

- 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 <u>not</u> 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 <u>not</u> 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.
- 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 dropdown list.

re 37 e Load Type and Direction © Forces Direction of Load Application Trapezoidal Loads 1. 2. 3. 4. Distance 0 0.25 0.75 1 Load 0 0 0 0 k		Load Pa	attern Name	Cla	dding	•	
Trapezoidal Loads         1.         2.         3.         4.           Distance         0         0.25         0.75         1           Load         0         0         0         0         ki	37 Load ment	<ul> <li>Load Type a</li> <li>Force:</li> <li>Direction of</li> </ul>	and Direction s © of Load Application	Moments Gravity	Optio	ons Add to Existing Loa Replace Existing Lo Delete Existing Loa	ids pads ids
Load 0 0 0 k	Juleu	- Trapezoidal Distance	Loads 1. 0	2.	3.	4.	
		Load	0	0	0	0	kip/ft
Relative Distance from End-I			Relative	Distance from End-I	Absolu	te Distance from End	9-1

- 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 **Plan** and select *Story4* from the Select Plan View form. Click the **OK** button.
- M. Click the **File menu > Save** command, or the **Save** button, **B**, 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.

New Developed Elevation Name RIGHT
RIGHT
Existing Developed Elevation Names
UK Cancel



- 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 counterclock-wise 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 Plan and select Story4 from the Select Plan View form. Click the OK button.
- F. Click the **File menu > Save** command, or the **Save** button, **b**, 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

form

Set View Options



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 <u>negative</u> global X direction. Note the global axes that are located at the origin of the model.



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 <u>positive</u> 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,  $e|_{V}^{e}$ , 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 

 Number
 Numer
 Number
 Number

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

	Shell Load Assignment - Wind Pressure Coe	fficient 🗾
Figure 45 Shell Load As- signment - Wind Pressure Coefficient form	Wind Load Pattern Name Wind pressure Coefficient, Cp -0.8 Windward (varies) Leeward or Sides (constant) OK	Windx

- 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 elevation 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.





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, 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 H Model node located under the Tables branch to expand the tree. The Tables tab should now look similar to that shown in Figure 47.

Model Display Tables	Reports Detailing	
- Tables		
ian <mark>Model</mark>		
Project Setting	s	
	ut	
Definitions		
⊕ Assignments		
Design		
Table Sets		



48

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.



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					
14	4   1 of 8	🕨 🕨 Reload	Apply			
	Story	Label	Unique Name	Load Pattern	Direction	Load Ib/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
						4

#### Figure 49

Table for Uniform Shell Loads

Step 9 Review Tabular Display of Input Data

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.
- 50 Step 9 Review Tabular Display of Input Data

### 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.

S Chead Live Live Sdead Cladding Eqv Windx Linear Static Cladding Linear Static Cladding Clader Static Cladding Clader Static Cladding Clader Static Cladding Clader Static Clader Static Clader Static Show Load Case Tri Clader Static Clader Static Clader Static Clader Static Clader Static Clader Static Clader Static Show Load Case Tri Clader Static Show Load Case Tri Clader Static Show Load Case Tri Clader Static Show Load Case Tri Show Load		Load Case Name	Load Case Type		Add New Case
Live     Linear Static       Sdead     Linear Static       Cladding     Linear Static       Eqy     Linear Static       Windx     Linear Static   Show Load Case Tr	Dead		Linear Static		Add Copy of Case
Sdead     Linear Static       Cladding     Linear Static       Eqy     Linear Static       Windx     Linear Static	Live		Linear Static		Modify/Show Case
Cladding     Linear Static       Eqy     Linear Static       Windx     Linear Static	Sdead		Linear Static		Delete Case
Eqy Linear Static Show Load Case Tri Windx Linear Static	Claddin	)	Linear Static	*	
Windx Linear Static	Eqy		Linear Static		Show Load Case Tre
	Windx		Linear Static	•	
					OK
ОК					Cancel

- 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,  $\blacktriangleright$ .

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,  $el_{V}^{el}$ , 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, ₩7, or the **Display** menu > Force/Stress Diagrams > Frame... Forces command to access the Member Force Diagram form shown in Figure 52.

	Load Case     Load Combination     Modal Case
	Dead
	Component
	C Axial Force C Torsion C Inplane Shear
jure 52	Shear 2-2 Moment 2-2 Inplane Moment
ember Force	Shear 3-3    Moment 3-3
agram torm	Scaling
	<ul> <li>Automatic</li> </ul>
	User Defined     Scale Factor
	Display Options
	Fill Diagram
	Show Values at Controlling Stations on Diagram
	Include
	Frames Piers Spandrels
	Links

- 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 Dia-grams 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.

#### Introductory Tutorial



#### 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.

#### 54 Step 11 Graphically Review the Analysis Results

- 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,  $\square$ , 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.



Step 11 Graphically Review the Analysis Results

Figure 56 Beam Information form

### 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.

Story4         IE24         149           GUID         bb9fc751-351b-4737-3804-68b577Bcaada           Geometry         Loado         Design           Section Froperly         Loado         Design           > Properly Modifiers         None         None           > End Pelaesee         T, M22; M33         None           > End Pelaesee         T, M22; M33         None           > Insetion Formit         CP at 8-1-top Center         Output: Stations           None         None         None           Load & Settion Spacing         Load Settion Spacing         None           Local Area:         Anne         None           > Tot Lintis         None         None           Matchaid Overwite         None         None           Auto Methy Type         at PromsLinearEdges		·		bel Unique Name		
GUD: bb/#6751-391b-47974804-686-6778caada dect Data Geenetry: Assignment: Loads Design Assignment: Loads Design Assignment: VI4/30 (A-CompBn) > Popperfy Model Mone > End Iongh Offsets Auto > End Iongh Offsets Auto > Dend Satirs None Line Maa (or 5/ht) > TC Lints None Spanda Overwite Auto Mah Type at ProfessLines/Edges	Story4	E	324	149		
ect Data Geomety Assignments Loads Design Assignments Sector Property W14/30 (A-CompBin) Property Modifiers None End Relaxes T, Nu22, M33 End Length Offsets Auro Destination C CP at 8-Top Center Octype Zarging (Roy Deshut) Destination Relations Spacing Load Ava 2 Angle (Roy Deshut) Load Ava 2 Angle (Roy Deshut) Load Ava 2 Angle (Roy Deshut) Destination Relations Spacing Load Ava 2 Angle (Roy Deshut) Destination Relations Space Relations Space Relations None Material Overwite Avone State Edges	GUID: 665	f6751-391b-479	97-880	4-68b67	78caada	
Construct         Loads         Design           Subtime Privative         VMLV30 (A.CompBin)           Property ModRers         None           Serial Privative         None           Property ModRers         None           Serial Privative         TAU2 M33           End Langth Offsets         Auro           Insident Privative         CArage (deg)           Incold And Zange (deg)         Descing           Local And Zange (deg)         None           Line Mass (brith)         None           Match Uorenty         None           Match Uorenty         None           Match Uorenty         None	oct Data					
Generativy         Lassignments         Loads         Design           4 Assignments         Sector Property         V14/30 (A.CompBin)           5 Extor Property         V14/30 (A.CompBin)           9 Popetry Modifies         None           9 End Preases         T, M22, M33           9 End Length Offsets         Auto           9 Outor Assignment         CP at 81-Top Center           9 Outor Assignment         None           Lone Mass (no 4/ht)         0           1 met Mass (no 4/ht)         None           Line Mass (no 4/ht)         None           Auto Mech Type         None           Mate Build Overwitte         None           Auto Mech Type         TorinsLines/Edges	our Data					
Assignments     Secton Provemant     Secton Provemant     Vir(X2) (A-Comp8m)     Property Modifiers     None     End Pieases     T. M22: M33     End Length Offsets     Auto     End Length Offsets     Auto     Insetion Prov     Cop at a 1-top Center     Output Stations     None     Lone Mass (be*th)     None     Matted Overwite	Geometry	Assignmen	ts	Loads	Design	
Section Property W14/30 (A-CompBin) > Property Modifiers None > End Releases T. Nu22; N33 > End Isengh Offsets Auro > Inselion Park CP at 8 - Top Center > Inselion Park CP at 8 - Top Center > Output Station Min: Station Spacing Local Aria 2 Aragle (deg) Default (de 5-RF) > To Linits None Material Overwitte None Auto MenTi Type at ForeitsLines/Edges	⊿ Assig	nments				
▷         Poperty Modifier         None           ▷         End Relaxes         T, M22, M33           ▷         End Longht Offsets         Auto           ▷         Indentin Prof.         CP at 8. Top Center           ▷         Output Stations         Max Station Spacing           Local Areal Z-Angle (deg)         Default         Springs           Ime Mass (b off ft)         0         None           ≥         TCL Inits         None           Auto Mech Type         ar ForeNault-Edges	Sectio	n Property	W	14X30 (4	-CompBm)	
> End Releases         T, M22, M33           > End Length Chef         Auto           > Insection Point         CP at 8.1 rop Center           > Option Mark         Mark Strain Spacing           > Option Mark Strain Spacing         Default           Sompa         None           > To Limits         None           Spanded Verwrite         None           Auto Ment Type         a FonisLines/Edges	Proper	ty Modifiers	No	ne		
<ul> <li>▷ End Langth Offsets Auro</li> <li>▷ Insetion Ports</li> <li>CP at 8 - Top Center</li> <li>▷ Output Stations</li> <li>Max Station Spacing</li> <li>Local Ava 2 Angle (deg)</li> <li>▷ Enture Mass (b eV/H)</li> <li>▷ TC Lints</li> <li>None</li> <li>∑orints</li> <li>None</li> <li>More Constructions</li> <li>Clints</li> <li>None</li> <li>Material Overwitte</li> <li>None</li> <li>Auto Mech Type</li> <li>at FortsUnars/Edges</li> </ul>	End R	eleases	T;	M22; M3	33	
▷ Insettion Point CP at 8 - Top Center O chrpt 32tions Max Station Spacing Local Avia 2 Angle (deg) Default Springa None Line Mass (be3/ft?) 0 T CLimts None Spandrel None Material Overwritt None Auto Meih Type at Points/Lines/Edges	End L	ength Offsets	A	ito		
▷ Output Stations Max Station Spacing Local /wis 2 Angle (deg) Dight (deg) Springs None Line Mass (bis9/R) 0           > TC Limts None Spandrel         None           Material Overwrdt         None Auto Mesh Type         at Points/Lines/Edges	Inserti	on Point	CI	at 8 - T	op Center	
Local Axis 2 Angle (deg) Default Springs Line Mass (be \$AR?) 0 > TC Lints Default Material Overwrite Auto Meah Type at Points/Lines/Edges	Output	Output Stations			Max Station Spacing	
Springs None Line Mass (b-s∿ft?) 0 ▷ TC Limits None Spandrel None Auto Mesh Type at Points/Lines/Edges	Local	Axis 2 Angle (de	ig) Di	efault		
Line Mass (j0.57π?) 0     TC Limits None     Spandrel None     Material Overwrite None     Auto Mesh Type at Points/Lines/Edges	Spring	5	No	ne		
Spandrel None Material Overwrite None Auto Mesh Type at Points/Lines/Edges	Line N	TC Limits				
Material Overwrite None Auto Mesh Type at Points/Lines/Edges	P IC UN	Spandrel		one		
Auto Mesh Type at Points/Lines/Edges	Materi	al Ovenuite	M	ne		
Auto Moan Type at Fombrenes/Euges	Auto	All Overwite	110	Doints//	inos/Edaos	
Include in Analysis Mesh Yes	Includ	e in Analysis Me	esh Ve	r ontere	iles/Luges	
Croups None	Group	s	Ne	ne		
- croope	, aroop					
Section Property Section property assigned to the frame object.	Section	Property operty assigned	to the	frame ob	ject.	

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.

jure 57	▶1	Item Shored Construction?	Value	construction.
jure 57	▶1	Shored Construction?		
ure 57			No	
ure 57	2	Middle Range, %	70	
·	3	Pattern Live Load Factor	0.75	
terences	4	Stress Ratio Limit	1	
		Design Code AISC 3	i60-10 •	Explanation of Color Coding for Values Blue: Default Value Black: Not a Default Value Red: Value that has changed during the current session
				the current acasion

- 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.



- 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.





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.

						Beam B29 at Story Story4		Last Analysis	W14X3
Section	Shear Studs	Camber	Ratio	ĥ					
W12X19	22	0	0.994		Percent Comp. 85 -	Strength Checks			
W12X22	10	0	0.974		Uniform Shear Studs				
W14X22	8	0	0.823				Factored	Design	Rati
W12X26	10	0	0.803		Camber 0.00 ≑	Shear at Ends (kip)	20.036	78.960	0.25
W16X26	10	0	0.560			Construction Bending (kip-ft)	32.9542	92.6250	0.35
W14X26	10	0	0.676		Reset the Above	Full Comp. Bending (kip-ft)	120.2182	240.8573	0.49
W14X30	12	0	0.579			Partial Comp. Bending (ki	120.2182	225.2629	0.53
W16X31	8	0	0.678	Ξ					
W18X35	8	0	0.498			-			
W18X40	8	0	0.415			Constructability and Serviceability	Checks		
W21X44	8	0	0.336						
W18X50	8	0	0.318				Actual	Allowable	Rati
W24X55	8	0	0.239			Shear Studs Distribution	22	23	0.95
W21X55	8	0	0.254			Pre-composite Defl. (in)	0.751	No Limit	N/A
W21X57	8	0	0.249			Post-composite Defl. (in)	0.4412	1.2	0.36
W24X62	8	0	0.210			Live Load Defl. (in)	0.3268	0.8	0.40
W21X62	8	0	0.223			Total Defl. (in)	1.1922	1.2	0.99
W24X68	8	0	0.181	-		Walking Acceleration ap/g	0.002224	0.005	0.44
Auto Select List	A-CompBm 👻	Specify	Section		Show Group Results	Temporary Sho	w Details		
Group	NONE 👻	Overv	vrites		Show All Alternates	Combos	Diagrams	Rep	ort

#### 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.





Figure 62

- 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 <u>do not</u> get this message, re-

peat Action Items H, I and J until you do get it, before proceeding to the next Action Item. Click the **OK** button.



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.

ETABS	×
<b></b>	All composite beams passed the design check.
	ОК

- L. Click the **Select All** button, and, 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, dr., to clear the selection.
- O. Click the **File menu > Save** command, or the **Save** button, **b**, to save your model. The composite beam design is now complete.
- 62 Step 12 Design the Composite Beams

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

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

# 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.

Clear Filter           Frame Properties           A-CompBm           A-LatCol           AUTOLATBM           ConcBm           ConcCol           SteelBm           SteelCol           W10X12           W10X15           W10X19           W12X14           W12X16           W12X22           W12X26           W12X36           W12X120           W12X120           W12X120	Filter	
Frame Properties           A-CompBm           A-LatCol           AUTOLATBM           ConcBm           ConcCol           SteelBm           SteelCol           W10X12           W10X15           W10X17           W10X19           W12X14           W12X16           W12X19           W12X20           W12X106           W12X106           W12X136           W12X152		Clear Filter
A-CompBm A-LatCol AUTOLATBM ConcBm ConcCol SteelBm SteelCol W10X12 W10X17 W10X17 W10X17 W10X19 W12X14 W12X16 W12X16 W12X16 W12X22 W12X26 W12X26 W12X26 W12X36 W12X136 W12X152	Frame Properties	1
AUTOLATEM           ConcBm           ConcCol           SteelBm           SteelCol           W10X12           W10X15           W10X17           W10X17           W10X18           W12X14           W12X16           W12X22           W12X26           W12X36           W12X106           W12X136           W12X152	A-Comp Bm A-LatCol	<u>.</u>
	ConcBm ConcCol SteelBm SteelCol W10X12 W10X15 W10X17 W10X17 W10X19 W12X14 W12X16 W12X19 W12X22 W12X26 W12X26 W12X96 W12X106 W12X100 W12X120 W12X136 W12X152	

- 1. Highlight *AUTOLATBM* in the Frame Properties list and click the **Select** button. This selects all of the beams assigned the AU-TOLATBM auto select section list.
- 2. Click the **Close** button to close the form.

Figure 65 Select by Frame Property

form

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.



- 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 Pref**erences command. Select *AISC360-10* from the Design Code dropdown 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, I, 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

Figure 68

warning

design

Beam Infor-

mation form





Step 13 Design the Steel Frame 66
- 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

Story	Story4			Analysis Sectio	n	W18X	65			
Beam	B12			Design Section	n	W16X	36			
COMBO	STATION /	MOI	MENI	INTERACTI	ON CH	ECK-	//	/-MAJ-SHR	-MIN-SHR-/	
ID	LOC	RATIO	=	AXL +	B-MA	J +	B-MIN	RATIO	RATIO	
DSt1S1	10,1076	0,104(C)	-	0.008 +	0.09	5 +	0.001	0.011	0.000	
DSt1S1	12,0000	0.110(C)	=	0.008 +	0.10	2 +	0.001	1.3E-04	0.000	
DSt1S1	13,8924	0.104(C)	=	0.008 +	0.09	5 +	0.001	0.011	0.000	=
DSt1S1	15.7847	0.086(C)	=	0.008 +	0.07	7 +	0.001	0.022	0.000	
DSt1S1	17,6771	0.055(C)	=	0.008 +	0.04	6 +	0.001	0.034	0.000	
DSt1S1	19.5694	0.014(C)	=	0.008 +	0.00	5 +	0.001	0.045	0.000	
DSt1S1	21.4618	0.058(C)	=	0.004 +	0.05	3 +	0.001	0.056	0.000	
DSt1S1	23.3542	0.127(C)	=	0.004 +	0.12	2 +	0.001	0.067	0.000	
DSt1S2	0.6458	0.185(C)	=	0.006 +	0.17	8 +	0.001	0.097	0.000	
DSt1S2	2.5382	0.087(C)	=	0.006 +	0.08	0 +	0.001	0.081	0.000	
DSt1S2	4.4306	0.155(C)	=	0.011 +	0.14	2 +	0.001	0.065	0.000	
DSt1S2	6.3229	0.301(C)	=	0.011 +	0.28	8 +	0.001	0.049	0.000	
DSt1S2	8.2153	0.405(C)	=	0.011 +	0.39	3 +	0.001	0.033	0.000	
DSt1S2	10.1076	0.468(C)	=	0.011 +	0.45	5 +	0.001	0.016	0.000	
DSt1S2	12.0000	0.489(C)	=	0.011 +	0.47	6 +	0.001	4.2E-04	0.000	Ψ.
			(	Overwrites		Detail	s			

#### 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 about 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.



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 Combina- tions Selec- tion form	Strength       Deflection         Strength       Deflection         Choose Combinations       Design Combinations         List of Combinations       DSdS1         DCmpC1       DSdS1         DCmpS1       DSdS4         DSdD1       DSdS5         DSdD2       Show         Show       DSdS10	
	OK Cancel	

	The Load Combination Data	
	Load Combination Name	DStIS6
	Load Combination Type	Linear Add
Figure 73	Define Combination	
Load	Case Name	Scale Factor
Data form	Dead Sdead Cladding Windx	0.9 0.9 0.9 -1
		ОК

### 70 Step 13 Design the Steel Frame

- O. Click the **Display menu > Undeformed Shape** command or the **Show Undeformed Shape** button,  $\square$ , 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, I, 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, and, 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, **I**, to clear the selection. Click the **File menu > Save** command, or the **Save** button, **I**, to save your model. The steel frame design tutorial is now complete.

# Tutorial

# Part II - Concrete Building Example

This manual provides step-by-step instructions for building a concrete ETABS model quickly by using a template. Each step of the model creation process is identified. If you follow the instructions, you will build the model shown in Figure 1.



### The Project

The example project is a six-story building rectangular in plan. There are 4-22' bays in the X direction and 3-18' bays in the Y direction. Each story is 12 feet high.

The lateral force resisting system consists of perimeter concrete moment frames with interior shear walls. The beams are 24" wide by 30" deep, and the columns are 24" square. The walls are 14" thick. The floors consist of 8 inch concrete slabs.

In addition to the self-weight of the structure, the building will also be loaded with an additional dead load of 25 psf for partitions and equipment, along with a live load of 80 psf for both the floors and roof. Lateral loading will be wind loads based on the ASCE 7-10 code.

### Step 1 Begin a New Model

In this Step, the dimensions and story height are set. Then column, beam, and slab sections are defined along with vertical loads.

- 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.

	Model Initialization	?
	Initialization Options	0
Figuro 2	Use Saved User Default Settings	0
rigule z Modol	Use Settings from a Model File	U
Initialization	Use Built-in Settings With:	
form	Display Units	U.S. Customary 👻 🚺
IOIIII	Steel Section Database	AISC14 👻
	Steel Design Code	AISC 360-10 🗸 🚺
	Concrete Design Code	ACI 318-11 🔹 🚺
	ОК	Cancel

- 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 (1). To change the units once initialized, click the **Options menu > Display Units** command.
- E. Select ACI 318-11 from the Concrete Design Code drop-down list. 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. In this example, the model is built using the Flat Slab with Perimeter Beams template.



F. Set the number of grid lines in the Number of Grid Lines in X Direction edit box to 5.

Quick

form

- G. Set the number of grid lines in the Number of Grid Lines in Y Direction edit box to **4**.
- H. Set the grid spacing in the Spacing of Grids in X Direction edit box to **22** ft.
- I. Set the grid spacing in the Spacing of Grids in Y Direction edit box to **18** ft.
- J. Set the number of stories in the Number of Stories edit box to 6.
- K. Click the **Flat Slab with Perimeter Beams** button in the Add Structural Objects area to display the Structural Geometry and Properties for Flat Slab with Perimeter Beams form shown in Figure 4.

Overhange			- Structural System Properties		
Along X Direction	0.5	•	Column	ConcCol	•
Right Edge Distance	0.5	ft	Beam Slab	ConcBm Slab1	▼ ▼
Along Y Direction Top Edge Distance	0.5	ft	Drop	Slab1	•
Drop Panels           Image: Drop Panels         Size	6	ft	Load Dead Load Case Dead Load (Additional) Live Load Case Live Load	Dead 25 Live 80	v b/ft² v b/ft²
Restraints at Bottom	⊘ Fixed		Floor Diaphragm Rigidity Rigid	Semi-Rigid 🔘 N	lo Diaphragm

#### Figure 4

Structural Geometry and Properties form

L. Set the drop panel size to **6** ft in the Size edit box in the Drop Panels area. This model will have 6 foot square drop panels at interior columns.

M. In the Structural System Properties area, click the ellipsis .... button adjacent to the Column drop-down list. The Frame Properties form shown in Figure 5 will display.

	Trame Properties	- <b>X</b> -
	Filter Properties List	Click to:
	Type All	Import New Properties
	Filter	ear Add New Property
	Properties	Add Copy of Property
Figure 5	Find This Property	Modify/Show Property
Properties	ConcBm ConcCol	Delete Property
form	SteelCol	Delete Multiple Properties
		Convert to SD Section
		Copy to SD Section
		Export to XML File
		OK Cancel
l		

N. Click the **Add New Property** button in the Click to area of the Frame Properties form. The Frame Property Shape Type form shown in Figure 6 appears.

Trame Property Shape Type	? <mark>- x</mark>
Section Shape	Rectangular
Frequently Used Shape Types	Steel
Rectangular Section T	
Special Section Designer Beckin Designer Neprismälic Neprismälic Sector Ligt	Steel Composite
OK	Cancel

**Figure 6** Frame Property Shape Type form

Step 1 Begin a New Model

O. Select *Rectangular* from the Section Shape drop-down list in the Shape Type area and then click on the **OK** button, or click on the **Rectangular Section** button under Concrete in the Frequently Used Shape Types area of the Frame Property Shape Type form. The Frame Section Property Data form shown in Figure 7 appears.

Property Name	Concrete Column		
Material	4000Pai	•]]	2
Display Color	0	hange	3
Notes	Modfy/Show	Notes	•+
Shape			
Section Shape	Rectangular	•	• •
Section Dimensions	24	'n	Property Modifiers Modify/Show Modifiers
With	24		Currently Default
The second se			Reinforcement
			Modify/Show Rebar



- 1. Type Concrete Column in the Property Name edit box.
- 2. Set the value in the Depth edit box to 24 in.
- 3. Click the **Modify/Show Modifiers** button to display the Property/Stiffness Modification Factors form. Reduce the section stiffness for the effects of cracking, if desired, and then click **OK**.
- 4. Click the **Modify/Show Rebar** button to display the Frame Section Property Reinforcement Data form. Review the default settings and then click the **OK** button to close the form.
- 5. Click the **OK** button on the Frame Section Property Data form to return to the Frame Properties form. *Concrete Column* should be shown on the Properties list.
- P. Click the **OK** button to return to the Structural Geometry and Properties for Flat Slab with Perimeter Beams form. *Concrete Column* should be shown as the selected Column section.

- Q. In the Structural System Properties area, click the ellipsis .... button adjacent to the Beam drop-down list. The Frame Properties form will display.
- R. Click the **Add New Property** button in the Click to area of the Frame Properties form. The Frame Property Shape Type form will appear.
- S. Select *Rectangular* from the Section Shape drop-down list in the Shape Type area and then click on the **OK** button, or click on the **Rectangular Section** button under Concrete in the Frequently Used Shape Types area of the Frame Property Shape Type form. The Frame Section Property Data form appears.
  - 1. On the Frame Section Property Data from, type **Concrete Beam** in the Property Name edit box.
  - 2. Set the value in the Depth edit box to **30** in.
  - 3. Click the **Modify/Show Rebar** button and on the Frame Section Property Reinforcement Data form select the *M3 Design Only* (*Beam*) option in the Design Type area. Click the **OK** button to return to the Frame Section Property Data form.
  - 4. Click the **OK** button on the Frame Section Property Data form (after adjusting property modifiers, if needed) to return to the Frame Properties form. *Concrete Beam* should be highlighted.
- T. Click the **OK** button to return to the Structural Geometry and Properties for Flat Slab with Perimeter Beams form. *Concrete Beam* should be shown as the selected Beam section.
- U. Verify that *Slab1* is selected from the Slab drop-down list in the Structural System Properties area. Slab1 is a default 8 in thick slab section.
- V. Verify that *Drop1* is selected from the Drop drop-down list. Drop1 is a default 15 in thick slab property.
- W. In the Load area of the Structural Geometry and Properties form, type **25** in the Dead Load (Additional) edit box.

- X. Enter 80 in the Live Load edit box.
- Y. Verify that the *Rigid* option is selected in the Floor Diaphragm Rigidity area. This option removes in-plane diaphragm deformations. Click the **OK** button to accept your selections and return to the New Model Quick Templates form. The Flat Slab with Perimeter Beams button should be highlighted with a dark blue border. Click the **OK** button on the New Model Quick Templates to display the model.

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 8. The number of view windows can be changed using the **Window List** button  $\checkmark$ . View windows may be closed by clicking on the **[X]** button.

The Plan View is active in Figure 8, indicated by the highlighted display title tab. 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/off using the **Options menu > Show Bounding Plane** command.



Figure 8 The ETABS main window with Plan View active

## Step 2 Add Floor Openings

In this Step, the program is set up to add openings to multiple stories simultaneously.

### 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 highlighted. The location of the Display Title Tab is indicated in Figure 8.

- A. Click the Drawing & Selection drop-down list that reads "*One Story*" at the bottom right of the Main window, which is shown in Figure 8.
- B. Highlight *All Stories* in the list. This activates the All Stories option for drawing and selecting objects.

With the All Stories option active, as additions or changes are made to a story—for example, Story6—those additions and changes will apply to every story in the building, Story1 thru Story6.

### **Draw Shell Objects**

Make sure that the plan view is active.

A. Click the Draw Rectangular Floor/Wall button or use the Draw menu > Draw Floor/Wall Objects > Draw Rectangular Floor command. The Properties of Object form for shells shown in Figure 9 will display "docked" in the lower left-hand corner of the program.



- B. Click once in the drop-down list opposite the Property item to activate it and then select *Opening* in the resulting list. Opening will create a void where the shell is drawn.
- C. Check that the **Snap to Grid Intersections and 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. In the plan view, click once at column C-3, and while holding down the left mouse button, drag the cursor to column D-2. Release the mouse button to draw a rectangular opening.

If you have made a mistake while drawing this object, click the **Select Object** button, **A**, to change the program from Draw mode to Select mode. Then click the **Edit menu > Undo Shell Object Add** command and repeat Action Items A thru D to re-draw the openings.

- E. Click the **Select Object** button, **A**, to change the program from Draw mode to Select mode.
- F. Hold down the Ctrl key on your keyboard and left click once in the Plan View on column C-3. A selection list similar to the one shown in Figure 10 pops up because multiple objects exist at the location that was clicked. In this example, a joint object, a column object, an opening and two floor objects (drop and slab) exist at the same location. Note that the selection list will only appear when the Ctrl key is used with the left click.





- G. Select the column from the list by clicking on it and then on the **OK** button. The column at C-3 is now selected, as indicated by the dashed lines in the 3-D view. It is selected over its entire height because the All Stories feature is active. Note that the status bar in the bottom left-hand corner of the main ETABS window indicates that 6 frames have been selected.
- H. Repeat the column selection process at D-3, D-2, and C-2. The status bar should indicate that 24 frames have been selected.
- I. Press the Delete key on your keyboard or click the **Edit menu > Delete** command to delete the selection because no columns should exist at these four locations.



The model should now appear as shown in Figure 11.



The example model with the slab openings 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 ConcreteBuilding.

## Step 3 Add Walls

In this Step, a wall stack is added to model the interior walls. 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 12.





- B. Click on the E-shaped Wall button to display a wall arrangement.
- C. On the Layout Data tab, click in the Length, LX (ft) edit box and type **18**.
- D. Type **11** into the Length, LY1 (ft) edit box.
- E. Type **11** into the Length, LY2 (ft) edit box.
- F. Type **14** into all four of the Thicknesses edit boxes all of the walls should be 14 inches thick.
- G. Click the **OK** button. The Properties of Object form for Wall Stacks shown in Figure 13 will display "docked" in the lower left-hand corner of the main window.

Figure 13 Properties of Object form for Wall Stack objects

Pr	operties of Object		×
	Angle	90	
	Top Story	Story6	
	Bottom Story	Story1	

- H. Click in the Angle edit box on the Properties of Object form, set the angle to **90**, and press the Enter key on your keyboard. This will rotate the wall stack object 90 degrees from the default position.
- I. Left click once in the Plan View such that the top-left corner of the wall stack shown using dashed lines is located at C-3. The wall stack should match the geometry of the slab opening.

Notice that the wall stack spans the entire height of the building as the Top Story and Bottom Story drop-down lists in the Properties of Object form were set to Story6 and Story1, respectively.

- J. Click the **Select Object** button, k, to change the program from Draw mode to Select mode.
- K. Click the File menu > Save command to save your model.



The model now appears as shown in Figure 14.

Figure 14

The example model with the wall stack drawn

## Step 4 Define Static Load Patterns

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

As discussed at the beginning of the tutorial, the dead load consists of self-weight of the building plus an additional 25 psf that was assigned in Step 1. The live load of 80 psf was also assigned in Step 1. The ASCE 7-10 wind load that will be applied to the building will be automatically calculated by the program.

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

- B. On the expanded tree, <u>right</u>-click on the Load Patterns branch to display a context sensitive menu. On this menu, click on the Add New Load Pattern command to display the Define Load Patterns form. Note that the Dead and Live load patterns are already defined.
- C. Click in the Load edit box and type the name of the new load pattern, **Wind**.
- D. Select *Wind* from the Type drop-down list. Make sure that the Self Weight Multiplier is set to zero for the Wind load pattern. Self weight should typically be included in only one load pattern.
- E. Use the Auto Lateral Load drop-down list to select *ASCE 7-10*; with this option ETABS will automatically apply wind load based on the ASCE 7-10 code requirements.
- F. Click the Add New Load button.
- G. With the Wind load pattern highlighted, click the **Modify Lateral Load** button. This will display the ASCE 7-10 Wind Load Pattern form as shown in Figure 15.

Exposure and Pressure Coefficients     Exposure from Extents of Rigid Diapl     Exposure frem Frame and Shell Objects     Include Shell Objects     Include Frame Objects (Ope Wind Exposure Parameters Wind Direction and Exposure Width	hragms rets en Structure) Modify/Show	Wind Coefficients Wind Speed (mph) Exposure Type Importance Factor Topographical Factor, Kzt Gust Factor	85 ■ 1 0.85 ■ ■
Windward Coefficient, Cp Leeward Coefficient, Cp Case (ASCE 7-10 Fig. 27.4-8) e1 Ratio (ASCE 7-10 Fig. 27.4-8) e2 Ratio (ASCE 7-10 Fig. 27.4-8)	0.8 0.5 Create All Sets V 0.15 0.15	Directionality Factor, Kd Solid / Gross Area Ratio Exposure Height Top Story Bottom Story Include Parapet Parapet Height	0.85 Story6 • Base • ft
	ОК	Cancel	

Figure 15 Wind Load Pattern form

Step 4 Define Static Load Patterns

1. Select the *Exposure from Extents of Rigid Diaphragms* option in the Exposure and Pressure Coefficients area.

The Exposure from Extents of Rigid Diaphragms option means that the program will automatically calculate all the wind load cases as prescribed in the ASCE 7-10 code. In this example, there will be a total of 12 different load permutations of varying magnitude and direction that will be applied to the rigid floor diaphragms. Hold the mouse cursor over the to display a table of the different wind cases.

- 2. Verify that the Case drop-down list in the Wind Exposure Parameters area shows *Create All Sets*.
- 3. Review all the other wind parameters, including the exposure heights, and then click the **OK** button to return to the Define Load Patterns form displayed in Figure 16.

Define Load Patterns				? <mark>- x</mark> -
Loads Load Wind Dead Live Wind	Type Wind ▼ Dead Live Wind	Self Weight Multiplier 0 0 0	Auto Lateral Load ASCE 7-10	Click To: Add New Load Modify Load Modify Lateral Load Delete Load OK Cancel

#### Figure 16 Define Load Patterns form

- H. Click the **OK** button to accept the load patterns.
- I. Click the **File menu > Save** command, or the **Save** button, **[1]**, to save the model.

## Step 5 Review Diaphragms

In this Step, the extent of the rigid floor diaphragms will be displayed. Rigid diaphragms are typically used to model floor systems that have a large stiffness in-plane by removing the in-plane degrees of freedom. A rigid diaphragm has no in-plane deformations, and therefore, no in-plane shell stresses are reported by the program. However, in reality, these diaphragms do carry in-plane forces (see the **Display menu > Force/Stress Diagrams > Diaphragm Forces** command), and thus users should make sure that they design and detail the diaphragms for these forces, e.g. by using chords and collectors to transfer forces from the diaphragms into the lateral resisting frames and walls.

Make sure that the Plan View is active.

A. Click the Set Display Options button, *I*, or use the View menu > Set Display Options command. The Set View Options form shown in Figure 17 will display.



Figure 18 Slab Information

form

- B. On the General tab, check the *Diaphragm Extent* option in the Other Special Items area.
- C. Click the **OK** button to display the rigid diaphragm links as illustrated by the dashed lines radiating out from the diaphragm center.
- D. Right-click anywhere on the slab (but not on a beam, drop, or wall) to display the Slab Information form shown in Figure 18.

	Story	Lab	bel		Unique Name
Sto	ry6	F1		1	
GU	ID: ba448868-3c9a-	48c2-9	4f8-6f4	866	5ef890
iect	Data				
-	- Cold		-		
Ge	ometry Assignm	ents	Loa	ds	
4	Assignments				
	Opening		No		
	Disphrage	-	blab1		
	Diaphragm Property Medifiers		UT Nasa		
V	Local Axis 2 Angle	(dea) I	Defaul		
	Springs	(acy) I	None		
	Area Mass (b-s²/ft3)		0		
$\triangleright$	Floor Meshing Optic	ons I	- Defaul		
	Create Auto Edge C	Constri 1	Yes		
	Material Overwrite	1	None		
$\triangleright$	Groups		1 Grou	p	

- E. Click on the Assignments tab on the Slab Information form and note that the Diaphragm assignment is *D1*.
- F. Click the **OK** button to close the form.
- G. Click the **Define menu > Diaphragms** command to display the Define Diaphragm form.

1. On the Define Diaphragm form, highlight D1 in the Diaphragms area and click the Modify/Show Diaphragm button. The Diaphragm Data form shown in Figure 19 will display.

	Diaphragm Data	? <b>x</b>
Figure 19 Diaphragm Data	Diaphragm	þ1
Ionn	Rigidity	🔘 Semi Rigid
	ОК	Cancel

- 2. Verify that the *Rigid* option is selected in the Rigidity area to ensure that the floor slabs of this building will not have any inplane deformations.
- 3. Click the **OK** button to return to the Define Diaphragm form.
- H. Click the **OK** button to close the Define Diaphragm form.
- I. Click the Set Display Options **v** button or use the View menu > Set Display Options command and the Set View Options form will appear.
- J. Uncheck the Diaphragm Extent option.
- K. Click the **OK** button to close the Set View Options form.
- L. Click the **File menu > Save** command, or the **Save** button, 💾 , to save the model.

### Step 6 Review the Load Cases

In this Step, the load cases generated from the load patterns will be reviewed.

A. Click the **Define menu > Load Cases** command to display the Load Cases form shown in Figure 20.

Notice that three load cases are listed, one for each of the three load patterns that were defined; Dead, Live and Wind.

Load Cases			Click to:
Load Case Nar	ne Load Case Type		Add New Case
Dead	Linear Static		Add Copy of Case
Live	Linear Static		Modify/Show Case
Wind	Linear Static	*	Delete Case Show Load Case Tree
			OK

### Figure 20

Load Cases form

- B. On the Load Cases form, highlight *Wind* in the Load Cases area and click the **Modify/Show Case** button. The Load Case Data form shown in Figure 21 will display.
  - 1. Verify that *Linear Static* is selected in the Load Case Type dropdown list.
  - 2. Verify that *Wind* is shown as the Load Pattern in the Loads Applied area.
  - 3. Select the *Use Preset P-Delta Settings* option in the P-Delta/Nonlinear Stiffness area and click the **Modify/Show** button to display the Preset P-Delta Options form.

Load	d Case Data					×
Ge	eneral					
	Load Case Name		Wind			Design
	Load Case Type		Linear Statio	0	•	Notes
	Exclude Objects in this Group	<b>b</b>	Not Applica	ble		
	Mass Source		MsSrc1			
- P-	Delta/Nonlinear Stiffness					
	O Use Preset P-Delta Settin	ngs None			Modify/Show	]
	Use Nonlinear Case (Loa Nonlinear Case	ads at End of Case N	OT Included)			[
Lo	oads Applied					-
	Load Type	Load Na	me	So	ale Factor	0
	Load Pattern	Wind		1		Add
						Delete
						J
		ОК		Cancel		

#### Figure 21 Load Case Data form

- 4. On the Preset P-Delta Options form, select the *Non-iterative Based on Mass* option in the Automation Method area. This will add P-Delta effects to the analysis. Click the **OK** button to close the Preset P-Delta Options form and return to the Load Case Data form.
- 5. The Use Preset P-Delta Settings value should now show *Noninterative based on mass.* Click the **OK** button on the Load Case Data form.
- C. Click the **OK** button to close the Load Cases form.
- D. Click the **File menu > Save** command, or the **Save** button, 💾, to save the model.

Figure 22 Check Model

form

## Step 7 Run the Analysis

In this Step, the analysis will be run.

A. Click the **Analyze menu > Check Model** command. The Check Model form shown in Figure 22 will display.

Check Model	? ×
Length Tolerance for Checks	
Length Tolerance for Checks 0.1	in
Joint Checks	
Joints/Joints within Tolerance	
Joints/Frames within Tolerance	
☑ Joints/Shells within Tolerance	
Frame Checks	
✓ Frame Overlaps	
Frame Intersections within Tolerance	
✓ Frame Intersections with Area Edges	
Shell Checks	
Shell Overlaps	
Other Checks	
Check Meshing for All Stories	
Check Loading for All Stories	
OK Cancel	]

B. Check all check boxes and click the **OK** button. A warning message similar to that shown in Figure 23 should display indicating that the model has no connectivity issues.



#### 94 Step 7 Run the Analysis

- 1. Click on the **[X]** in the top-right corner to close the warning message.
- C. Click the Analyze menu > Run Analysis command or the Run Analysis button, ▶.

The program will create the analysis model from your object based input. After the analysis has been completed, the program will automatically display a deformed shape similar to that shown in Figure 24, and the model is locked. The model is locked when the **Lock/Unlock Model** button,  $\bigcirc$ , appears locked. Locking the model prevents any changes to the model that would invalidate the analysis results.



Figure 24 Deformed shape display

## Step 8 Display the Results

In this Step, the analysis results will be displayed and reviewed.

- A. Make sure that the 3D View is active this can be done by clicking on the Display Title Tab.
- B. Click on the **Display Shell Stresses/Forces...** button, **Section 20**, or the **Display menu > Force/Stress Diagrams > Shell Stresses/Forces...** command to access the Shell Forces/Stresses form shown in Figure 25.

Component Ty	be						
Resultant Fo	proes		•				
Component							
© F11	) F	Max	© V	13	● M <sup>-</sup>	11 (	) MMax
F22	© F	Min	© V.	23	© M2	22 (	) MMin
F12	© F	VM	© V	Max	© M	12	
Contour Appea	rance						
Contour Opt	ion	(	Display	on Defo	rmed Sł	ape	•
Show Lir	nes		Line	Nidth			
Show Fil	I		Trans	parenec	у	0.0	•
Show Va	lues						
Show An	rows						
Contour Values							
Min/Max Ra	ange	0			0		kip-ft/ft
Contour Ave	eraging at	Nodes	E	ly Select	ed Grou	ps	Groups
Scaling							
Automati	c (	User S	Scale Fa	ctor	600		

Figure 25 Shell Forces/Stresses form

- 1. Select *Dead* from the Load Case drop-down list.
- 2. Select *Resultant Forces* as the Component Type.
- 3. Select the *M11* component.
- 4. Select *Display on Deformed Shape* from the Contour Option drop-down list.
- 5. Check the Show Fill checkbox.
- 6. Click the **OK** button to generate the moment contours shown in Figure 26.



### Figure 26

M11 moment contours in a 3D view

- C. Make the Plan View active by clicking on the Plan View Title Tab.
- D. Click the **Show Deformed Shape** button, **D**, or the **Display menu** > **Deformed Shape** command to display the Deformed Shape form.
  - 1. On the Deformed Shape form, select *Wind* from the Load Case drop-down list.

- 2. Set the Step Number to *1* there should be a total of 12 steps available for the Wind load case.
- 3. Click the **OK** button to display the deformed shape shown in Figure 27.



#### Figure 27 Deformed Shape for Wind load case

Note that even though the wind load for Step 1 (Set 1) is applied in the X direction, there is a rotation of the structure due to the lateral stiffness eccentricity caused by the non-symmetric layout of the walls.

- E. Each of the deformed shapes due to the 12 different wind load permutations (as specified by the ASCE 7-10 code) may be viewed by clicking on the VCR buttons, 
   , located in the lower right-hand corner of the display.
- F. After reviewing the different deformed shapes, set the plan view back to an undeformed view by clicking on the Show Undeformed Shape button, , or the Display menu > Undeformed Shape command.

### Step 9 Design the Concrete Frames

In this Step, the concrete beams and columns will be designed. Note that the analysis (Step 7) should be run before performing design.

- A. Make sure that the Plan View is active this can be done by clicking on the Display Title Tab.
- B. In the Plan View, right click on the perimeter beam along grid line 1 between grids A and B. The Beam Information form shown in Figure 28 appears.

Story6     B1     121       GUID:     9e9ed9ea-49f6-432e-be33-9f96da88fe3f       jject Data       Geometry     Assignments       Loads     Design       Assignments     Concrete Beam       >     Property Modifiers     None       >     End Releases     None       >     End Releases     None       >     Instelin Point     CP at 8 - Top Center       >     Output Stations     Max Station Spacing       Local Avis 2 Angle (deg)     Default       Springs     None       Line Mass (b-s?ft?)     O       >     TC Limits     None       Spandrel     None       Material Overwrite     None       Auto Mesh Type     at Points/Lines/Edges       Include in Analysis Mesh     Yes       >     Groups     1 Group	Sto	vry	La	bel		Unique Name			
GUID: 9e9ed9ea-49f6-432e-be33-9f96da88fe3f ect Data Geometry Assignments Loads Design Assignments Section Property Concrete Beam Property Modifiers None End Releases None End Releases None End Releases None End Releases None End Releases None Insertion Point CP at 8 - Top Center Output Stations Max Station Spacing Local Avis 2 Angle (deg) Default Springs None Line Mass (b-s?/t?) 0 TC Limits None Spandrel None Material Overwrite None Auto Mesh Type at Points/Lines/Edges Include in Analysis Mesh Yes Groups 1 Group	Story6	-	B1		121				
Geometry     Assignments     Loads     Design       4     Assignments     Section Property     Concrete Beam       >     Property Modifiers     None       >     End Releases     None       >     End Length Offsets     Auto       >     Insertion Point     CP at 8 - Top Center       >     Output Stations     Max Station Spacing       Local Axis 2 Angle (deg)     Default       Springs     None       Line Mass (Ib-s²ft?)     0       >     TC Limits       None     Material Overwrite       Auto Mesh Type     at Points/Lines/Edges       Include in Analysis Mesh     Yes       >     Groups     1 Group	iUID: 9e9e	d9ea-49f6-4	- 132e-b	e33-9f9	6da8	38fe3f			
Geometry         Assignments         Loads         Design           Assignments         Section Property         Concrete Beam           Property Modifiers         None           End Releases         None           End Length Offsets         Auto           Insertion Point         CP at 8 - Top Center           Output Stations         Max Station Spacing           Local Axis 2 Angle (deg)         Default           Springs         None           Line Mass (Ib-s?ft?)         0           T C Limits         None           Material Overwrite         None           Auto Mesh Type         at Points/Lines/Edges           Include in Analysis Mesh         Yes           Groups         1 Group	ect Data								
Assignments Section Property       Concrete Beam         Property Modifiers       None         End Releases       None         End Length Offsets       Auto         Insertion Point       CP at 8 - Top Center         Output Stations       Max Station Spacing         Local Axis 2 Angle (deg)       Default         Springs       None         Line Mass (Ib-s²/ft²)       0         TC Limits       None         Spandrel       None         Auto Mesh Type       at Points/Lines/Edges         Include in Analysis Mesh       Yes         Groups       1 Group	Geometry	Assignme	ents	Loa	ds	Design			
Section Property     Concrete Beam       Property Modifiers     None       End Releases     None       End Length Offsets     Auto       Insertion Point     CP at 8 - Top Center       Output Stations     Max Station Spacing       Local Axis 2 Angle (deg)     Default       Springs     None       Line Mass (b-s <sup>2</sup> /ft <sup>2</sup> )     0       T C Limits     None       Material Overwrite     None       Auto Mesh Type     at Points/Lines/Edges       Include in Analysis Mesh     Yes       Groups     1 Group	⊿ Assign	ments							
▷     Property Modifiers     None       ▷     End Releases     None       ▷     End Length Offsets     Auto       ▷     Insettion Point     CP at 8 - Top Center       ▷     Output Stations     Max Station Spacing       Local Axis 2 Angle (deg)     Default       Springs     None       Line Mass (lb-s?ft?)     0       ▷     TC Limits     None       Spandrel     None       Material Overwrite     None       Auto Mesh Type     at Points/Lines/Edges       Include in Analysis Mesh     Yes       ▷     Groups     1 Group	Section	Property		Concre	te Be	am			
▷     End Releases     None       ▷     End Length Offsets     Auto       ▷     Insertion Point     CP at 8 - Top Center       ○     Output Stations     Max Station Spacing       Local Axis 2 Angle (deg)     Default       Springs     None       Line Mass (b-s²/ft²)     0       ▷     TC Limits     None       Spandrel     None       Material Overwrite     None       Auto Mesh Type     at Points/Lines/Edges       Include in Analysis Mesh     Yes       ▷     Groups     1 Group	Property	y Modifiers		None					
<ul> <li>▷ End Length Offsets Auto</li> <li>▷ Insertion Point CP at 8 - Top Center</li> <li>▷ Output Stations Max Station Spacing Local Axis 2 Angle (deg) Default Springs None Line Mass (lb-s²/ft²) 0</li> <li>▷ TC Limits None Spandrel None Material Overwrite None Auto Mesh Type at Points/Lines/Edges Include in Analysis Mesh Yes</li> <li>▷ Groups 1 Group</li> </ul>	End Re	leases		None					
▷       Insertion Point       CP at 8 - Top Center         ▷       Output Stations       Max Station Spacing         Local Axis 2 Angle (deg)       Default         Springs       None         Line Mass (lb-s²/ft²)       0         ▷       TC Limits       None         Spandrel       None         Material Overwrite       None         Auto Mesh Type       at Points/Lines/Edges         Include in Analysis Mesh       Yes         ▷       Groups       1 Group	End Ler	ngth Offsets		Auto					
▷         Output Stations         Max Station Spacing           Local Axis 2 Angle (deg)         Default           Springs         None           Line Mass (lb-s²/ft²)         0           ▷         TC Limits         None           Spandrel         None           Material Overwrite         None           Auto Mesh Type         at Points/Lines/Edges           Include in Analysis Mesh         Yes           ▷         Groups         1 Group	Insertior	n Point		CP at 8 - Top Center Max Station Spacing					
Local Axis Z Angle (deg)     Default       Springs     None       Line Mass (lb-s²/ft²)     0       ▷     TC Limits     None       Spandrel     None       Material Overwrite     None       Auto Mesh Type     at Points/Lines/Edges       Include in Analysis Mesh     Yes       ▷     Groups     1 Group	Output	Stations							
Springs     None       Line Mass (b-s²/ft²)     0       ▷ TC Limits     None       Spandrel     None       Material Overwrite     None       Auto Mesh Type     at Points/Lines/Edges       Include in Analysis Mesh     Yes       ▷ Groups     1 Group	Local A	xis 2 Angle (	(deg)	Defaul	t				
Dire Mass (jb 5/ft')     0       TC Limits     None       Spandrel     None       Material Overwrite     None       Auto Mesh Type     at Points/Lines/Edges       Include in Analysis Mesh     Yes       ▷     Groups     1 Group	Springs	/L -2/0.7		None					
For Links     None       Spandrel     None       Material Overwrite     None       Auto Mesh Type     at Points/Lines/Edges       Include in Analysis Mesh     Yes       ▷     Groups     1 Group	D TC Limit	TC Limits			None				
Material Overwrite None Auto Mesh Type at Points/Lines/Edges Include in Analysis Mesh Yes ▷ Groups 1 Group	Spapda	ið el		None					
Auto Mesh Type at Points/Lines/Edges Include in Analysis Mesh Yes Groups 1 Group	Material	l Overwrite		None					
Include in Analysis Mesh Yes ▷ Groups 1 Group	Auto M	esh Type		at Poin	ts/Lie	es/Edges			
Groups 1 Group	Include	in Analysis I	Mesh	Yes	Carl Vision	-car coyea			
• • • • • • • • • • • • • • • • • • •	Groups			1 Grou	D				
Section Property Section property assigned to the frame object.				- 1	e obi	ect.			



Figure 29 Concrete Frame Design Preferences form Note that the Design tab reports that the Design Procedure is Concrete Frame Design.

- C.. Click the Cancel button to close the Beam Information form.
- D. Click the **Design menu > Concrete Frame Design > View/Revise Preferences** command. The Concrete Frame Design Preferences form shown in Figure 29 displays.

	Item	Value	The selected design code.
▶ 01	Design Code	ACI 318-11	Subsequent design is based on thi selected code.
02	Multi-Response Case Design	Step-by-Step	
03	Number of Interaction Curves	24	
04	Number of Interaction Points	11	
05	Consider Minimum Eccentricity?	Yes	
06	Seismic Design Category	D	
07	Design System Omega0	2	
08	Design System Rho	1	
09	Design System Sds	0.5	
10	Phi (Tension Controlled)	0.9	
11	Phi (Compression Controlled Tied)	0.65	
12	Phi (Compression Controlled Spiral)	0.75	
13	Phi (Shear and/or Torsion)	0.75	
14	Phi (Shear Seismic)	0.6	
15	Phi (Joint Shear)	0.85	
16	Pattern Live Load Factor	0.75	Explanation of Color Coding for Va
17	Utilization Factor Limit	1	Blue: All selected objects are
et To D All	efault Values Reset T Items Selected Items Al	o Previous Values I Items Selected Items	Program determined Black: Some selected objects an user defined Red: Value that has changed o the current session

- 1. Verify that the Design Code drop-down list is set to ACI 318-11.
- 2. Review the design parameters shown on this form and then click the **OK** button to accept any changes made.
- E. With the Plan View active, click the **Design menu > Concrete Frame Design > Start Design/Check** command to start the design process. The program designs the concrete beams and columns, specifying the required reinforcing based on the shape and size of the members defined in Step 1.



When the design is complete, the longitudinal reinforcing is displayed on the model. The model appears as shown in Figure 30.

Figure 30 Longitudinal Reinforcing

- F. With the Plan View active, click on the **Move Down in List**  $\overline{\clubsuit}$  button until Plan View Story 1 is displayed.
- G. Right click on one of the perimeter beams in the Plan View shown in Figure 30. The Concrete Beam Design Information form shown in Figure 31 displays.

This form shows the required reinforcing steel calculated for each design combination at different locations along the length of the beam. The largest reinforcing value, either top or bottom steel, is highlighted.

Story Beam	Story1 B1	Se	ection Name	Concrete Beam		
COMBO	STATION	TOP	BOTTOM	SHEAR		
ID	LOC	STEEL	STEEL	STEEL		
DCON2	3,4000	0.2683	0.2683	0.0000		
DCON2	3,4000	0.2683	0.2683	0.0299		
DCON2	5.3500	0.2683	0.5205	0.0299		
DCON2	7.3000	0.2683	0.8160	0.0299		E
DCON2	7.3000	0.2683	0.8212	0.0000		
DCON2	9.2500	0.2683	0.9320	0.0000		
DCON2	11.2000	0.2683	1.0053	0.0000		
DCON2	11.2000	0.2683	1.0038	0.0000		
DCON2	13.1500	0.2683	0.8682	0.0000		
DCON2	15.1000	0.2683	0.6956	0.0000		
DCON2	15.1000	0.2683	0.6875	0.0749		
DCON2	17.0500	0.2683	0.3140	0.0749		
DCON2	19.0000	0.3792	0.2683	0.0749		
DCON2	19.0000	0.3969	0.2683	0.0000		
DCON2	21.0000	1.0821	0.5381	0.0000		-
	Overwrites	Summary	Flex. De	tails Shear	Envelope	

#### Figure 31

Concrete Beam Design Information form

1. Click the **Envelope** button on the Concrete Beam Design Information form. The Beam Section Design Report shown in Figure 32 displays.

orge Catalo										
V Ferred Decare   Prese Decare   Tructure	L									
C Page 2 at 2 . Dann	reviets -	a.	8	_						
ETARS 2013 13.0										
		EL	ARS	201	3 Cor	crete E	ame	Desi	an	
				ACI 3	18-11 B	am Sectio	Design		a.,	
				Dear	n Elemer	Chatalis (E	welcoe)			
	-	evel Elerne		lement Section		tion ID Length		RF	Type	-1
	3	1000	187	0	ungele th	um 254		1. 3	leavy Spin	0.01
		Section Properties								
	-		(in)	* (1	n) 30-0	n) di.(in)	$\mathbf{d}_{<}(\mathbf{in})$	d <sub>e</sub> (	peo.	
			24	30	0 24	0	2.5	2.5	5	
	-		-		Mater	al Propertie	E			
	- 6.4	(Baller)	F. (8	blie')	LLWI	Factor (Unit)	ms) 1	(Coller) ROCOC	1,0	believ")
		1	Q.14	-	0		~			P
	0.9		0.65	1	0.75	0	18	6.6	1	1.85
			lexural I	Reink	or come m	for Major A	ds More	ent, M.		
			End	14	End-I	Middle	Middl	. E.	-Up	End-J
			iv		%	in*	15	1000	in'	hebar
	Top (-)	(Avid)	0.53	67	0.07	0.2583	0.04		3821	0.16
		Provide)			and the	ine Manual		1.0		
		f.		1.0	ALL I	Minutes	Midd		Perch 1	Facil
		Desig	n M. 1	Intio	e Lec	Design M.	Station	Lee D	Pesign N	A. Station Los


This report shows detailed design information about the beam. Review the information on each page. Note that you can print this information using the **Print Report** button,  $\triangleq$ , on the menu bar.

- 2. Click the **[X]** button in the upper right-hand corner of the Report Viewer to close it.
- H. Click the **Cancel** button to close the Concrete Beam Design Information form.
- I. Click the **Set Elevation View** button,  $e|\xi$ , and select *A* and click the **OK** button to set the view to an Elevation View of grid line A.

The elevation view shows the area of the longitudinal reinforcing required for the columns. A right click on any column will display more detailed information about the reinforcing required.

J. Click the Design menu > Concrete Frame Design > Verify All Members Passed command. A form similar to that shown in Figure 33 should appear indicating that the shape and reinforcing of all concrete frame members is adequate.



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

- K. Click the **Assign menu > Clear Display of Assigns** command to clear the longitudinal reinforcing display.
- L. Make the 3-D View active by clicking on the Display Title Tab.

Figure 33

Verify All Members

Passed message

- M. Click the **Display menu > Undeformed Shape** command or the **Show Undeformed Shape** button,  $\Pi$ , to clear the display of the moment diagram.
- N. Click the **File menu > Save** command, or the **Save** button, 💾, to save your model.

## Step 10 Design the Shear Walls

In this Step, the concrete shear walls will be designed. Note that the analysis (Step 7) should be run before performing design.

- A. Make the Elevation View active by clicking on the Elevation View Title Tab.
- B. Click the **Set Plan View** button, Plan, select *Story6* and click the **OK** button to set the view to a Plan View of Story6.
- C. Click the **Select menu > Select > Object Type** command and the Select by Object Type form will display.
  - 1. On the Select by Object Type form, highlight *Walls*.
  - 2. Click the **Select** button, and then the **Close** button.
- D. Click the **View menu > Show Selected Objects Only** command to display only the elevator core walls in the Plan View.
- E. Click the **Define menu > Pier Labels** command. The Pier Labels form shown in Figure 34 appears.

Wall Biam	Click to
vvaii rieis	Click to.
P1	Add New Name
P1 P2	Change Name
	Delete Name
	OK Cancel



104 Step 10 Design the Shear Walls

- 1. Type **P3** in the Wall Piers edit box and click the **Add New Name** button.
- 2. Type **P4** in the Wall Piers edit box and click the **Add New** Name button.

Note that pier labels P1 and P2 are predefined by the program, with P2 pre-assigned to all walls in the wall stack.

- 3. Click the **OK** button to close the form.
- F. Make sure that *All Stories* is still selected in the Drawing & Selection drop-down list in the lower right-hand corner of the Main window.
- G. In the Plan View, click on the wall that lies along grid line C. The status bar should indicate that 6 shells have been selected.
- H. Click the **Assign menu > Shell > Pier Label** command and the Shell Assignment Pier Label form will display.
  - 1. On the Shell Assignment Pier Label form, highlight *P1* and click the **Apply** button.
  - 2. Without closing the Shell Assignment Pier Label form, click on the middle wall parallel to grid line C that is set between grid lines C and D. Highlight *P2* on the Shell Assignment Pier Label form and click the **Apply** button.
  - 3. Without closing the Shell Assignment Pier Label form, click on the wall that lies along grid line D. Highlight *P3* in the Piers area and click the **Apply** button.
  - 4. Without closing the Shell Assignment Pier Label form, "window" around the wall that lies on grid line 2. To "window," click the left mouse button above and to the left of grid intersection C-2 and then, while holding the left mouse button down, drag the mouse until it is below and to the right of grid intersection D-2. A selection box similar to that shown in Figure 35 should expand around the wall as the mouse is dragged across the model. Release the left mouse button and the program will select the wall the status bar should indicate that 12 shells have been selected.

- 5. Highlight *P4* on the Shell Assignment Pier Label form and click the **Apply** button.
- 6. Click the **Close** button to close the Shell Assignment Pier Label form.



Figure 35 Selecting Walls

- I. With the Plan View active, click the **Set Default 3-D View** <sup>3-d</sup> button to show the core in a 3-D view.
- J. Click the **Design menu > Shear Wall Design > View/Revise Pref**erences command. The Shear Wall Design Preferences form will display.
  - 1. Verify that the Design Code drop-down list is set to ACI 318-11.
  - 2. Review the design parameters shown on this form and then click the **OK** button to accept any changes made.
- K. Click the **Design menu > Shear Wall Design > Start Design/Check** command to start the design process. The program designs the shear

106 Step 10 Design the Shear Walls

walls, specifying the required reinforcing based on the shape and size of the members defined in Step 3.

When the design is complete, the pier longitudinal reinforcing is displayed on the model. The model appears as shown in Figure 36.



Figure 36 Pier Longitudinal Reinforcing

- L. Zoom in on the walls using the **Rubber Band Zoom** Q button.
- M. Right click on one of the walls in the Pier Longitudinal Reinforcing Areas view to display the Shear Wall Design Report.

This report shows detailed design information about the pier. Click the **[X]** button in the top right corner to close the report.

- N. With the Pier Longitudinal Reinforcing Areas view active, click the **Assign menu > Clear Display of Assigns** command.
- O. Click the **Set Plan View** button, Plan, select *Story6* and click the **OK** button.

- P. Click the **View menu > Show All Objects** command.
- Q. Make sure that the Model Explorer window is visible; if not, click the **Options menu > Show Model Explorer** command.
- R. Click the Tables tab in the Model Explorer to display the tables tree. Click on the **Design** node to expand the tree.
- S. Click on the 庄 Shear Wall Design node to expose the Shear Wall Pier Summary leaf. Right click on the Shear Wall Pier Summary leaf and from the context sensitive menu select Show Table. A table summarizing the design of the shear wall piers now appears across the bottom of the main window as shown in Figure 37.



## Figure 37 Shear Wall Pier Summary table

- T. Click the **[X]** button on the title bar of the Shear Wall Pier Summary table to close the table.
- U. Click the **File menu > Save** command, or the **Save** button, 💾, to save your model. This tutorial is now complete.

108 Step 10 Design the Shear Walls