# FROM START TO FINISH: MODEL, DESIGN AND OPTIMIZE A MULTI-STORY STEEL STRUCTURE USING ETABS

COMPUTERS & STRUCTURES INC.

**Computers and Structures, Inc.**
1995 University Avenue
Berkeley, California 94704, USA
http://www.csiberkeley.com

**From Start to Finish: Model, Design and Optimize a Multi-Story Steel Structure using ETABS**



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

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

Phone: (510) 845-2177
Fax: (510) 845-4096

Email: *education@csiberkeley.com* (for general questions)
Email: *support@csiberkeley.com* (for technical support questions)
Web: *www.csiedu.com*

# PREFACE

This lecture is generally geared towards the intermediate user level of ETABS. However, if you have never used ETABS before, do not be set back. We have designed this course in such a way that even the inexperienced ETABS user will have no problem following along. The end-to-end example that we present will be drawn from scratch to exhibit the most general and common modeling techniques mentioned above.

The morning will be spent discussing general modeling techniques for steel beams, columns, and braces. Part of the morning, as well as part of the afternoon will be spent on Steel Frame and Composite Beam Design. We will finish the afternoon presentation discussing the design and detailing of concrete foundations and output creation of reports for submittal.

We have chosen a very specific end-to-end example that we will describe in the presentation. In these seminar notes, you will find descriptions, computer model definitions, and results for the steel structure. As we present the model, please feel free to follow along.

# SEMINAR TOPICS

# PART I

# General Modeling

## I.1 Model Description

This is a ten-story steel braced building that contains two elevator cores at the center of the structure. There are two stories of shear walls located at the bottom of the building. Steel braces exist on certain levels on the outside bays of the structure. The building is subjected to vertical static loading and computer-generated earthquake loading per the 2006 International Building Code. Automated wind loads are also applied to the structure per the ASCE 7-02 code. The building consists of steel beams and columns along with a concrete deck on every level. Please refer to Figure I.1 for a three dimensional view of the structure.

## I.2 New Model Creation: Import Grid and 3-D Model using DXF File

The grid definition for this model contains Cartesian coordinates systems. The grid spacing in the X and Y directions are non-uniform. In this case, it is easiest to import an architectural grid from a .DXF file.

Click the File menu > Import > DXF File of Architecture Grid command to access the DXF Import form. Locate the filename/path of the .DXF file to be imported and click the Open button.

Use the drop-down boxes to select the DXF layer names that contains the lines and insertion points in the DXF file as the ETABS corresponding elements. For this model, we select the GRIDS layer and click OK. ETABS then imports the lines from

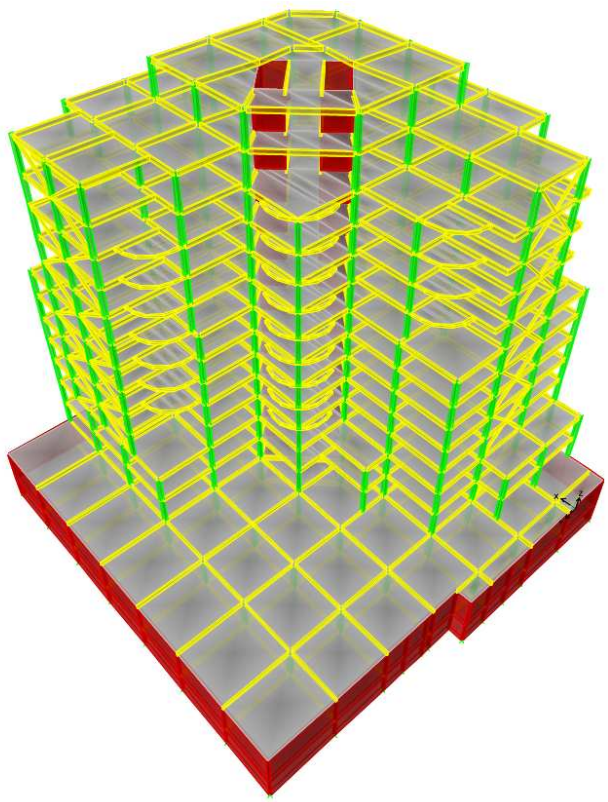


**Figure I.1:** 3D View of Structure

any layer in the DXF file as ETABS grid lines. *Note: Make sure that the gridlines are drawn as a part of the GRIDS layer in the AutoCAD .DXF file.* In the Coordinate System box, choose the GLOBAL coordinate system for importing the file. Also select the units as *kip-ft*. Your grid system should look like Figure I.2. If you want to import multiple grid systems, you could do so by following the same steps to import another grid system to the same model.

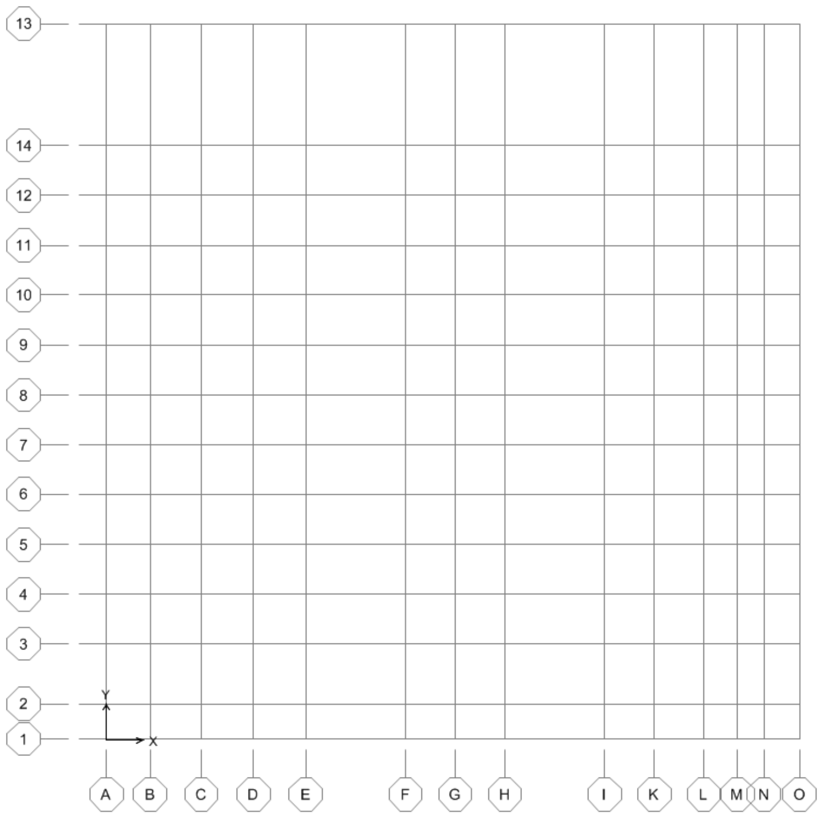



**Figure I.2:** Architectural Grid

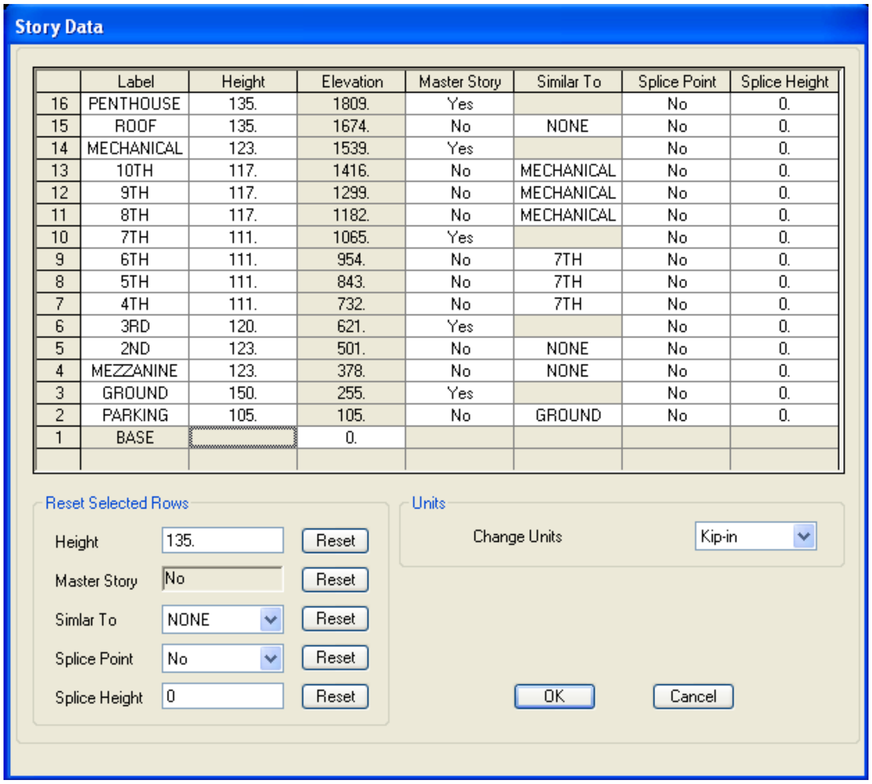
Right now, the model consists of a single story. To insert the additional 15 stories needed for the completion of the grid, go to Edit > Edit Story Data > Insert Story. Enter the New Story Data and New Story Location as shown in Figure I.3:

Your model now contains 16 total stories. As you can see in Figure I.1, stories 4-5 are identical to each other; just as stories 11-13 are identical to each other. In this


**Figure I.3:** Insert New Story Dialogue Box

case, it would be beneficial to use the ‘Similar Stories’ option in ETABS. This means when you draw an object in plan, this object will be drawn on all stories that have been designated to be similar to that story. An assignment made to an object in a plan view also occurs at all levels designated as similar to the story. When an object is selected in plan view, objects of the same type in the same location at different story levels that are designated as similar to the story where the selection is actually made are also selected. If desired, change the similarity option in the drop-down box in the status bar at any time in a plan view. When you are in an elevation view or a three-dimensional view, the similar stories drop-down box displays the word “Inactive” to remind you that the feature is inactive in these views. When you switch from plan view to an elevation or three-dimensional view and then back to a plan view, ETABS will retain the similarity option that it last had in the drop-down box in the status bar for the previous plan view. To activate the Similar Stories option, go to Edit > Edit Story Data > Edit Story. Make changes (including story heights) as shown in Figure I.4:

Now, we can import the steel structure into ETABS. Click the File menu > Import > DXF 3D Model command to access the DXF Import form. Use the form to locate the filename/path of the .DXF file to be imported. Highlight the filename and




| | Label | Height | Elevation | Master Story | Similar To | Splice Point | Splice Height |
|---|---|---|---|---|---|---|---|
| 16 | PENTHOUSE | 135. | 1809. | Yes | | No | 0. |
| 15 | ROOF | 135. | 1674. | No | NONE | No | 0. |
| 14 | MECHANICAL | 123. | 1539. | Yes | | No | 0. |
| 13 | 10TH | 117. | 1416. | No | MECHANICAL | No | 0. |
| 12 | 9TH | 117. | 1299. | No | MECHANICAL | No | 0. |
| 11 | 8TH | 117. | 1182. | No | MECHANICAL | No | 0. |
| 10 | 7TH | 111. | 1065. | Yes | | No | 0. |
| 9 | 6TH | 111. | 954. | No | 7TH | No | 0. |
| 8 | 5TH | 111. | 843. | No | 7TH | No | 0. |
| 7 | 4TH | 111. | 732. | No | 7TH | No | 0. |
| 6 | 3RD | 120. | 621. | Yes | | No | 0. |
| 5 | 2ND | 123. | 501. | No | NONE | No | 0. |
| 4 | MEZZANINE | 123. | 378. | No | NONE | No | 0. |
| 3 | GROUND | 150. | 255. | Yes | | No | 0. |
| 2 | PARKING | 105. | 105. | No | GROUND | No | 0. |
| 1 | BASE | | 0. | | | | |



**Figure I.4:** Edit Story Data Dialogue Box

click the Open button.

A form will appear where you can select DXF layers that contain entities that will be imported into ETABS as column, beam, brace, floor, wall or ramp objects. ETABS will import lines in the vertical plane as columns, lines in the horizontal plane as beams and lines in inclined planes as braces. 3DFaces in the DXF file will be imported as ramp objects if they are in the incline plane, walls if in the vertical plane, and floors if in the horizontal plane. Polylines in horizontal planes will be imported as floors. An option is available to select properties for the beams, columns, braces, walls, ramps and floors, which will be imported into ETABS as assignments. *Select the Centerlines, and Walls, layers to be imported into ETABS and click OK.*

As shown in Figure I.5, the entire steel model has been imported into ETABS. There are some default sections that have been assigned to the beams and columns. The user will have to assign specific sections to all frame elements. In the next section, the drawing tools will be used to create beams, columns, walls and brace elements.

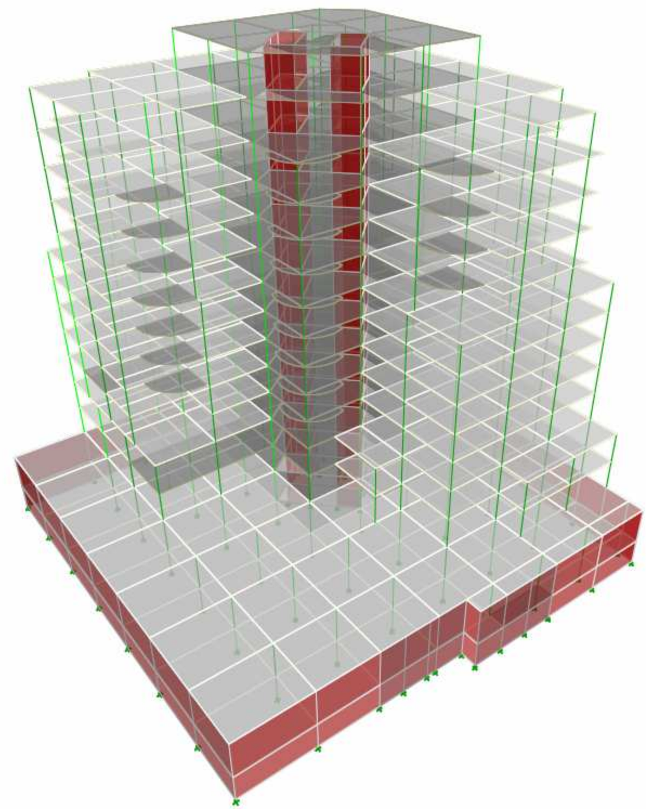
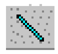


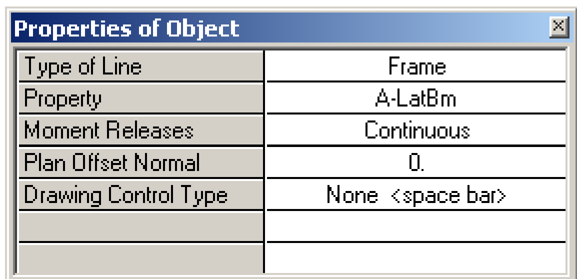
**Figure I.5:** Imported Model from .DXF file

## I.3 Model Creation Using the GUI (Steel Beams, Columns and Braces)

ETABS provides an icon or button for virtually every command. These icons reside in the toolbars located on the top and left side of the screen. For this model, all of the steel beams and columns have been imported from a .DXF file. The braces will be drawn in manually using the drawing tools. Many changes are typically made to a model after it has been imported. Listed below are the most commonly used drawing buttons.

- Draw Lines 

  When drawing objects, a floating Properties of Object form will pop up. The dialogue box shown in Figure I.6 pops up when you click on the Draw Lines button. The user must select the line type, the section, moment releases (continuous or pinned), plan offsets and the drawing control type. The plan-offset option automatically draws a line directly parallel to the line drawn. The user just specifies the distance. The drawing control type allows the user to draw line parallel to the $X$ or $Y$-axis and draw a line with a fixed length or angle. For some objects, moment releases, number of beams, spacing, orientation, bracing configuration, and the like can be assigned.



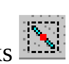
Properties of Object

| | |
|---|---|
| Type of Line | Frame |
| Property | A-LatBm |
| Moment Releases | Continuous |
| Plan Offset Normal | 0. |
| Drawing Control Type | None <space bar> |
| | |
| | |

**Figure I.6:** Beam Properties Dialogue Box

There are a few steps simple steps to follow when drawing a line element. First, left click once at the beginning of the line. Drag the mouse to the end location of the line and left click again. Note that as the mouse is dragged a dashed line is visible, indicating the current extent of the line object.Left click once on the end point of the line object to draw another line object starting from the end of the first; continue as needed. Double left click or single left click and press the Enter key on the keyboard to terminate the drawing of the next line.

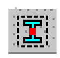
When using this command in an elevation view or 3D view, if a line object is drawn that crosses story levels, ETABS immediately breaks the object at the story levels. For example, if a line object is drawn that has its top at the 4th story level and it bottom at the 2nd story level, ETABS immediately breaks the object into two objects with the break point at the 3rd story level.

- Create Lines at Regions or at Clicks 

  The Draw menu > Draw Line Objects > Create Lines in Region or at Clicks (plan, elev, 3D) command allows the user to draw beam elements at any location in the model. There are two ways you can draw the columns.

You can click on any grid line (in plan view only) and a line object is drawn on that grid line between the two adjacent intersecting grid lines from the same coordinate/grid system.

Alternatively, in all views, depress and hold down the left button on your mouse. While keeping the left button depressed, drag the mouse to "rubber band" a window around one or more grid line segments. Then release the left mouse button. Line objects are automatically placed at each grid line segment included in the "rubber band" window. The term grid line segment in this paragraph means that portion of a grid line between the two adjacent intersecting grid lines *from the same coordinate/grid system*.

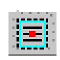
- Create Columns at Regions or at Clicks 

After you have activated the Draw menu > Draw Line Objects > Create Columns in Region or at Clicks (plan) command there are two ways you can draw the columns. They are:

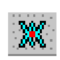
Left click at any location in a plan view to draw a column (vertical line object below).

Working in plan view, depress and hold down the left button on your mouse. Perform the same operation when creating line at regions. Columns (vertical line objects below) are automatically placed at each grid line intersection of two grid lines *in the same coordinate/grid system* included in the "rubber band" window.

The columns (vertical line objects) extend from the story level that you draw them on to the story level below, and, of course, also to other story levels if you have the similar stories feature in the ETABS status bar activated.

- Create Secondary Beams at Regions or at Clicks 

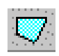
The Draw menu > Draw Line Objects > Create Secondary Beams in Region or at Clicks (plan) command allows you to draw typical infill (secondary) beams for an entire grid line space (beam bay) in a single click. The grid line space is defined by four adjacent intersecting grid lines. If beams are already in the grid line space, the spacing and extent (length) of the secondary beams is based on the existing beams rather than the grid lines. The figure below shows an example of a grid line space and secondary beams. Note that secondary beams are not included on the grid lines.

- Create Braces at Regions or at Clicks 

Draw menu > Draw Line Objects > Create Braces in Region or at Clicks Use this command to quickly draw brace elements in a space bounded by two adjacent grid lines (from the same coordinate/grid system) and two adjacent story levels.

Click inside the space created by the intersection of two adjacent grid lines (from the same coordinate/grid system) and two adjacent story levels.

Alternatively, depress and hold down the left mouse button. While keeping the left button depressed, drag the mouse to "rubber band" a window around one or more grid line/story level spaces. Then release the left mouse button. Braces are automatically placed in each grid line/story level space within the region that is fully included in the "rubber band" window. The term grid line/story level space in this paragraph means the space created by the intersection of two adjacent grid lines (from the same coordinate/grid system) and two adjacent story levels.

- Draw Area Object 

  To draw an area object using the Draw menu > Draw Area Objects > Draw Areas (plan, elev, 3D) command, left click once at the first corner point of the area, drag the mouse to the next corner point and left click, and so on to define each corner point of the area object. Note that as you drag the mouse a dashed line is visible indicating the current extent of the area object.

  When you reach the last corner point of the area object, double left click to finish the object or single left click and then press the Enter key on the keyboard.

  Area objects drawn using this command must have at least three corner points. Typically area objects are limited to no more than four corner points; however, there is no limit on the maximum number of corner points allowed for horizontal area objects (in the global $X$-$Y$ plane).

Since the model was imported from a .DXF file, some of these draw tools will not be implemented in the construction of this model. The next step is to assign the correct beam and column section properties to the line elements.

Use the Assign menu > Frame/Line > Frame Section command to assign frame section properties to line objects. The frame section property can be a previously defined property or you can define it as part of the assignment process.

The Assign menu > Frame/Line > Frame Section command can be used to

- import sections from pre-defined databases,
- define frame section properties on the basis of their dimensions,
- review and modify section properties, and
- delete section properties.

To assign frame sections:

- Select the line objects to which section properties are to be assigned.
- Click the Assign menu > Frame/Line > Frame Section command to bring up the Assign Frame Properties form.
- On the Assign Frame Properties form, the “Properties” area of this form lists the names of all the currently defined frame section properties; ETABS automatically creates this list, which you can use or add to, modify, or delete as necessary using the two drop-down lists.
- To view or modify a certain section, click on the modify/show section button. See Figure I.7:

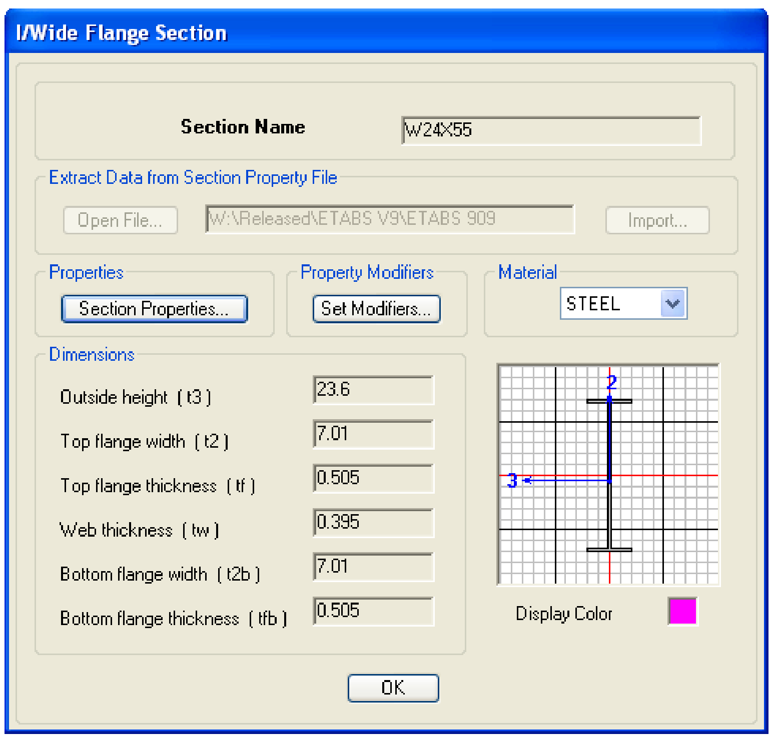



**Figure I.7:** Beam Section Properties Dialogue Box

## I.4 Importing/Exporting to/from various BIM Implementation

Building Information Modeling (BIM) is a buzzword that is becoming more commonly used by the structural engineering community. BIM implements the concept of storing all of the information about a building in one single computer model. It allows users to maintain a database from which project lifecycles can manage. BIM implementations allow users to create and manipulate construction and fabrication documents with great ease.

Currently on the market, there are a handful of BIM implementations available. CSI recognizes the value in such products and as responded accordingly. Over the past few years, CSI has proactively worked to create interfaces with various BIM implementations. These interfaces allow users to import and export their Building Information Models into and out of ETABS to make use of the state of the art analysis and design functionality that the structural engineering community has been using with confidence for the past 30 years. Currently CSI has built interfaces with the following BIM implementations:

- Autodesk®Revit®Structure (Autodesk)
- Archicad ®(Graphisoft)
- Tekla Structures (Tekla Corporation)

In today's seminar, we will touch on how ETABS works with these various products. For additional information on the ETABS interfaces with various these third party BIM products, please contact CSI.

## I.5 OpenGL Graphics Option in ETABS

The new viewer in ETABS is a new feature that enables the user to see the model using OpenGL graphics. In later versions of ETABS, this will be the default view used. With the active window displaying a Plan or 3D view, click the View menu > Create OpenGL View command to display a rendered view of your model in the Rendered Viewer window.

This Rendered Viewer window has a number of buttons for manipulating the rendered image. Some of the most useful buttons are explained below:

This Rendered Viewer window has a number of buttons for manipulating the rendered image.

**Save Image** Saves the image as a .bmp, .gif, .jpeg, .png, or tif file.

- EntireForm
- PlotOnly
- UserRegion

**Reset and Refresh View** Re-displays the image. As you manipulate the image, you may wish to return to the original orientation. Click this button to restore the display. This is also useful if the Viewer appears blank.

**Light Control** Allows you to select "Lighting" from the top and bottom, or from eight compass directions. With this option the perspective and orientation of the model in the Viewer window remains unchanged.

**Modify Light Source** Choose between Bright, Dim, or Dark to decrease or increase shadow effects.

**Modify Dynamic Movement Speed** Modifies the speed at which changes are made in the Viewer window using the other commands.

**Move Camera** Moves the model in the Viewer window while maintaining the perspective and orientation.

**Pan**. Allows panning of the rendered view.

**Zoom In/Out** Use this command with the left mouse button to Zoom In and the right mouse button to Zoom Out.

**Rubber Band Zoom**. Allows zooming in on the rendered view by windowing. To use the command, depress and hold down the left button on your mouse. While keeping the left button depressed, drag the mouse to "rubber band" a window around the portion of the view that you want to zoom in on. The rubber band window that shows the extent you have dragged the mouse appears as a dashed line on your screen. When you release the mouse left button, the new "zoomed" view is displayed.

**Restore Previous Zoom** Restores the previous view.

**Rotate** Moves the image in the Viewer window left and right while maintaining the perspective.

**Walk** Moves forward, back, left and right "through" the images as if you were walking within the structure.

**Side Views** Changes the orientation of the model relative to the x and y coordinate systems.

In Figure I.8, you can see how the structure looks using the OpenGL viewer. When using the OpenGL viewer, the user can create .AVI movie files while moving through the structure. To do this, press the 'Start Capturing Scene' button the toolbar on the top of the screen. When finished, press the 'Stop Capturing Scene' button and save the .AVI file.

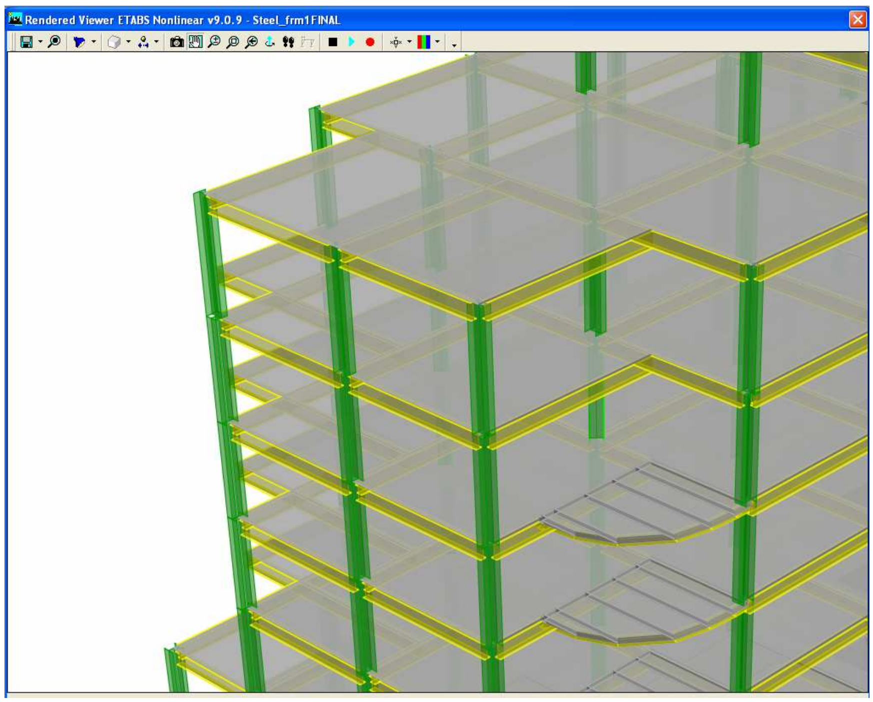



**Figure I.8:** OpenGL Rendered View

## I.6 Slab Meshing Options

*NOTE: During analysis, ETABS automatically meshes (divides) area objects that are assigned deck properties or slab properties with membrane behavior only*. If a slab element is assigned a shell property, you must assign floor auto-meshing options or manually mesh the slab. Meshing helps distribute loads realistically.

A wall or slab section can have shell, membrane or plate-type behavior. Shell-type behavior means that both in-plane membrane stiffness and out-of-plane plate bending stiffness are provided for the section. Membrane-type behavior means that only in-plane membrane stiffness is provided for the section. Plate-type behavior means that only out-of-plane plate bending stiffness is provided for the section.

During analysis, ETABS automatically meshes (divides) area objects that are assigned deck properties or slab properties with membrane behavior only. For this example, we are using slab type elements; therefore, we must mesh the slabs manually. First, select all of the floor elements (from Select > Area Object Type > Floors) and go to Assign > Shell/Area > Area Object Mesh Options. Click on the Auto Mesh Objects into Structural Elements button. Then, select the $1^{st}$ three options. These options are 1) Mesh at beams and other meshing lines 2) Mesh at wall and ramp edges and finally 3) Mesh at visible grid lines. Meshing helps distribute loads realistically. In some cases you may not want ETABS to automatically mesh an area object into the analysis model.

When an element does not frame into the corner point of a shell element, but instead frames into the edge of the shell element, no connection exists between the element and the shell element. ETABS is able to perform the analysis in this fashion. However, ETABS also has a very powerful feature called an Auto Line Constraint. The ETABS auto line constraints feature allows you to specify that elements framing into the edge of a shell element be connected to the shell element. ETABS internally takes care of connection between the elements by constraining points lying along an edge of the shell element to move with that edge of the element. This option is located under the Assign Menu > Shell/Area > Auto Line Constraint. By default, the Auto Line Constraint feature in ETABS is active (i.e. turned on). You have to flexibility to use the line constraint on the entire model or specific elements of the model. Additional information regarding the auto line constraint functionality in ETABS can be found at the end of this manual in a paper entitled "*Mesh Transitioning and Compatibility using the auto line constraint in ETABS and SAP2000*."

To demonstrate how the different slab meshing options work, let's take a look at a small model. In this example, there are 6 individual frame systems, each with different slab element properties. You will see that certain slab property types distribute load in different ways.

For this example, the self-weight multiplier in the DEAD load case has been set to zero. Instead, a 50 psf surface load has been applied to all of the slab elements. Listed below are the slab property names and definitions:

**DECK1:** A deck property has been defined. ETABS has built-in default area object properties for DECK1, a metal deck. Changes can be made to the type of section, the geometry, the material, the unit weight of the section. DECK properties apply load to an area object as a one-way slab spanning in the local 1 direction of the area object.

**1WAYM:** A membrane type property has been defined. Membrane-type behavior means that only in-plane membrane stiffness is provided for the section. Also, the *Use Special One-Way Load Distribution* checkbox has been selected which means the load applied to area objects with this section property is to be distributed as a one-way slab spanning in the local 1 direction of the area object.

**SLAB6:** A 6” slab with a shell type property has been defined. Shell-type behavior means that both in-plane membrane stiffness and out-of-plane plate bending stiffness are provided for the section.

**SLAB6M:** A 6” slab with a membrane type property has been defined. For this slab element, the Special One-Way Load Distribution box *has not* been checked.

The two structures on the right side both have SLAB6 sections assigned to them. The only difference is that these slabs have been meshed. To do this, select the top right slab element, go to Edit > Mesh Areas > select the Mesh Quads/Triangles into 2 by 2 sections. This will break the slabs into 4 total pieces. Next, select the bottom right slab element and repeat this procedure but this time enter a $4 \times 4$ mesh. Please refer to Figure I.9.

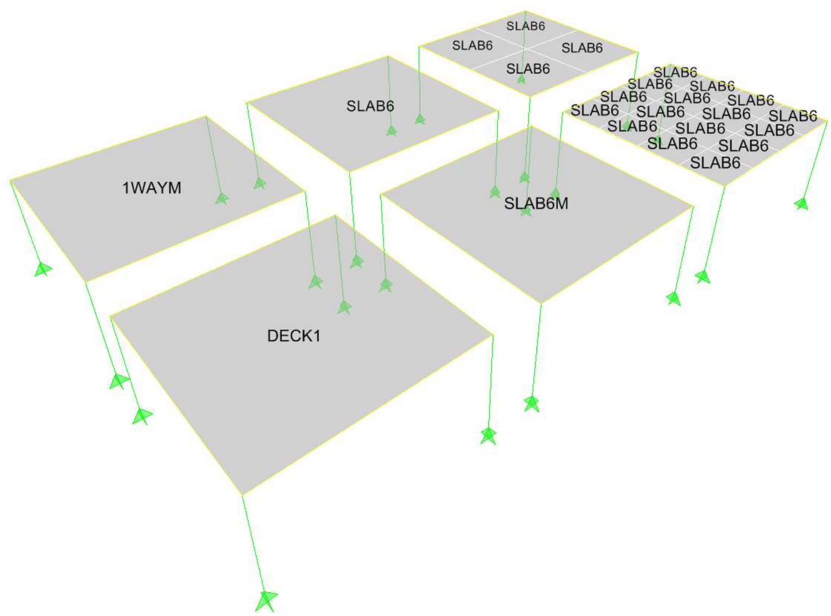



**Figure I.9:** 3D View — Meshing Example

After you run the analysis, go to Display > Show Member Forces/Stresses Diagram > Frame/Pier forces. From the pull down menu, select the DEAD load case and the Moment 3-3 component. The moments for all of the frame systems will be shown graphically. Please refer to Figure I.10:

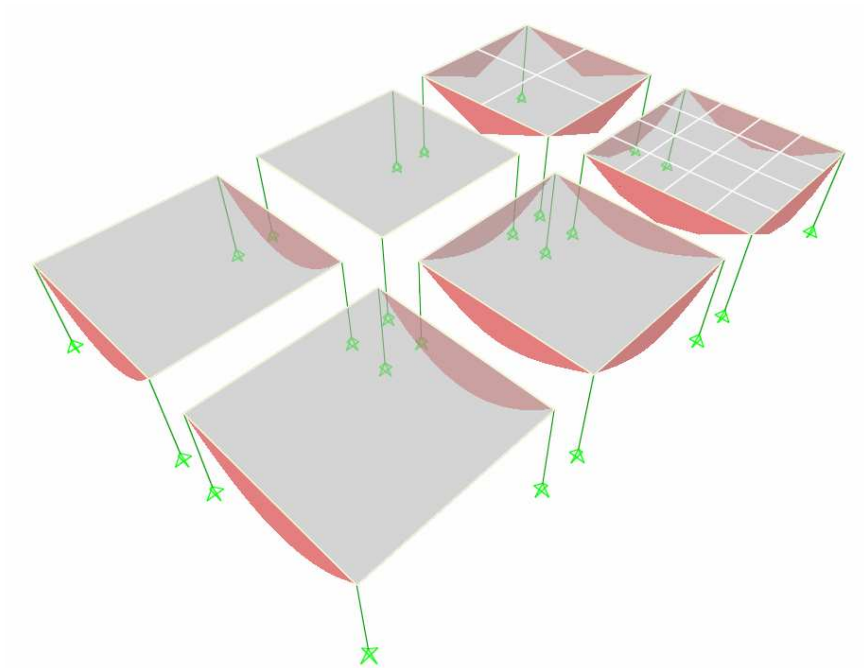


**Figure I.10:** 3D View — Moment Diagram

As you can see, the 1WAYM and DECK1 properties behaved very similarly. The 1WAYM slab was assigned a membrane property, so it was automatically meshed internally. The one-way load distribution spanning in the local 1 direction is clearly shown on the left 2 frame systems. There is no moment on the top and bottom frame elements. SLAB6 and SLAB6M behaved very differently. SLAB6 was assigned a shell property type, so it was *not* meshed internally. The loads were applied to the four corners of the slab; therefore no moments are shown on the frame elements. SLAB6M was assigned a membrane property type, which was meshed automatically. Loads were distributed to the frame elements in both directions. The frame systems on the right hand side both have SLAB6 properties assigned to them. The difference is that one slab was meshed more densely than the other. Meshing of slab elements allows loads to be distributed at the mesh points. As shown in Figure I.10, if the mesh is more refined, there will be more points of load application. That is why the moment diagrams are different for the frames on the right hand side.

# PART II

## Static and Dynamic Loading of Diaphragms

### II.1 Rigid and Semi-Rigid Floor Diaphragms

Assigning a diaphragm to an area object provides a diaphragm constraint to all of the corner points of the area object and to any additional point objects that are enclosed within the boundaries of the area object. This includes any points (joints) that are created as a result of automatic area object meshing.

**Important**: *Diaphragms can be horizontal only*. Thus diaphragm assignments are not applicable to wall-type and ramp-type area objects. They are applicable only to floor type area objects and to null-type area objects that happen to be in a horizontal plane.

In this model, we will assign semi-rigid diaphragms to all floor elements. This is a new option in ETABS. First, select all floor elements, and under the Assign > Shell/Area > Diaphragms option, click the Add New Diaphragm button to access the Diaphragm Data form. Use the default name suggested in the *Diaphragm* edit box, or enter another name.

Specify the rigidity of the diaphragm by choosing the *Rigid* or *Semi Rigid* option. These options affect only the analysis of the model. If the rigid option is selected, a fully rigid diaphragm is assumed. If the semi rigid option is selected, the in-plane rigidity of the diaphragm comes from the stiffness of the objects that are part of the diaphragm. Select the Semi-Rigid option and click OK. You will now see the semi-rigid diaphragm has been assigned the all floors of the structure. The point where all of the lines of the diaphragm intersect is the center of mass. See Figure II.1:

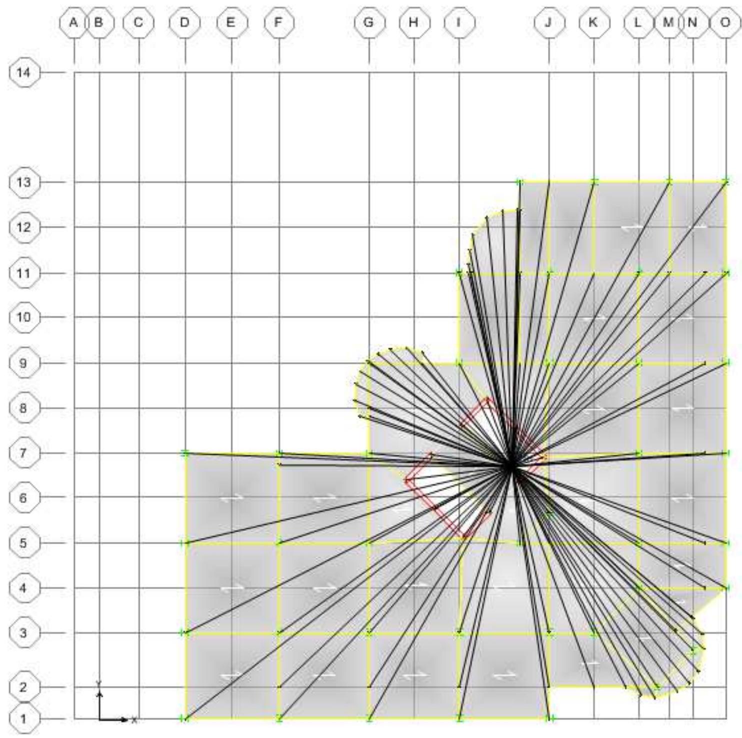



**Figure II.1:** Semi-Rigid Diaphragm Assignment

## II.2 IBC 2003 Seismic Loads

To define earthquake and wind loads to the structure, you must go to Define > Static Load Cases. You can see that there are two default load cases already defined, DEAD and LIVE. Type EQX in the LOAD box, select a QUAKE type case, and then select the IBC 2003 seismic code from the pull down menu. Click Add New Load followed by the Modify Lateral Load button. The menu shown in Figure II.2 will pop up.

Choose to specify the $X$ or $Y$ direction of the seismic loading, or to specify the direction with a percentage of eccentricity that is applicable to all diaphragms. Use the *% Eccentricity* edit box to specify a value for eccentricity. For this example, we will select to apply the loads in the $X$-dir.

Choose the **Top Story** and **Bottom Story** to specify the elevation range over which the automatic static lateral loads are to be calculated. By default the bottom

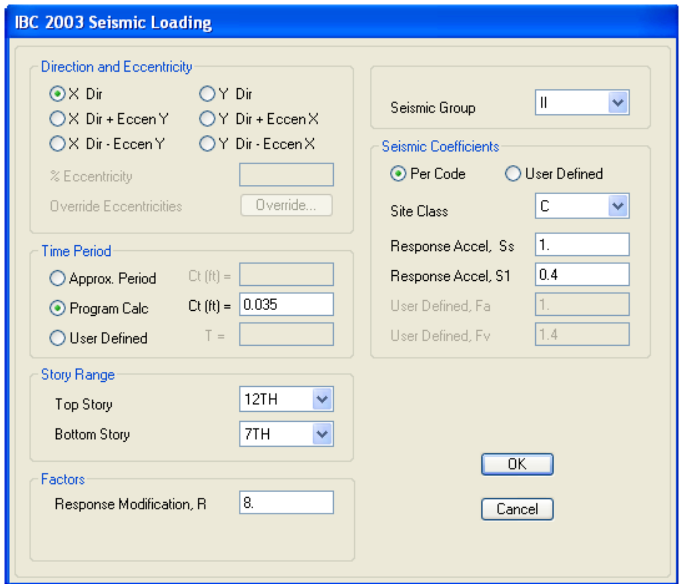



**Figure II.2:** IBC 2003 Seismic Load Definition

story is the base of the building and the top story is the uppermost level of the building.

In most instances, specify the top story as the upper-most level in the building, typically the roof. The bottom level would typically be the base level. However, for this example, the building has several below-grade levels, and the seismic loads are assumed to be transferred to the ground at ground level, so it is best to specify the bottom story to be the $7^{th}$ Story (level of the ground). Enter the Response Factor, Seismic Group and Seismic Coefficients as shown in Figure II.2.

**IBC2006 Seismic and Wind loads will be included in the next version of ETABS.**

## II.3 Response Spectra Load Application

In this example, we are interested in using a pre-defined response spectrum function from the ETABS database. Go to Define > Response Spectrum Function > Select the IBC 2003 Spectrum from the pull-down menu. As shown, in Figure II.3, the user can change the Design Spectral response for the curve. The period and acceleration values are shown on the right hand side of the menu. These values are associated with the IBC2003 code and cannot be edited. To enter your own period and acceleration values, you must select the “Add user Spectrum” option from the pull-down menu.

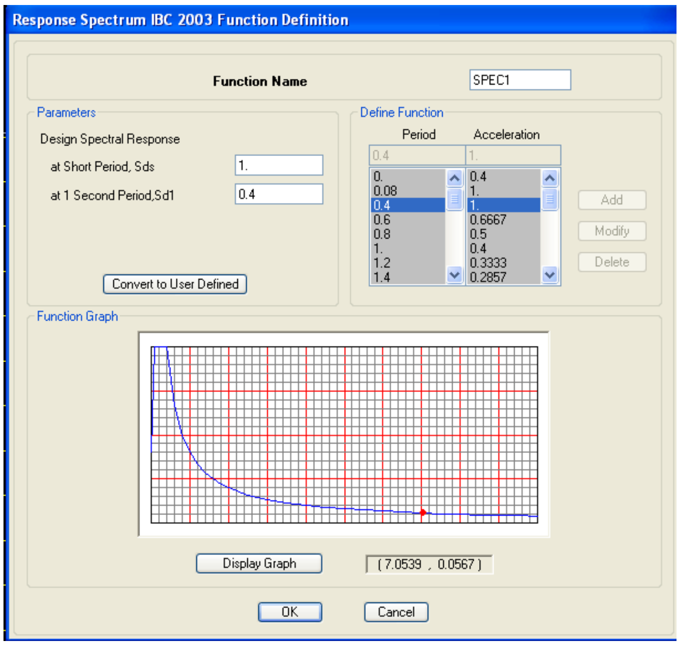



**Figure II.3:** Response Spectrum Function Definition

Next, we define the Response Spectrum case data. Go to the Define Menu > Response Spectrum Cases > Add New Spectrum. There are many options available including Structure and Function Damping, Modal and Directional Combination, Input Response Spectra and Eccentricities. It is important that you understand the structural and function damping item. This item specifies modal damping that is present for all modes in the response spectrum analysis. Also, ETABS assumes that the response spectrum functions specified for the response spectrum case are all specified for this particular damping ratio.

For example if you specify 2% damping for this term, there is 2% modal damping in all modes for the response spectrum analysis **and the response spectrum functions specified for this response spectrum case are for 2% damping.**

The user must specify the method ETABS uses to combine modal responses in the response spectrum analysis and also define a damping value. The following options are available for modal combinations:

**CQC:** This is the Complete Quadratic Combination method described by Wilson, Kiureghian and Bayo (1981). This modal combination technique takes into account the statistical coupling between closely spaced modes caused by modal damping. Increasing the modal damping increases the coupling between closely spaced modes. If the modal damping is 0 for all modes, then the CQC method degenerates to the SRSS method.

**SRSS:** This is the Square Root of the Sum of the Squares method. This modal combination technique does not take into account any coupling of modes as do the CQC and GMC methods.

**ABS:** This is the Absolute method. This modal combination technique simply combines the modal results by taking the sum of their absolute values. This method is usually over-conservative.

**GMC:** This is the General Modal Combination method that is also known as the Gupta method. This method is the same as the complete modal combination procedure described in Equation 3.31 in Gupta (1990). The GMC method takes into account the statistical coupling between closely spaced modes similar to the CQC method, and it also includes the correlation between modes with rigid-response content.

We are interested in assigning the response spectrum function loading (SPEC1) in the X-direction. Enter a scale factor of 386.4 if you are using k-in units. The CQC modal combination was selected as well as the SRSS directional combination. Please see Figure II.4:

## II.4 Time History Load Application

In this example, we are interested in defining a pre-defined time history function from a text file. Go to Define > Time History Function > Add Function from File. Click the Browse button and select the time history text file. The record we have selected is the 1940 El Centro earthquake record. This text file contains time and acceleration values for the specified earthquake. You can view time history graph by clicking on the Display Graph button. Fill in the Function File information as in Figure II.5:

Next, we define the Time History case data. Go to Define > Time History Cases > Add New History. There are many options available including Analysis Type, Number of Output Time Steps and Output Time Step Size.The output time step size is the time in seconds between each of the equally spaced output time steps. Do not confuse this with the time step size in your input time history function. The number of output time step size can be different from the input time step size in your input time history function. The number of output time steps multiplied by the output time step size is equal to the length of time over which output results are reported.

We are interested in assigning the time history acceleration in the x-direction. The time history is a 12-second record, so enter 6000 time steps and time step size of

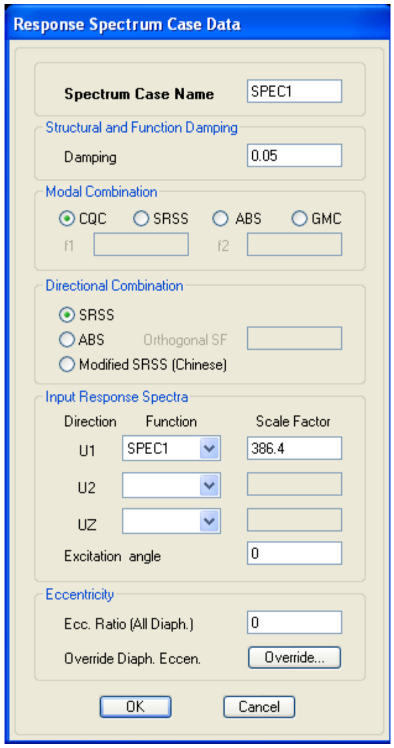



**Figure II.4:** Response Spectrum Case Function Definition

0.002 seconds. Enter a scale factor of 386.4 if you are using k-in units. Please see Figure II.6:

## II.5 Use of Special Seismic Load Effects

ETABS can automatically calculate the Rho factor, which is a reliability factor based on system redundancy. Go to the Define Menu > Special Seismic Load Effects to enter special seismic load data. There are many options in this menu, including selection of Rho factor, IBC2000 seismic design category, and later force resisting system type.

If you select the **Include Special Seismic Design Data** option, the program calculated (or user defined) Rho factor and the user defined DL Multiplier are *automatically* applied to program default design load combinations for American codes (ACI, AISC, UBC) that include contributions from earthquake loads. Earthquake loads in this case are assumed to be all static loads of type Quake, and all response spectrum

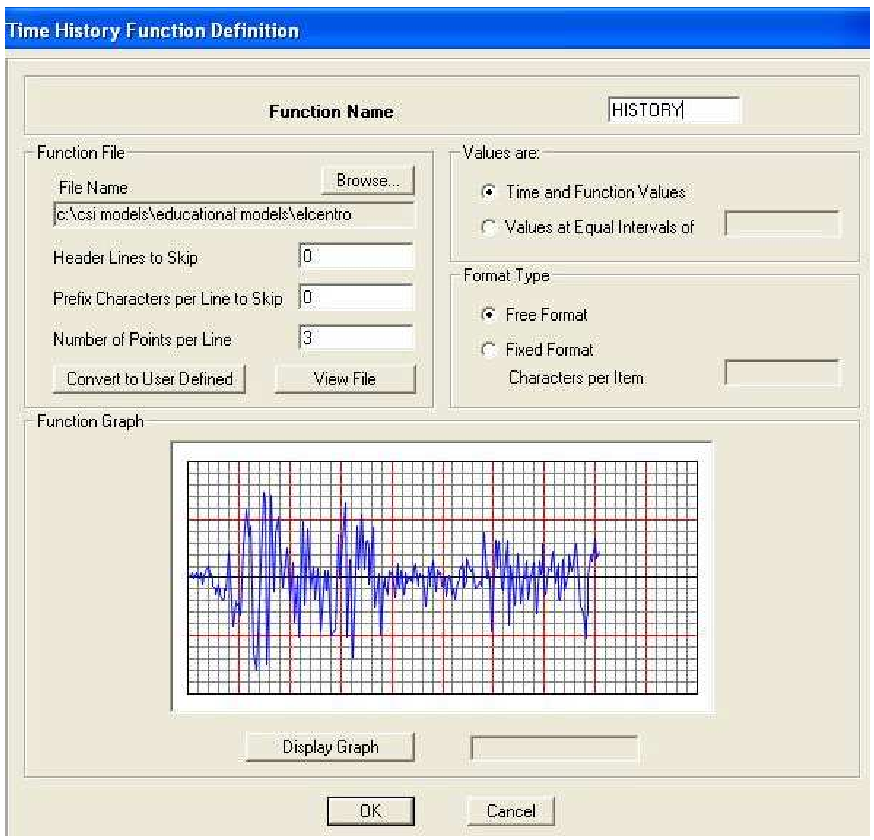



**Figure II.5:** Time History Function Definition

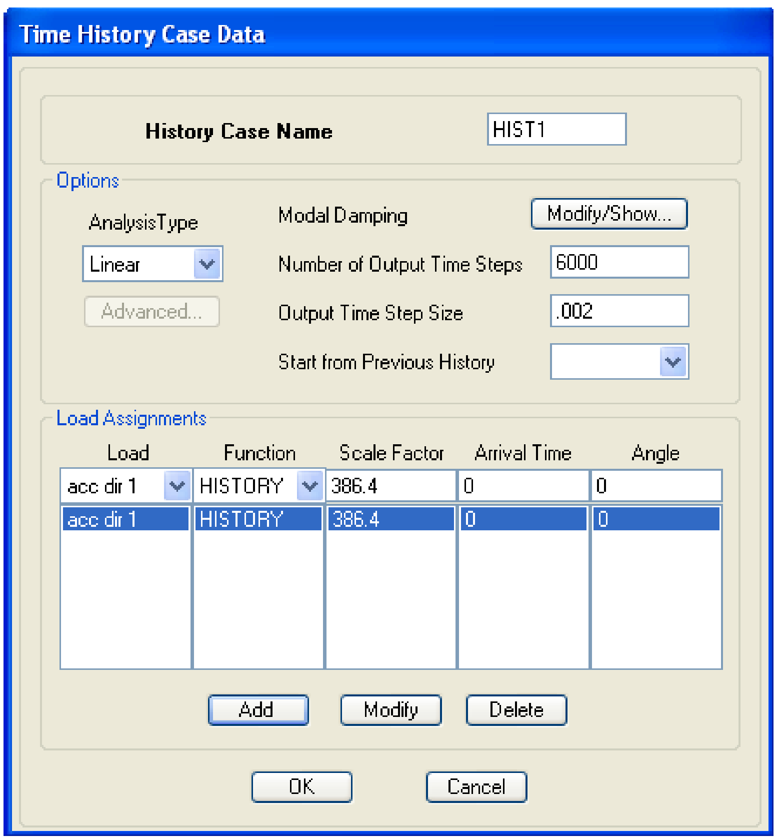



**Figure II.6:** Time History Case Definition and time history cases. If the **Do not Include Special Seismic Design Data** option is selected, the Rho factor and the DL Multiplier are not applied to any design load combinations.

Under the Rho Factor (Reliability Factor based on Redundancy) section, the Program Calculated ETABS calculates the Rho factor in accordance with Section 1617 of the 2000 International Building Code. The automatic calculation of the Rho factor depends on the floor area. ETABS calculates the floor area at each story level by summing the areas of the floor-type area objects at each story level. If no floor type area objects exist at a story level, that story level is ignored when calculating Rho. If no floor type area objects exist at *any* story level, and ETABS is to calculate the Rho factor, Rho is assumed to be equal to 1.5.

**Important Note**: The calculation of the Rho factor also depends on the ratio of the design story shear resisted by the most heavily loaded element in a story divided by the total story shear. This ratio is designated $r_{max}$. The value of $r_{max}$ can only be calculated if there is lateral load in the model. The Rho factor can only be calculated if $r_{max}$ is nonzero. *Thus, the Rho factor is only calculated when there is lateral load present in the model.* Refer to Figure II.7:

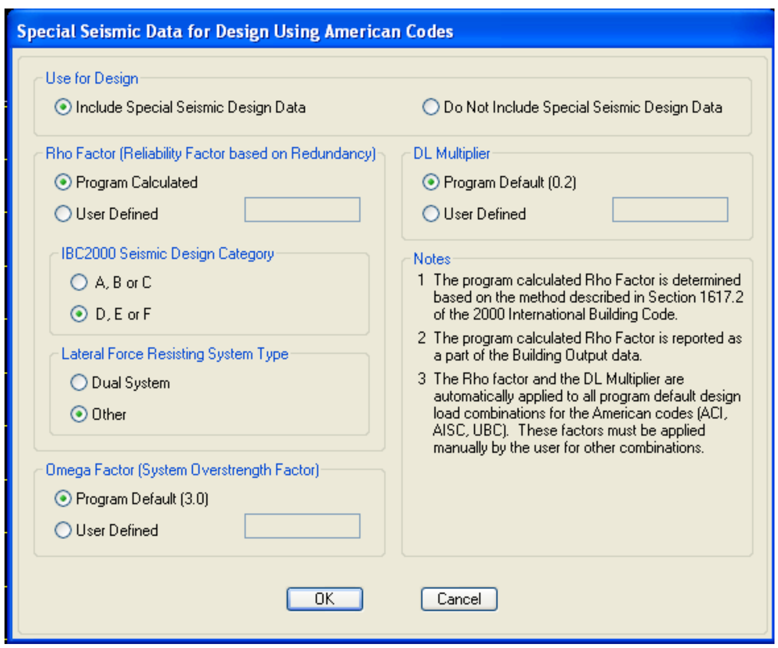



**Figure II.7:** Special Seismic Data for Design

## II.6 Auto-permutation of Wind Directions and Eccentricities

Next, we will define the wind load cases to the model. Perform the same operation as discussed earlier to define a wind load case. This time we will apply a wind load in the Y-direction. Give the load case a name of WY, select a WIND load type and select the ASCE 7-02 code from the pull down menu. Click the Modify Lateral Load button to bring up the menu shown in Figure II.8:

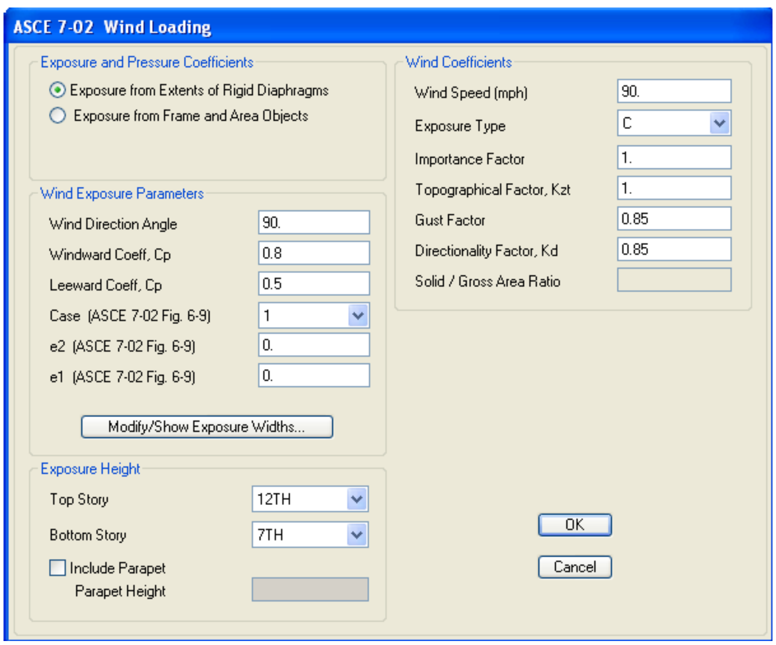



**Figure II.8:** ASCE 7-02 Wind Load Definition

We have defined rigid diaphragms to this model, so select the ‘Exposure from Extents of Rigid Diaphragms’ option. The width of the diaphragm is calculated by the ETABS. To apply the wind load in the Y-direction, enter 90 degrees for the wind direction angle. The windward and leeward coefficients are defined as 0.8 and 0.5 respectively.

A new feature in ETABS allows the user to define one wind load case and the program will automatically create all of the remaining wind load cases. The ASCE 7-02 wind code must be used for this feature to be activated. The e1 and e2 values changes for each subsequent wind load case based on Figure 6.9 as well as the wind direction angles in the ASCE 7-02 code. A total of 12 different wind load cases will be defined. Enter all of the remaining Wind Coefficients shown in Figure II.8.

# PART III

## Steel Frame Design
## Using ANSI/AISC 341-05 Steel Code)

### III.1 Example 1

This Simple Beam example is intended to demonstrate the ETABS AISC 2005 steel design code features. The first example below shows the program checks for various flexure limit states. The Example 1 definitions are described in Figure III.1:

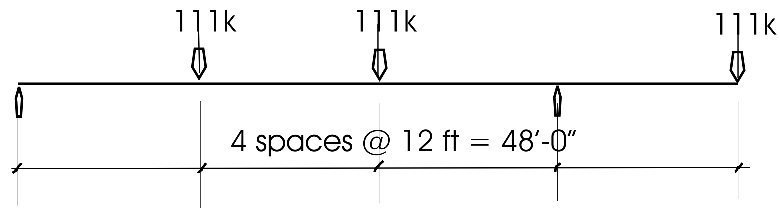



**Figure III.1:** Example 1

The section properties for W30X148 beam shown in Figure III.2:

Reference is made to ASCE 360-05 Specifications, Chapter F, for flexural design. Chapter F is divided into sections F1 through F11 which are referenced in this For clarity, the SAP2000 and ETABS capabilities are discussed with reference to the subject sections. The ASCE 360-05 Table F1.1 is presented in Figure III.3 and includes the subjects sections.

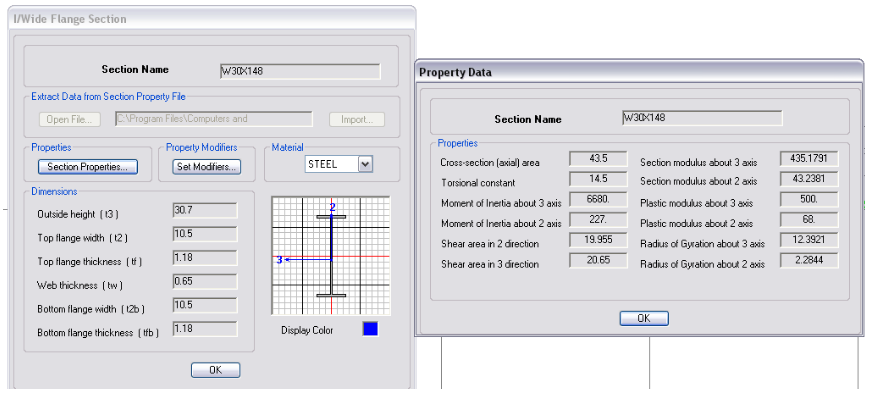



**Figure III.2:** W30X148 Section Properties

### III.1.1 Check for Compact Section Requirements

Check for compact section requirements:

$$\frac{b_f}{2t_f} = \frac{10.5}{2(1.18)} = 4.45 \leq \lambda_p = 0.38\sqrt{\frac{E}{F_y}} = 9.15 \tag{III.1}$$

$$\frac{h}{t_w} = \frac{26.5}{0.65} = 40.8 < \lambda_p = 3.76\sqrt{\frac{E}{F_y}} = 90.55 \tag{III.2}$$

Since $\lambda$ is less then $\lambda_p$ for the flange and web the section is compact. The limit state for local buckling will not control.

### III.1.2 Check The Limit State For Flexural Yielding

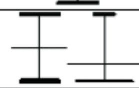
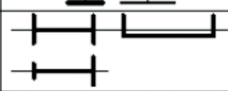
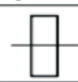
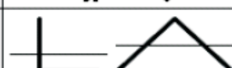
$M_u \leq \phi M_n$ (in LRFD) and $M \leq M_n/\Omega_b$ in (ASD)

$$M_n = F_y Z_x,$$

$$M_n = 50ksi\text{X}500in^3\text{=}25,000kip - in \tag{III.3}$$

Check for lateral torsional buckling where $L_b$ is the unbraced length, $L_p$ and $L_r$ are limiting lengths:

**TABLE User Note F1.1**
**Selection Table for the Application of Chapter F Sections**

| Section in Chapter F | Cross Section | Flange Slenderness | Web Slenderness | Limit States |
|---|---|---|---|---|
| F2 |  | C | C | Y, LTB |
| F3 |  | NC, S | C | LTB, FLB |
| F4 |  | C, NC, S | NC | Y, LTB, FLB, TFY |
| F5 |  | C, NC, S | S | Y, LTB, FLB, TFY |
| F6 |  | C, NC, S | N/A | Y, FLB |
| F7 |  | C, NC, S | C, NC | Y, FLB, WLB |
| F8 |  | N/A | N/A | Y, LB |
| F9 |  | C, NC, S | N/A | Y, LTB, FLB |
| F10 |  | N/A | N/A | Y, LTB, LLB |
| F11 |  | N/A | N/A | Y, LTB |
| F12 | Unsymmetrical shapes | N/A | N/A | All limit states |

Y = yielding, LTB = lateral-torsional buckling, FLB = flange local buckling, WLB = web local buckling, TFY = tension flange yielding, LLB = leg local buckling, LB = local buckling, C = compact, NC = noncompact, S = slender

**Figure III.3:** ASCE 360-05, Table F1.1

a. When $L_b \leq L_p$

$$M_n = M_p \qquad \text{(ASCE F2-1)}$$

b. When $L_p < L_b \leq L_r$

$$M_n = C_b \left[ M_p - (M_p - 0.7 F_y S_{33}) \left( \frac{L_b - L_p}{L_r - L_p} \right) \right] \leq M_p \qquad \text{(ASCE F2-2)}$$

c. When $L_b > L_r$

$$M_n = C_b S_{33} \frac{\pi^2 E}{\left(\frac{L_b}{r_{ts}}\right)^2} \sqrt{1 + 0.078 \frac{Jc}{S_x h_o \left(\frac{L_b}{r_{ts}}\right)^2}} \leq M_p \qquad \text{(ASCE F2-3,4)}$$

Where,

$$L_p = 1.76 r_y \sqrt{\frac{E}{F_y}} \qquad \text{(ASCE F2-5)}$$

$$L_r = 1.95 r_{ts} \frac{E}{0.7 F_y} \sqrt{\frac{Jc}{S_{33} h_o}} \sqrt{1 + \sqrt{1 + 6.76 \left( \frac{0.7 F_y}{E} \frac{S_{33} h_o}{Jc} \right)^2}} \qquad \text{(ASCE F2-6)}$$

For this example,

1. $L_b = 12 ft (144 in)$
2. $L_p = 1.76 r_y \sqrt{\frac{E}{F_y}}$, which gives, $L_p = 96.6$
3. $L_r = 1.95 r_{ts} \frac{E}{0.7 F_y} \sqrt{\frac{Jc}{S_{33} h_o}} \sqrt{1 + \sqrt{1 + 6.76 \left( \frac{0.7 F_y}{E} \frac{S_{33} h_o}{Jc} \right)^2}}$

where,

$$r_{ts} = \frac{b_f}{\sqrt{12 \left(1 + \frac{1}{6} \frac{h t_w}{b_f t_f}\right)}} = \frac{10.5}{\sqrt{12 \left(1 + \frac{1}{6} \frac{26.5(0.65)}{10.5(1.18)}\right)}} = 2.73 in \qquad \text{(III.4)}$$

$$J = 14.5 in^4 \text{ (from section property data)} \qquad \text{(III.5)}$$

$$h_o = 30.7 - 1.18 = 29.52 in \qquad \text{(III.6)}$$

$$c = 1.0 \text{ (for doubly symmetric I-shape)} \tag{III.7}$$

$$L_r = 1.95(2.73)\frac{29500}{0.7(50)}\sqrt{\frac{14.1(1.0)}{436(29.52)}}\sqrt{1+\sqrt{1+6.76\left(\frac{0.7(50)}{29500}\frac{436(14.1)}{14.1(1.0)}\right)^2}} \tag{III.8}$$

$$L_r = 293in \tag{III.9}$$

Since $L_p < L_b \leq L_r$,

$$M_n = C_b\left[25000 - \left(25000 - 0.7(50)436\left(\frac{144-96.6}{293-96.6}\right)\right)\right] \quad \text{(ASCE F2-6)}$$

$$M_n = C_b(21,317)kip - in \tag{III.10}$$

where,

$$C_b = \frac{12.5M_{max}}{2.5M_{max}+3M_A+4M_B+3M_C}R_m \leq 3.0 \quad \text{(ASCE F1-1)}$$

where:

$M_{max}$ is the absolute value of the maximum moment in the unbraced segment and MA, MB and MC are the absolute values of the moment at quarter, half, and three-quarter points of the unbraced segment, respectively. $R_m$is the cross-section monosymmetry parameter that is given by:

1.0 = for doubly symmetric sections

1.0 = for singly symmetric members with single curvature

$0.5 + 2(\frac{I_{yc}}{I_y})^2$ for reverse bending

Cb should be taken as 1.0 for cantilevers. However, the SAP2000 and ETABS are unable to detect whether the member is a cantilever. The user should overwrite Cb for cantilevers. The program also defaults Cb to 1.0 if the minor un-braced length, l22, is redefined to be more than the length of the member by either the user or the program, i.e., if the unbraced length is longer than the member length. The user can overwrite the value of Cb for any member. The nominal bending strength depends on the following criteria: the geometric shape of the cross-section; the axis of bending; the compactness of the section; and a slenderness parameter for lateral-torsional buckling. The nominal bending strength is the minimum value obtained according to the limit states of yielding, lateral-torsional buckling, flange local buckling, and web local buckling.

Reference is made to ASCE 360-05 Specifications, Chapter F, for flexural design. Chapter F is divided into sections F1 through F11 which are referenced in this For clarity, the SAP2000 and ETABS capabilities are discussed with reference to the subject sections. The ASCE 360-05 Table F1.1 is presented below and includes the subjects sections

For this example:

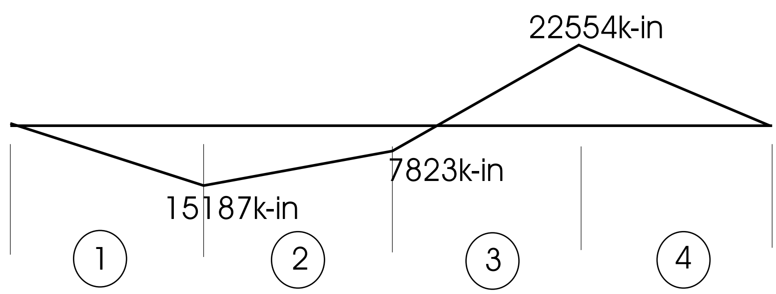



**Figure III.4:** Example 2

- Segment 1:

$$C_b = \frac{12.5M_{max}}{2.5M_{max} + 3M_A + 4M_B + 3M_C} R_m \leq 3.0 \quad \text{(III.11)}$$

$$C_b = \frac{12.5(15187)}{2.5(15187) + 3(3797) + 4(7594) + 3(11390)} 1.0 \leq 3.0 \quad \text{(III.12)}$$

$$C_b = 1.67 \quad \text{(III.13)}$$

- Segment 2:

$$C_b = \frac{12.5(15187)}{2.5(15187) + 3(13346) + 4(11505) + 3(9664)} 1.0 \leq 3.0 \quad \text{(III.14)}$$

$$C_b = 1.24 \quad \text{(III.15)}$$

- Segment 3:

$$C_b = \frac{12.5(22554)}{2.5(22554) + 3(229) + 4(7365) + 3(14960)} 1.0 \leq 3.0 \quad \text{(III.16)}$$

$$C_b = 2.15 \quad \text{(III.17)}$$

- Segment 4:

$$C_b = \frac{12.5(22554)}{2.5(22554) + 3(5638) + 4(11277) + 3(16916)} 1.0 \leq 3.0 \quad \text{(III.18)}$$

$$C_b = 1.67 \quad \text{(III.19)}$$

For this example, $C_b = 2.15$

$$M_n = 2.15(21,317) = 45,831 kip - in > M_p = 25,000 kip - in \quad \text{(III.20)}$$

therefore,

$$M_n = M_p = 25,000 kip - in \quad \text{(III.21)}$$

### III.1.3 Shear Check

SAP2000 and ETABS design for shear without consideration of the post-buckling *tension field action*. The design equations for shear are as follows:

$$V_n = 0.6 F_y A_w C_v$$

Where

$$V_u \leq \phi_v V_n \text{ (in LRFD) and } V \leq V_n/\Omega_v \text{ in (ASD)}$$

And,

$V_u$= factored applied shear force

V= service(un-factored) applied shear force

$A_w$ = area of web

$\phi_v$ = 1.00 (for LRFD)

$\Omega_v$ = 1.50 (for ASD)

$C_V$ = shear reduction factor

1. For $h/t_w \leq 1.10\sqrt{\frac{k_v E}{F_y}}$

$$C_v = 1.0 \qquad \text{(ASCE G2-3)}$$

2. For $1.10\sqrt{k_v E/F_y} < h/t_w \leq 1.10\sqrt{k_v E/F_y}$

$$C_v = \frac{1.10\sqrt{k_v E/F_y}}{h/t_w} \qquad \text{(ASCE G2-4)}$$

3. For $h/t_w > 1.37\sqrt{k_v E/F_y}$

$$C_v = \frac{1.51 E k_v}{(h/t_w)^2 F_y} \qquad \text{(ASCE G2-5)}$$

For Example 1, the maximum shear, $V_u = 211.4 kips$.

$$h/t_w \leq 1.10\sqrt{\frac{k_v E}{F_y}} \text{ where } k_v = 5.0 \text{ when } h/t_w < 260 \qquad \text{(III.22)}$$

$$26.5/0.65 = 40.8 \leq 53.9 \text{ therefore, } C_v = 1.0 \qquad \text{(III.23)}$$

$$V_n = 0.6 F_y A_w C_v = 0.6(50)0.65(1.0) = 598.65 kips \qquad \text{(III.24)}$$

ETABS reports the demand/capacity ratio as,

$$V_u/\phi V_n = 211.3/598.65 = 0.353 \qquad \text{(III.25)}$$

## III.2 Example 2

Using the same span and loading as defined for Example 1, the demand/capacity ratios are calculated for a built-up section in lieu of a wide flange section. The built-up section is defined in Figure III.5:

### III.2.1 Check for Compact Section Requirements

$$\frac{b_{f_{top}}}{2t_f} = \frac{12}{2(1.5)} = 4.00 \leq \lambda_p = 0.38\sqrt{\frac{E}{F_y}} = 9.15 \qquad \text{(III.26)}$$

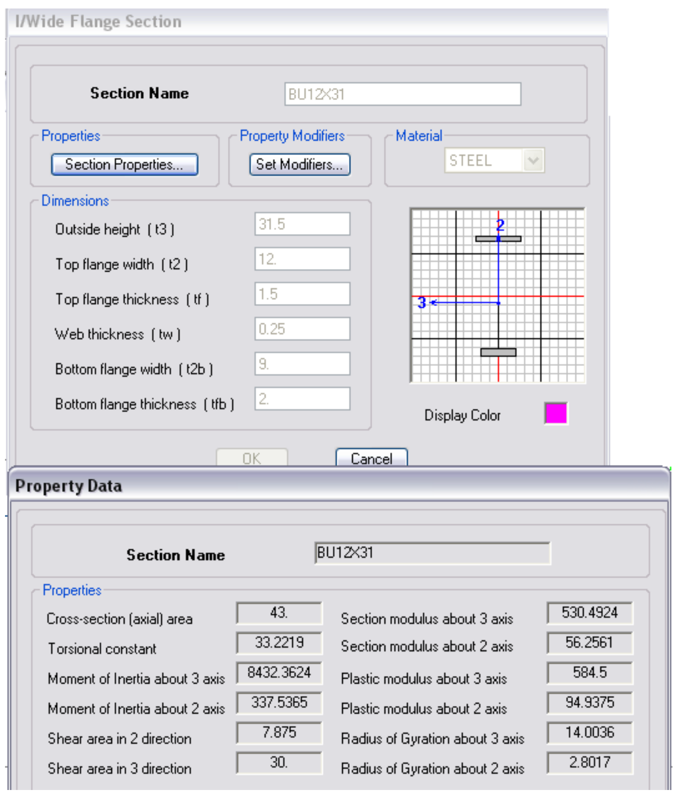



**Figure III.5:** Built-Up Section Properties

$$\frac{b_{f_{bot}}}{2t_f} = \frac{9}{2(2.0)} = 2.25 \leq \lambda_p = 0.38\sqrt{\frac{E}{F_y}} = 9.15 \tag{III.27}$$

$$\frac{h}{t_w} = \frac{28.0}{0.25} = 112 > \lambda_p = \frac{\frac{h_c}{h_p}\sqrt{\frac{E}{F_y}}}{\left(0.54\frac{M_p}{M_y} - 0.09\right)^2} = 94.4 \tag{III.28}$$

$$\frac{h}{t_w} = \frac{28.0}{0.25} = 112 < \lambda_r = 5.70\sqrt{\frac{E}{F_y}} \tag{III.29}$$

Since $\lambda$ is greater than $\lambda_p$ for the web, the section is non-compact. The limit state for local buckling will may control.

### III.2.2 Check Limit State of Yielding

For the maximum negative moment causing compression in the bottom flange, $M_u \leq \phi M_n$, and, $M_n = R_{pc}F_yS_{33c}$

where, $R_{pc}$ is the web plastification factor given as,

$$R_{pc} = \left[\frac{M_p}{M_{yc}} - \left(\frac{M_p}{M_yc} - 1\right)\left(\frac{\lambda - \lambda_{pw}}{\lambda_{rw} - \lambda_{pw}}\right)\right] < \frac{M_p}{M_{yc}} \tag{III.30}$$

where,

$$M_{yc} = F_y(S_{x_{bot}}) = 26,525 kip - in \tag{III.31}$$

$$M_p = Z_{x_{bot}}(F_y) = 50(584.5) = 29,225 kip - in \tag{III.32}$$

$$\text{thus, } R_{pc} = 1.06 \tag{III.33}$$

$$M_p = R_{pc}Z_{x_{bot}}(F_y) = 1.06(50)(530.5) = 29,225 kip - in \tag{III.34}$$

For the maximum negative moment causing tension in the top flange, the section modulus, $S_{top} > S_{bot}$, therefore $S_{x_{tension}} > S_{x_{comp}}$, and tension yielding will not control.

### III.2.3 Check Lateral Torsional Buckling

For this example,

- $L_b = 12ft(144in)$
- $L_p = 1.76 r_y \sqrt{\frac{E}{F_y}}$, which gives, $1.76(2.80)\sqrt{\frac{29500}{50}} = 118.7in$
- $L_r = 1.95 r_{ts} \frac{E}{0.7F_y} \sqrt{\frac{Jc}{S_{33}h_o}} \sqrt{1 + \sqrt{1 + 6.76\left(\frac{0.7F_y}{E}\frac{S_{33}h_o}{Jc}\right)^2}}$

  where

$$r_{ts,top} = \frac{b_f, top}{\sqrt{12\left(1 + \frac{1}{6}\frac{ht_w}{b_f t_f}\right)}} = \frac{12}{\sqrt{12\left(1 + \frac{1}{6}\frac{28(0.25)}{12(1.5)}\right)}} = 3.35in \quad \text{(III.35)}$$

$$r_{ts,bot} = \frac{b_f, bot}{\sqrt{12\left(1 + \frac{1}{6}\frac{ht_w}{b_f t_f}\right)}} = \frac{9}{\sqrt{12\left(1 + \frac{1}{6}\frac{28(0.25)}{9(2.0)}\right)}} = 2.52in \quad \text{(III.36)}$$

$$J = 37.65in^4 \text{ (from section property data)} \quad \text{(III.37)}$$

$$h_o = 31.5 - 1.0 - 0.75 = 29.95in \quad \text{(III.38)}$$

$$c = 1.0 \quad \text{(III.39)}$$

$$L_r, top = 1.95(3.35)\frac{29500}{0.7(50)}\sqrt{\frac{37.65(1.0)}{540.3(29.75)}}\sqrt{1 + \sqrt{1 + 6.76\left(\frac{0.7(50)}{29500}\frac{540.3(29.55)}{37.65(1.0)}\right)^2}} \quad \text{(III.40)}$$

$$L_r, top = 433in \quad \text{(III.41)}$$

$$L_r, bot = 1.95(2.52)\frac{29500}{0.7(50)}\sqrt{\frac{37.65(1.0)}{540.3(29.75)}}\sqrt{1 + \sqrt{1 + 6.76\left(\frac{0.7(50)}{29500}\frac{540.3(29.55)}{37.65(1.0)}\right)^2}} \quad \text{(III.42)}$$

$$L_r, bot = 326in \tag{III.43}$$

Since $L_p < L_b \leq L_r$,

•

$$M_n = \left[R_{pc}M_{yc} - (R_{pc}M_{yc} - F_L S_{xc})\left(\frac{\lambda - \lambda_p}{\lambda_r - \lambda_p}\right)\right] \qquad \text{(ASCE F3-1)}$$

where,

$$F_L = 0.7F_y \text{for } \frac{S_{xt}}{S_{xc}} \geq 0.7 \tag{III.44}$$

$$R_{pc} = \left[\frac{M_p}{M_{yc}} - \left(\frac{M_p}{M_{yc}} - 1\right)\left(\frac{\lambda - \lambda_{pw}}{\lambda_{rw} - \lambda_{pw}}\right)\right] \leq \frac{M_p}{M_{yc}} \tag{III.45}$$

$$M_{yc} = F_y S_{x,top} = 27,015kip - in \tag{III.46}$$

$$M_p = 50(584.5) = 29,225kip - in \tag{III.47}$$

• Segment 1:

$$C_b = \frac{12.5M_{max}}{2.5M_{max} + 3M_A + 4M_B + 3M_C} R_m \leq 3.0 \tag{III.48}$$

$$C_b = 1.67 \text{ (See Example 1)} \tag{III.49}$$

$$M_n = \left[R_{pc}M_{yc} - (R_{pc}M_{yc} - F_L S_{xc})\left(\frac{\lambda - \lambda_p}{\lambda_r - \lambda_p}\right)\right] \tag{III.50}$$

$$M_n = C_b\left[1.06(27,015) - (1.06(27015) - 0.7(50)(540.3))\left(\frac{144 - 118.7}{433 - 118.7}\right)\right]$$

(III.51)

$$M_n = 46,515k - in \text{ ,but ,} M_n \leq R_{pc}M_{yc} = 1.06(50)540.3 = 28,635k - in \quad \text{(III.52)}$$

$$M_n = 28,635k - in \quad \text{(III.53)}$$

- Segment 2:

$$C_b = 1.24 \text{ See Example 1} \quad \text{(III.54)}$$

$$M_n = C_p\left[R_{pc}M_{yc} - (R_{pc}M_{yc} - F_LS_{xc})\left(\frac{\lambda - \lambda_p}{\lambda_r - \lambda_p}\right)\right] \quad \text{(III.55)}$$

$$M_n = 1.24\left[1.06(27,015) - (1.06(27015) - 0.7(50)(540.3))\left(\frac{144 - 118.7}{433 - 118.7}\right)\right] \quad \text{(III.56)}$$

$$M_n = 1.24(27,853) = 34,538k - in \text{ ,but ,} M_n \leq R_{pc}M_{yc} = 28,635k - in \quad \text{(III.57)}$$

- Segment 3:

$$C_b = \frac{12.5(22554)}{2.5(22554) + 3(229) + 4(7365) + 3(14960)}R_m \quad \text{(III.58)}$$

$$C_b = 2.15R_m \quad \text{(III.59)}$$

where,

$$R_m = 0.5 + 2\left(\frac{I_{yc}}{I_y}\right)^2 = 0.5 + 2\left(\frac{2(9)^3/12}{337.5}\right)^2 = 0.7592 \quad \text{(III.60)}$$

thus,

$$C_b = 2.15(0.7592) = 1.62 \quad \text{(III.61)}$$

$$M_{yc} = 50(530.5) = 26,525k - in \tag{III.62}$$

$$M_n = 1.62\left[1.06(26,525) - (1.06(26,525) - 0.7(50)(530.5))(\frac{144 - 118.7}{326 - 118.7})\right] \tag{III.63}$$

$$M_n = 43,660k - in \text{ ,but } ,M_n \leq R_{pc}M_{yc} = 28,611k - in \tag{III.64}$$

- Segment 4:

$$C_b = \frac{12.5(22554)}{2.5(22554) + 3(5638) + 4(11277) + 3(16916)}1.0 \tag{III.65}$$

$$C_b = 1.67 \tag{III.66}$$

$$M_n = 1.67[1.06(26,525) - (1.06(26,525) - 0.7(50)(530.5))(\frac{144 - 118.7}{326 - 118.7})]] \tag{III.67}$$

$$M_n = 45,008k - in \text{ ,but } ,M_n \leq R_{pc}M_{yc} = 28,611k - in \tag{III.68}$$

### III.2.4 Shear Check

For Example 2, the maximum shear, $V_u = 211.4kips$.

1. For $h/t_w \leq 1.10\sqrt{\frac{k_vE}{F_y}}$

$$C_v = 1.0 \qquad \text{(ASCE G2-3)}$$

2. For $1.10\sqrt{k_vE/F_y} < h/t_w \leq 1.10\sqrt{k_vE/F_y}$

$$C_v = \frac{1.10\sqrt{k_vE/F_y}}{h/t_w} \qquad \text{(ASCE G2-4)}$$

3. For $h/t_w > 1.37\sqrt{k_vE/F_y}$

$$C_v = \frac{1.51Ek_v}{(h/t_w)^2F_y} \tag{III.69}$$

Since $h/t_w = 28.0/0.25 = 112$, $k_v = 5.0$, and,

$$C_v = \frac{1.51Ek_v}{(h/t_w)^2F_y} = \frac{1.51(29500)5.0}{(112)^2 50} \tag{III.70}$$

$$C_v = 0.349 \tag{III.71}$$

$$V_n = 0.6F_yA_wC_v = 0.6(50)0.25(31.5)(0.349) = 82.45kips \tag{III.72}$$

ETABS reports the demand/capacity ratio as,

$$V_u/\phi V_n = 211.3/82.45 = 2.56 \tag{III.73}$$

## III.3 Example 3

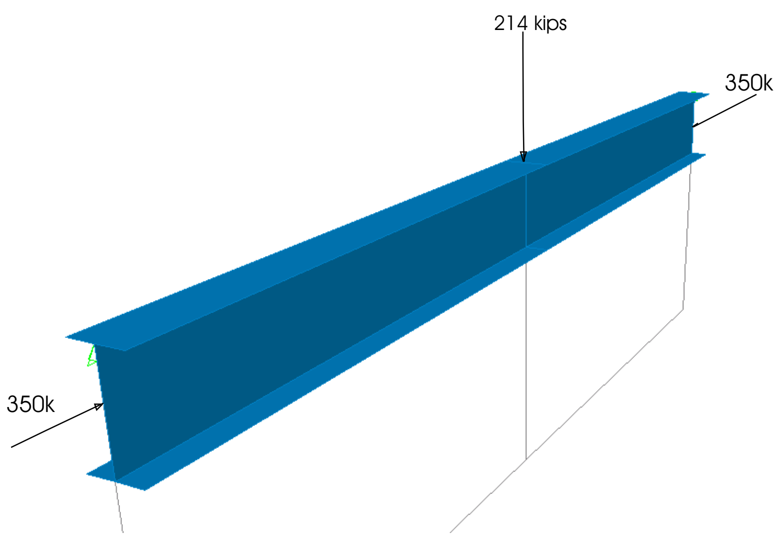



**Figure III.6:** Example 3

The beam/column above is to be designed for the dead loads shown. The member spans a total of 40 feet with a midspan load of 214 kips and an axial load of 350 kips.

For this example, the member is laterally braced at midspan. The demand/capacity ratio for the subject member is found as follows:

The member section properties are:

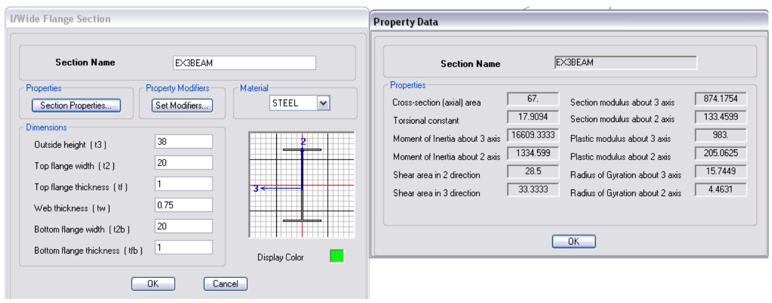



**Figure III.7:** Built-Up Section Properties

The warping constant, $C_w = I_y h^2/4 = 456,500 in^6$

### III.3.1 Check Flexural Buckling

The limit state for flexural buckling is checked as,

$$P_n = F_{cr} A_g$$

Where,

$$P_u \leq \phi_c P_n \text{ (in LRFD) and } P \leq P_n/\Omega_c \text{ in (ASD)}$$

And,

$P_u$= factored applied compressive force

P= service(un-factored) applied compressive force

$A_g$ = gross area of member

$\phi_c$ = 0.90 (for LRFD)

$\Omega_v$ = 1.67 (for ASD)

$F_{cr}$ = flexural buckling stress, determined as shown below,

a. When $\frac{kL}{r} \leq 4.71\sqrt{\frac{E}{F_y}}$

$$F_{cr} = [0.658^{\frac{F_y}{F_e}}]F_y \qquad \text{(ASCE E3-2)}$$

b. When $\frac{kL}{r} > 4.71\sqrt{\frac{E}{F_y}}$

$$F_{cr} = 0.877 F_e \qquad \text{(ASCE E3-3)}$$

where, $F_e$ = elastic critical buckling stress,

$$F_e = \frac{\pi^2 E}{(\frac{KL}{r})^2} \qquad \text{(ASCE E3-4)}$$

For this example,

$$kl/r_x = 1.0(40x12)/15.74 = 30.5 \qquad \text{(III.74)}$$

$$kl/r_y = 1.0(20x12)/4.46 = 53.8\text{——controls} \qquad \text{(III.75)}$$

Since $kl/r_y = 53.8 < 4.71\sqrt{\frac{E}{F_y}} = 113$, use, $F_{cr} = [0.658^{\frac{F_y}{F_e}}]F_y$

$$F_e = \frac{\pi^2 E}{(\frac{KL}{r})^2} = \pi^2 29500/(53.8)^2 = 99ksi \qquad \text{(III.76)}$$

$$F_{cr} = [0.658^{\frac{F_y}{F_e}}]F_y = [0.658^{\frac{50}{99}}]50 = 40.47ksi \qquad \text{(III.77)}$$

$$P_u = \phi F_{cr} A_g = .9(40.74)(67.0) = 2440kips \qquad \text{(III.78)}$$

## III.3.2 Check Torsional and Flexural-Torsional Buckling

$F_{cr}$, is determined using ASCE 360-05 E3-2 and E3-3 and $F_e$, is determined as follows:

For Doubly Symmetric members

$$F_e = \left[\frac{\pi^2 E C_w}{(K_z L)^2} + GJ\right]\left(\frac{1}{I_{22} + I_{33}}\right) \qquad \text{(ASCE E4-4)}$$

$$F_e = \left[\frac{\pi^2 29500(456,500)}{(1.0(40)12)^2} + (11,200(18.4))\right]\left(\frac{1}{16609+1334}\right) = 30.7ksi \tag{III.79}$$

$P_u = \phi F_{cr} A_g = .9(30.7)(67.0) = 1848kips$, $P_u$ is controlled by Flexural-Torsional Buckling (III.80)

### III.3.3 Check Limits States for Flexure

$$\frac{b_f}{2t_f} = \frac{20}{2(1.0)} = 10.0, \text{ should be} \leq \lambda_p = 0.38\sqrt{\frac{E}{F_y}} = 9.15 \tag{III.81}$$

Therefore, local buckling can occur and $M_p$ must be reduced.

$$\lambda_r = 0.95\sqrt{\frac{k_c E}{F_L}} = 0.95\sqrt{\frac{0.577(29500)}{35}} = 20.95 \tag{III.82}$$

Where,

$$k_c = 4/\sqrt{h/t_w} = 4/\sqrt{36/0.75} = .577 \tag{III.83}$$

$$F_L = 0.7F_y = 0.7(50) = 35.0ksi \tag{III.84}$$

Therefore, local buckling can occur and $M_p$ must be reduced.

$$\frac{h}{t_w} = \frac{36.0}{0.75} = 48.0 < \lambda_p = 3.76\sqrt{(E/F_y} = 90 \tag{III.85}$$

$$M_n = \left[R_{pc}M_{yc} - (R_{pc}M_{yc} - F_L S_{xc})\left(\frac{\lambda - \lambda_p}{\lambda_r - \lambda_p}\right)\right] \tag{III.86}$$

where,

$$M_p = 50(983) = 49,150k - in \tag{III.87}$$

$$M_{yc} = 50(874) = 43,700k - in \tag{III.88}$$

$$R_{pc} = M_p/M_{yc} = \frac{49,150}{43,700} = 1.12\text{, is the web plastification factor} \tag{III.89}$$

$$M_{yc} = F_y(S_x) = 50(874)43,700kip - in \tag{III.90}$$

$$M_p = Z_{x_{bot}}(F_y) = 50(983) = 49,152kip - in \tag{III.91}$$

Thus,

$$M_n = \left[1.12(43,700) - (1.12(43,700) - 0.7(50)(874))\left(\frac{10.0 - 9.5}{20.95 - 9.5}\right)\right] \tag{III.92}$$

$$M_n = 48,142k - in \tag{III.93}$$

### III.3.4 Combined Loads

$$\frac{P_r}{P_c} + \frac{8}{9}\left(\frac{M_{rx}}{M_{cx}} + \frac{M_{ry}}{M_{cy}}\right) \leq 1.0 \tag{III.94}$$

$$\frac{490}{1848} + \frac{8}{9}\left(\frac{35,952}{\phi 48,142}\right) = 1.003 \tag{III.95}$$

# PART IV

## Detailing of Concrete Foundations

### IV.1 CSI Detailer Add-On Module

CSiDETAILER is a user friendly detailing and drawing program for preparing engineering drawings of concrete and steel structures using analysis and design output from ETABS and SAFE. The program performs the entire detailing operation using preferences that may be set by the user and the relevant requirements of the building codes and detailing standards. Additional parameters can be specified to customize the output to meet other requirements.

**Note:** CSiDETAILER generates drawings using data from ETABS and SAFE. The user is responsible for thoroughly examining the CSiDETAILER-generated drawings and, if necessary, refining them. To facilitate refinement and editing, various tools are provided in CSiDETAILER. Moreover the program can generate DXF and DWG files of the drawings. Those drawings then can be opened and edited using AutoCAD, or other CAD software. CSiDETAILER prepares detailed engineering drawings in accordance with detailing codes, such as ACI 315-99, or by preferences set by the user. Those preferences may comply with building codes as well as be customized to meet additional requirements.

### IV.2 Drawing and Detailing of Complex Slabs

As indicated previously, CSiDETAILER generates detailed engineering drawing based on the output from ETABS and SAFE. Thus, to begin the detailing process, prepare a structural model. The concrete outlines and the layouts of the structural members can be viewed at any time using the **Detailing** > **Start Detailing** command. However, to obtain view, the reinforcing details of a member, the analysis and design has to be carried out first. Refer to Figure IV.1 to see the initial setup screen for CSiDETAILER.

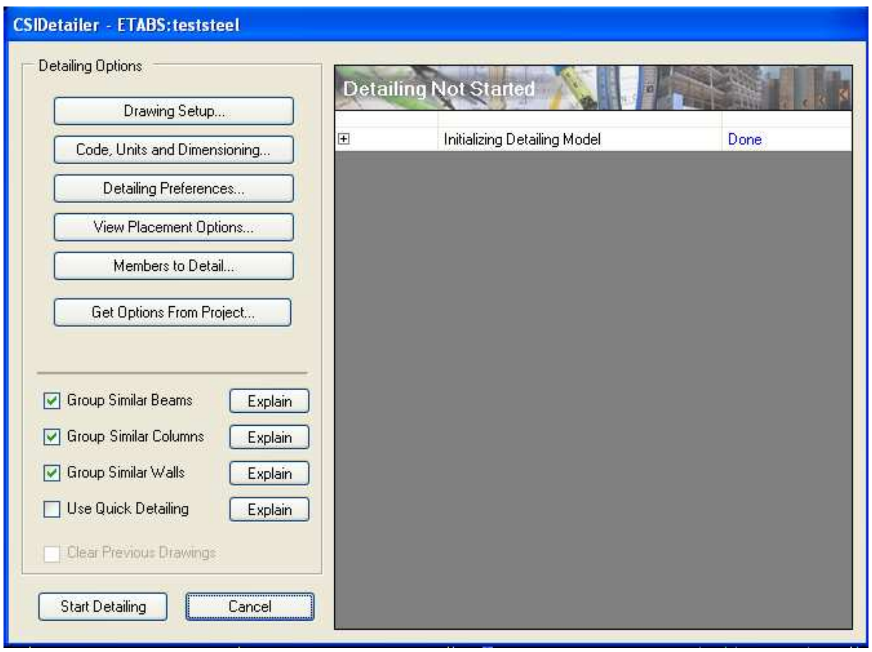



**Figure IV.1:** CSiDETAILER Setup

In this menu, the user can define all of the detailing options. The Setup drawing button allows the user to specify the drawing units (e.g., ANSI Engineering, ANSI Architecture, standard), drawing size, drawing scale, text symbol and gap size, margins, and title block. If the drawing size or scales have been changed, click the Match Default Scales to Drawing Size button to quickly and accurately adjust the options related to text and symbol size for consistent output. The Specify codes, units, and dimensioning button specify standards, dimension units, tolerances, material quantity units, and formatting.

Specify the parameters for detailing objects. The Detailing Preferences button accesses the *Detailing Preferences* form where the user can make changes to column, wall, footing and mat preferences. Use the form to assign detailing options for the various design objects.

## IV.3 Creation of Slabs and Sections

Set various preferences using this form (Step 3) and click **Start Detailing**. As the program details the model, progress will be shown on the *Detailing Status* form. When the detailing is complete, a "Detailing Complete" message will be displayed on the form. The right panel of the screen will show the list (collapsible list) of all items detailed for the model. Scroll the generated report to view errors and warnings reported by the program. Once finished click the **Show Drawings** button and the program will automatically generate typical drawings and add them to the project file. Browse the Drawings by clicking on the drawing title on the *Drawing Explorer* panel. When expanded, each item on the panel will show the names of views available on each drawing. See Figure IV.2:

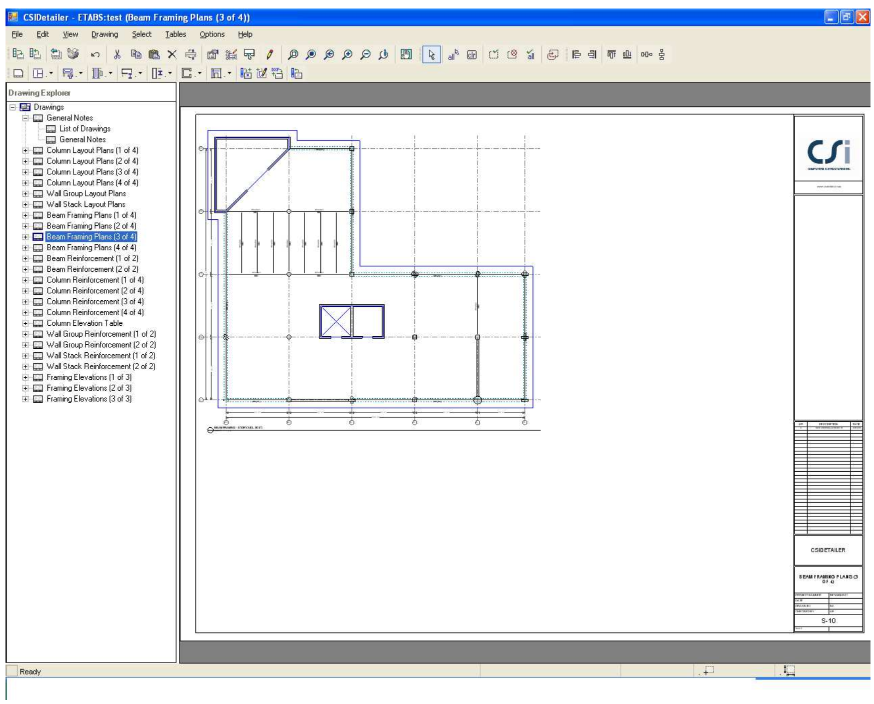



**Figure IV.2:** CSiDETAILER Beam Framing Plan

CSiDETAILER provides extensible and easy-to-use tools for managing views and drawings. With the comprehensible interface of the program, users can easily add new drawings to their projects, add, move, delete, and arrange views on drawings, and save views and drawings to a library. See Figure IV.3 for a beam reinforcing example.

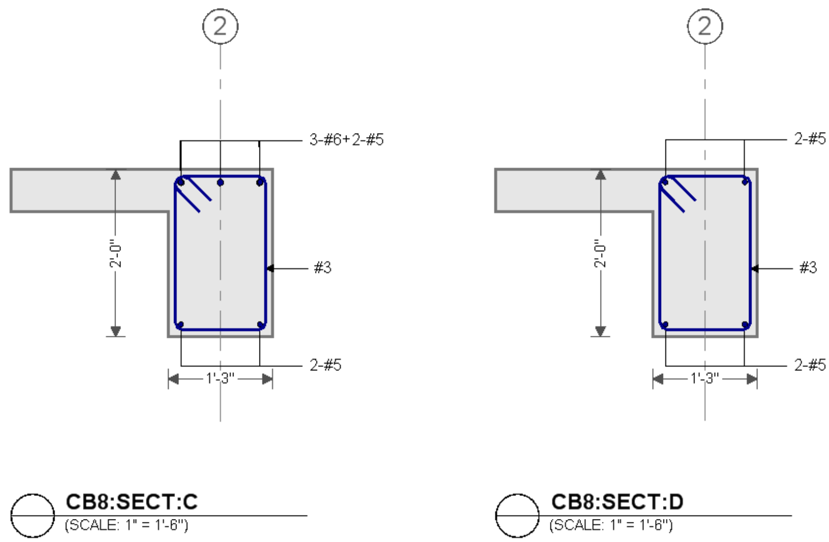



**Figure IV.3:** CSiDETAILER Beam Reinforcing Section

## IV.4 Reinforcing Details and Bar Schedules

Reinforcement can be modified graphically using the *Check and Edit Reinforcement* form. Click the **Edit** > **Check and Edit Reinforcement** > **Mat Reinforcement** command to display the *Check and Edit Reinforcement* form shown in Figure IV.4.

Select the design strip for which reinforcement is to be modified by clicking on it. Use the various zooming and panning options to facilitate the selection process.

When the strip is selected, the strip detail area is updated showing the selected strip with all provided reinforcements, rebar calls and extents. Graphical illustrations of the required, provided and minimum reinforcement profiles are also displayed along the length of the strip.

Under the Tables > Rebar Schedule, you see rebar information, including bar marks, bar sizes, horizontal dimensions, cut lengths, total lengths of each bar type and their shapes, codes and graphical displays of all the bars for the detailed objects. The tables can be viewed and verified against the drawings but they cannot be edited. See Figure IV.5 to see this Table:

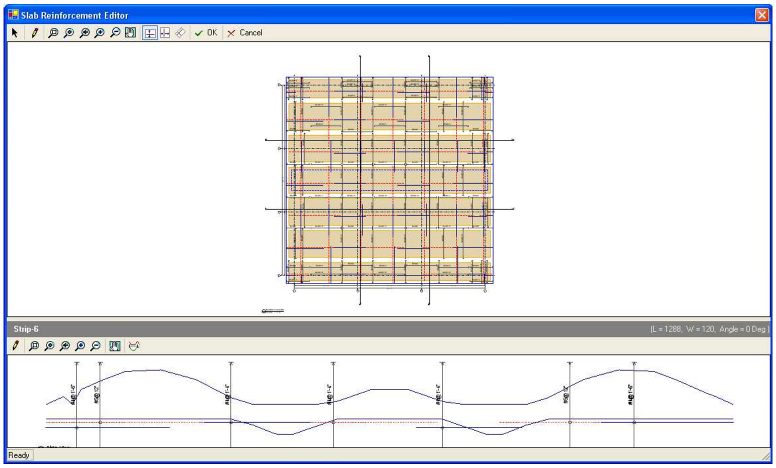
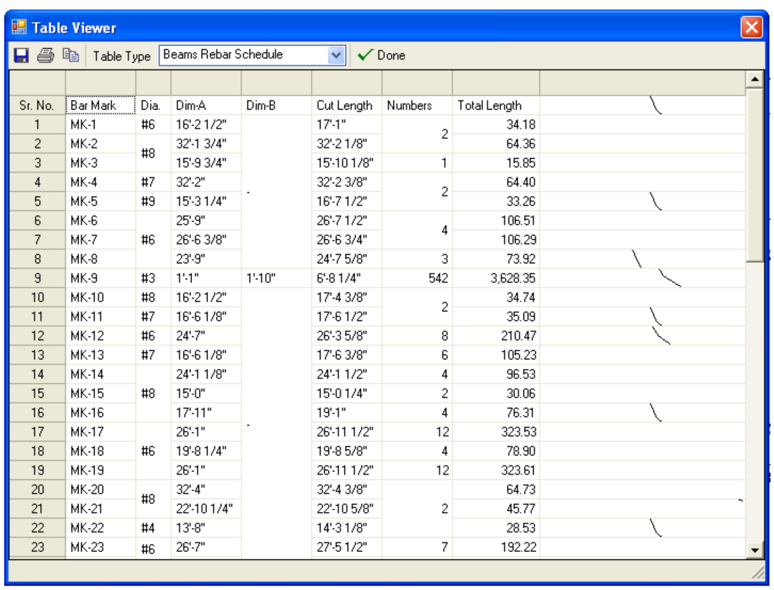



**Figure IV.4:** CSiDETAILER Check/Edit Reinforcement




| Sr. No. | Bar Mark | Dia. | Dim-A | Dim-B | Cut Length | Numbers | Total Length |
|---|---|---|---|---|---|---|---|
| 1 | MK-1 | #6 | 16'-2 1/2" | - | 17'-1" | 2 | 34.18 |
| 2 | MK-2 | #8 | 32'-1 3/4" | | 32'-2 1/8" | | 64.36 |
| 3 | MK-3 | | 15'-9 3/4" | | 15'-10 1/8" | 1 | 15.85 |
| 4 | MK-4 | #7 | 32'-2" | | 32'-2 3/8" | 2 | 64.40 |
| 5 | MK-5 | #9 | 15'-3 1/4" | | 16'-7 1/2" | | 33.26 |
| 6 | MK-6 | #6 | 25'-9" | | 26'-7 1/2" | 4 | 106.51 |
| 7 | MK-7 | | 26'-6 3/8" | | 26'-6 3/4" | | 106.29 |
| 8 | MK-8 | | 23'-9" | | 24'-7 5/8" | 3 | 73.92 |
| 9 | MK-9 | #3 | 1'-1" | 1'-10" | 6'-8 1/4" | 542 | 3,628.35 |
| 10 | MK-10 | #8 | 16'-2 1/2" | - | 17'-4 3/8" | 2 | 34.74 |
| 11 | MK-11 | #7 | 16'-6 1/8" | | 17'-6 1/2" | | 35.09 |
| 12 | MK-12 | #6 | 24'-7" | | 26'-3 5/8" | 8 | 210.47 |
| 13 | MK-13 | #7 | 16'-6 1/8" | | 17'-6 3/8" | 6 | 105.23 |
| 14 | MK-14 | #8 | 24'-1 1/8" | | 24'-1 1/2" | 4 | 96.53 |
| 15 | MK-15 | | 15'-0" | | 15'-0 1/4" | 2 | 30.06 |
| 16 | MK-16 | | 17'-11" | | 19'-1" | 4 | 76.31 |
| 17 | MK-17 | #6 | 26'-1" | | 26'-11 1/2" | 12 | 323.53 |
| 18 | MK-18 | | 19'-8 1/4" | | 19'-8 5/8" | 4 | 78.90 |
| 19 | MK-19 | | 26'-1" | | 26'-11 1/2" | 12 | 323.61 |
| 20 | MK-20 | #8 | 32'-4" | | 32'-4 3/8" | 2 | 64.73 |
| 21 | MK-21 | | 22'-10 1/4" | | 22'-10 5/8" | | 45.77 |
| 22 | MK-22 | #4 | 13'-8" | | 14'-3 1/8" | | 28.53 |
| 23 | MK-23 | #6 | 26'-7" | | 27'-5 1/2" | 7 | 192.22 |

**Figure IV.5:** CSiDETAILER Rebar Schedule

# PART V

## Creation of Output Reports for Submittal

### V.1 Detailed Steel Beam Output

When modeling large structures, it is very easy to for ETABS to create an enormous amount of output. ETABS has a feature that enables the user to select what specific output they need. The user can also select the elements (frames, walls, floor, etc) for which output to print.

We are interested in printing steel frame design output for columns at the base of the structure. First, select a few columns for which you wish to see design output. Go to Print Tables > Steel Frame Design > click on the Output Summary and Selection Only options and click OK. The information shown in Figure V.1 will be printed for each column selected. You can organize output reports for submittal by selecting all controlling concrete sections and printing design output information for each of them.

Click on the Output Summary, Detailed Output, and Selection only buttons. This will print only the design output for the selected shear walls as shown in Figure V.1.

### V.2 Added Design Output to Database

Another way to extract output data from ETABS is to use one of the export options. Again, select the shear walls that you want information for and Go to Export > Save Input/Output as Access Database File. Check the check box associated with an item to include that item in the export. Expand the tree associated with a table type by clicking on the plus (+) symbol that precedes the table type name. Continue expanding the tree until the table appears. Make selections as shown in Figure V.2 and then click the OK

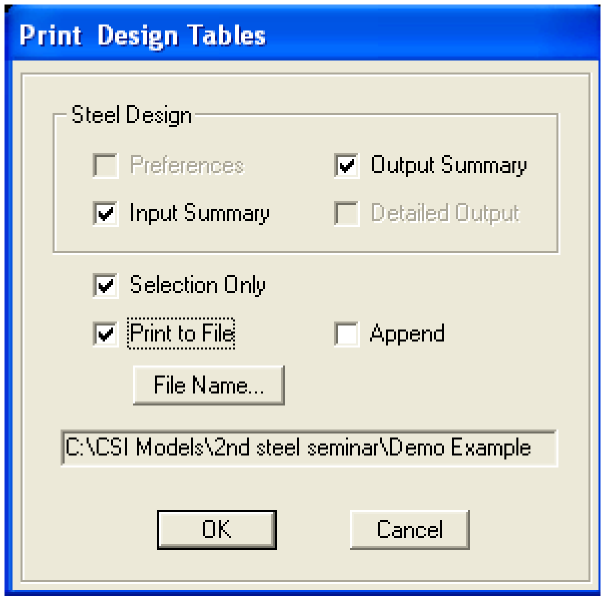



**Figure V.1:** Print Steel Design Tables

button. You will be asked to give the database a filename.

To open the Access Database file, double-click on the file in the saved location. Once the file has been opened, double-click on the Pier Design Forces Table. A spreadsheet will appear that displays all of the design forces (P, V2, V3, M2, M3, etc) in a tidy format. See Figure V.3:

Another way to access output from ETABS is to Click the File menu > Print Tables > Analysis Output command to access the Print Output Tables form. Use the check boxes, drop-down list, edit box, and buttons on the form to specify the type of analysis output to be printed, the sort order, and if data is printed to file, the path and filename for storing the file. *If some objects are selected before executing the File menu > Print Tables > Output command, the printed output will be for the selected objects only.* The Type of Analysis Results check boxes are used to select (or deselect) the type of data to be included in the printed output. If an item is “grayed out” and the check box is unavailable, the item is not included in the model. The Select Loads button accesses the Select Output form. Click on a load to select or deselect it. Click the Clear All button to deselect all previously selected loads. The Select Cuts buttons chooses the section cuts to be included in the printed output. Click on a section cut name to select or deselect it. Click the Clear All button to deselect all previously selected section cuts. The Print to File checkbox prints the specified data to a text file. Use the File Name

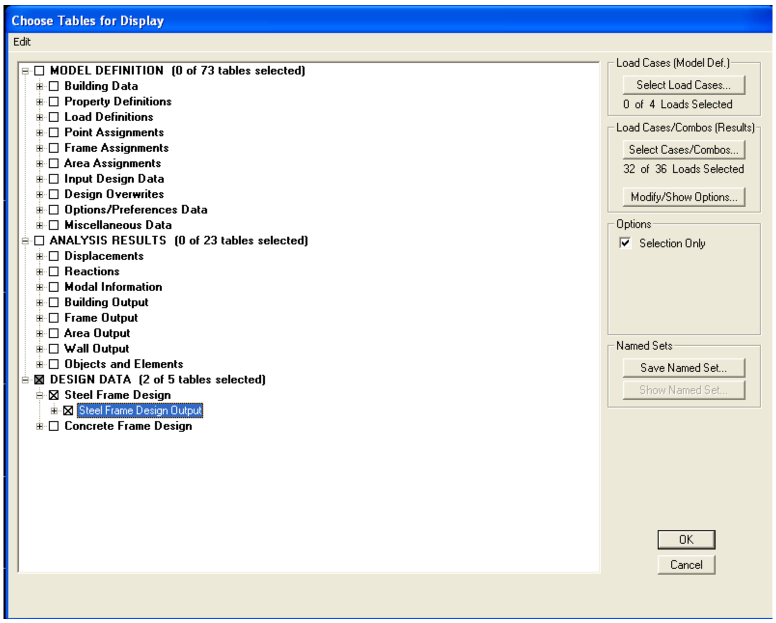
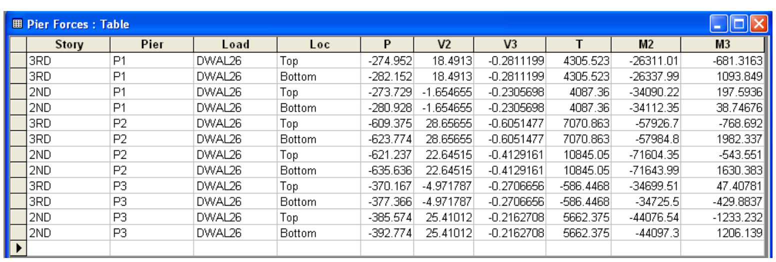



**Figure V.2:** Export Output to Access Database File



Pier Forces : Table

| Story | Pier | Load | Loc | P | V2 | V3 | T | M2 | M3 |
|---|---|---|---|---|---|---|---|---|---|
| 3RD | P1 | DWAL26 | Top | -274.952 | 18.4913 | -0.2811199 | 4305.523 | -26311.01 | -681.3163 |
| 3RD | P1 | DWAL26 | Bottom | -282.152 | 18.4913 | -0.2811199 | 4305.523 | -26337.99 | 1093.849 |
| 2ND | P1 | DWAL26 | Top | -273.729 | -1.654655 | -0.2305698 | 4087.36 | -34090.22 | 197.5936 |
| 2ND | P1 | DWAL26 | Bottom | -280.928 | -1.654655 | -0.2305698 | 4087.36 | -34112.35 | 38.74676 |
| 3RD | P2 | DWAL26 | Top | -609.375 | 28.65655 | -0.6051477 | 7070.863 | -57926.7 | -768.692 |
| 3RD | P2 | DWAL26 | Bottom | -623.774 | 28.65655 | -0.6051477 | 7070.863 | -57984.8 | 1982.337 |
| 2ND | P2 | DWAL26 | Top | -621.237 | 22.64515 | -0.4129161 | 10845.05 | -71604.35 | -543.551 |
| 2ND | P2 | DWAL26 | Bottom | -635.636 | 22.64515 | -0.4129161 | 10845.05 | -71643.99 | 1630.383 |
| 3RD | P3 | DWAL26 | Top | -370.167 | -4.971787 | -0.2706656 | -586.4468 | -34699.51 | 47.40781 |
| 3RD | P3 | DWAL26 | Bottom | -377.366 | -4.971787 | -0.2706656 | -586.4468 | -34725.5 | -429.8837 |
| 2ND | P3 | DWAL26 | Top | -385.574 | 25.41012 | -0.2162708 | 5662.375 | -44076.54 | -1233.232 |
| 2ND | P3 | DWAL26 | Bottom | -392.774 | 25.41012 | -0.2162708 | 5662.375 | -44097.3 | 1206.139 |

**Figure V.3:** Access Database Output File edit box and the Browse button to specify the path and filename for storing the .txt file. The Selection Only check box is enabled if point, line or area objects were selected before the File menu > Print Tables > Input command was selected; only data for the selected objects will be include in the printed output. Unchecking this check box will cause data for all objects in the model to be included in the printed output. The Envelopes Only box is checked when only envelopes (i.e., minimums and maximums) for the specified data would be provided as the analysis output.

# APPENDIX

# APPENDIX A

## Mesh Transitioning and Compatibility The Automated Line Constraint

**Ashraf Habibullah[1], S.E.**
**M. Iqbal Suharwardy[2], S.E., Ph.D.**

## A.1 Introduction

In the application of the Finite Element Analysis Method, the most time consuming task is usually the creation and modification of the finite element mesh of the system. Not to mention the fact that creation of mesh transitions from coarse to fine meshes can be very tedious. Also matching up node points to create compatible meshes at intersecting planes, such as walls and floors can be very labor intensive. And even if the mesh generation is automated the mesh transitioning usually produces irregular or skewed elements that may perform poorly. This may have adverse effects on the design, especially in regions of stress concentration, such as in the vicinity of intersecting planes.

The object based modeling environment of ETABS & SAP2000 clearly addresses these time-consuming shortcomings of the Finite Element Method.

In the object-based modeling environment the Engineer generates the structural model by creating only a few large area objects that physically define the structural units such as wall panels, floors or ramps. The finite element mesh is not explicitly created by the user, but is automatically generated by assigning meshing parameters to the area objects. These parameters may include variables, such as mesh size, mesh spacing and mesh grading among others. With this capability the engineer can study the effects of mesh refinement by just defining a few control parameters. The new model with the desired level of refinement is thus created with minimal effort.

If the meshes on common edges of adjacent area objects do not match up, automated line constraints are generated along those edges. These Line Constraints en-

[1] President & CEO, Computers & Structures, Inc.
[2] Director of Research & Development, Computers & Structures, Inc.

force displacement compatibility between the mismatched meshes of adjacent objects and eliminate the need for mesh transition elements.

What makes this technology really powerful is that while making modifications to the model the Engineer need only be concerned about the few large physical objects of the structure. The modified finite element analytical model gets recreated automatically with any changes to the base objects.

The following examples are designed to illustrate the power and practicality of this technology.

## A.2 Example 1: Simply Supported Plate (Mismatched Meshing)

As illustrated in Figure A.1, this is a model of a simply supported plate, which has been modeled in two different ways. In one case the mesh is uniform across the plate and in the other case the mesh is fine on one half of the plate and coarse on the other half of the plate. In the latter case, an interpolating line constraint is automatically generated to enforce displacement compatibility between the adjacent halves of the plate where the mesh does not match. As shown in the figure, correlation between the two models is very good.

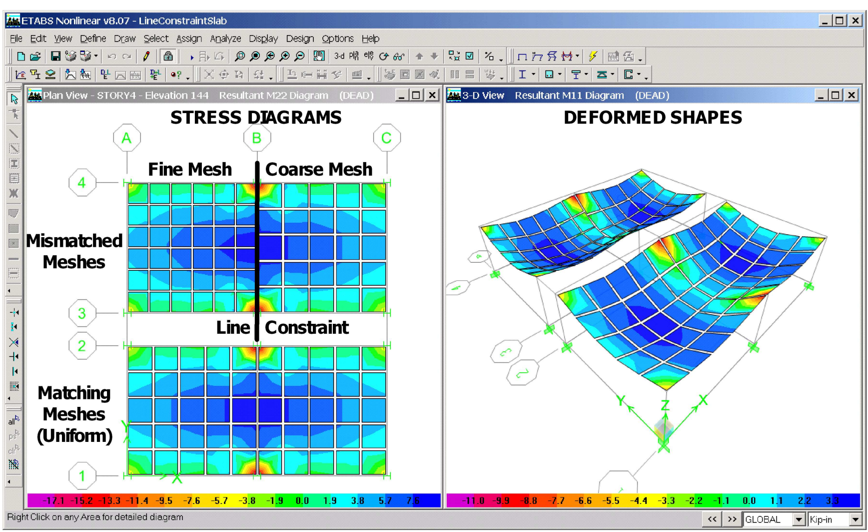



**Figure A.1:** Simply Supported Plate with Mismatching Edges

## A.3 Example 2: Curved Ramp Supported by Curved Wall

This example, Figure A.2, illustrates the use of Line Constraints to capture the interaction of a curved shear wall supporting a curved ramp. Notice that there are no joints at the points where the ramp element edges intersect the wall element edges. Displacement compatibility along the lines of intersection of the ramp and the wall is enforced automatically by the generation of Line Constraints along those lines. Notice how the application of Line Constraints allows the wall and ramp mesh to retain a simple rectangular (or quadrilateral) configuration. A conventional finite element model would be very irregular because it would need all the additional joints (and corresponding elements) to allow for the ramp element and wall element edge intersections.

## A.4 Example 3: Floor Slab – Shear Wall Compatibility

This example, Figure A.3, illustrates a 3D Concrete Flat Plate Building with shear walls and an elevator core. Again, in this model, Line Constraints automatically appear at the lines where the floor and wall objects intersect. This, of course, as in previous examples, will enforce displacement compatibility when mesh geometries do not match. As shown in the deformed shape of the Elevator Core, in many places the wall meshing does not match the floor meshing. All elements meeting at common edges, however, still show no displacement incompatibilities, even though the element nodes do not coincide.

## A.5 Example 4: Shear Wall – Spandrel Transition

This example, Figure A.4, models a Shear wall – Spandrel System, illustrating mesh transitioning from the spandrel to the shear wall. Line Constraints are generated as needed in any direction. In this case the Line Constraints are vertical as well as horizontal.

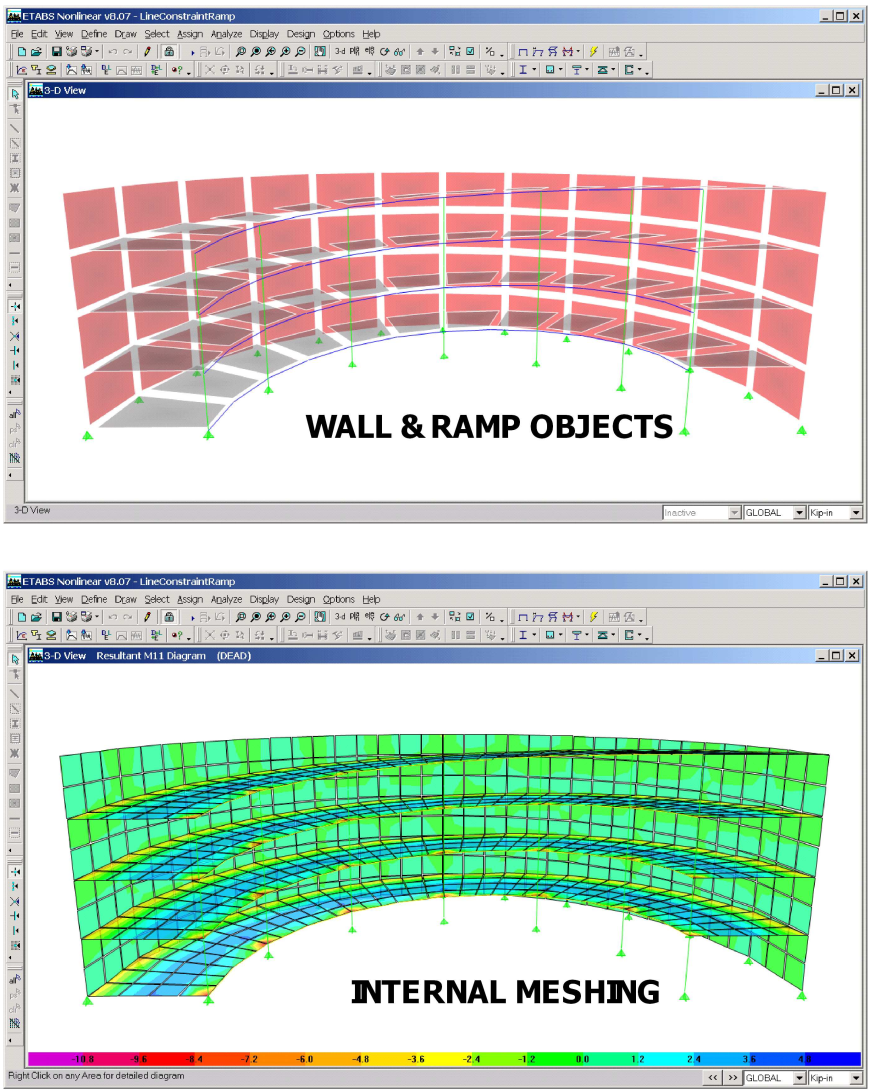



**Figure A.2:** Curved Ramp Supported by Curved Wall

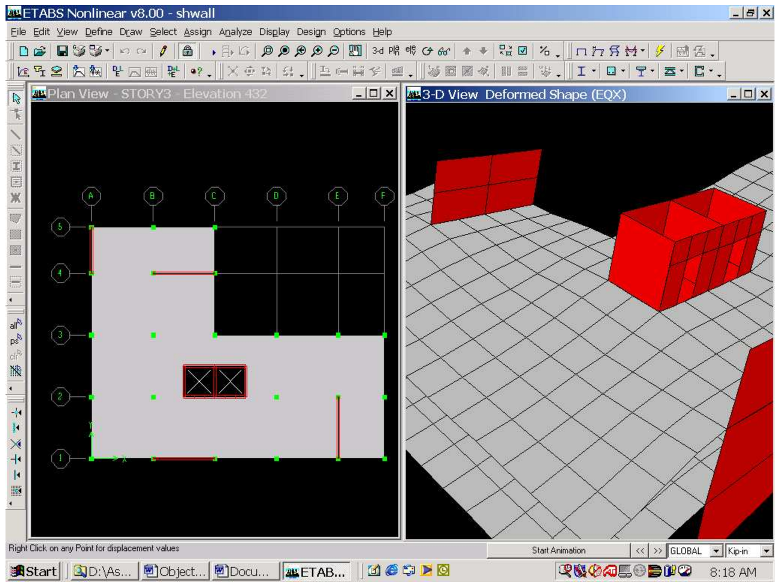



**Figure A.3:** Floor Slab – Shear Wall Compatibility

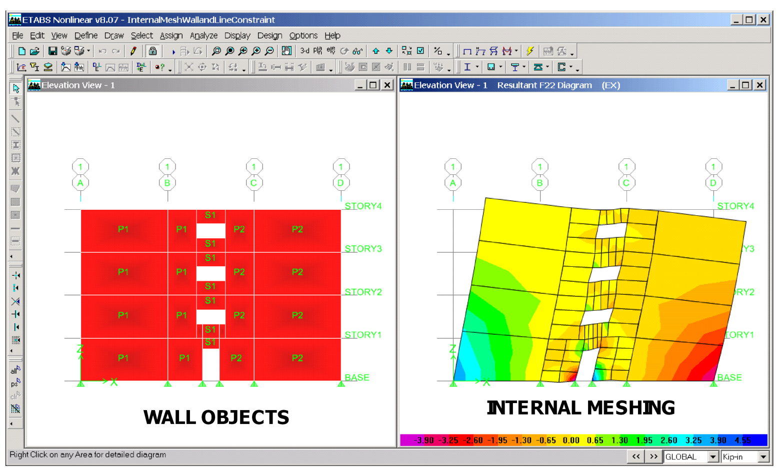



**Figure A.4:** Shear Wall – Spandrel Transition

# BIBLIOGRAPHY

Computers and Structures Inc. [2006*a*], *ETABS — Three Dimensional Analysis of Building Systems*, Berkeley, California. Technical Reference Manual.

Computers and Structures Inc. [2006*b*], *SAP2000 — Integrated Structural Analysis and Design Software*, Berkeley, California. Technical Reference Manual.

Computers and Structures Inc. [2006*c*], 'Website', www.computersandstructures.com. See the latest web contents.

# ABOUT THE SPEAKERS

**Robert Tovani, SE:** Robert Tovani has twenty-five years of experience in structural analysis, design, project management, and construction administration. He is currently president of Engineering Analysis Corporation and an employee of Computer and Structures, Inc. Mr. Tovani received his Bachelors and Masters of Science Degrees for the University of California, Berkeley and is licensed in California as a Civil and Structural Engineer.

Mr. Tovani has developed an extensive background in computer-aided analysis and design. His analysis background includes work on a variety of structures using linear and nonlinear analysis of new and existing structures in static and dynamic loading environments. He has developed computer models on high rise structures in excess of 100 stories and has provided design work on a variety of structural framing systems including base isolation and other complex framing systems. Mr. Tovani has been using the SAP and ETABS computer programs for over twenty-five years and has worked at CSI providing training, analysis and modeling assistance to CSI and Engineering Analysis clients.

**Atif Habibullah, PE:** Atif Habibullah has extensive experience using CSI products, having worked in CSI's Software Support department for five years. For the past two years, Atif has helped instruct engineers through CSI Educational Services training seminars. He has a strong background in modeling a variety of structural systems, solving special modeling problems and in the interpretation of analysis results. Prior to working at CSI, Atif worked at a leading design firm for 4 years using CSI products, particularly in the design of multi-story steel and concrete building structures such as hospitals, office buildings, towers, bridges, stadiums and dams.