

“Optimized Modeling and Design of Steel Structures using ETABS”

presented by



Seminar Topics

General Modeling Techniques

- ❑ The advantages of an Object Oriented Integrated product
- ❑ Model Creation & Editing, Locating with Coordinates, Grids and Snaps
- ❑ Shortcuts and productivity features
- ❑ Fast Draw, Replication, Extrusion, Nudge, Trim, Align, Center, Offset and Mesh
- ❑ Perspective Views, Developed Elevations, Reference Planes and Reference Lines
- ❑ Import/Export Features using AutoCAD

Static and Dynamic Loading of Diaphragms

- ❑ Code Static
- ❑ Response Spectra
- ❑ Time History
- ❑ Special Seismic Load Effects
- ❑ Pushover - Static and Nonlinear
- ❑ Rigid, Semi-rigid, and Flexible Floor Diaphragms

Steel Frame Design

- ❑ Steel Frame Design with no preliminary member sizing by engineer
- ❑ Lateral Drift Control with Automated Virtual Work and no second guessing of member sizes
- ❑ Detailed Concentric Brace Design with ETABS and verification by hand calculations
- ❑ Detailed Eccentric Brace Design discussion
- ❑ Unbonded Brace Design
- ❑ Moment Frame Design
- ❑ Composite Beam Design
- ❑ Vibration Analysis

Copyright

The computer program ETABS and all associated documentation are proprietary and copyrighted products. Worldwide rights of ownership rest with Computers and Structures, Inc. Unlicensed use of the program or reproduction of the documentation in any form, without prior written authorization from Computers and Structures, Inc., is explicitly prohibited.

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

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

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

Table of Contents

Introduction	4
Example I – General Modeling Techniques	5
Example II – More General Modeling Techniques	11
Example III – Advanced Modeling Techniques	19
Example IV – Concentric Braced 4-Story Steel Frame Structure	25
Example V – Two Story Steel Braced Frame Structure	31
Example VI – Composite Beam Design	39
Example VII – Nonlinear Time History Analysis	47
Example VIII – Pushover Analysis	55
About the Speakers	59

Introduction

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. All of the examples that we present (except for the few real life models in the beginning) will be drawn from scratch to exhibit the most general and common modeling techniques mentioned above.

The morning will be spent discussing most of the General Modeling Techniques, and Static Loading. Part of the morning, as well as part of the afternoon will be spent on Steel Frame Design. We will finish the afternoon presentation discussing Nonlinear Time History (Dynamic Loading) and Push-over Analysis.

We have chosen eight examples that we will describe in the presentation. In these seminar notes, you will find descriptions, computer model definitions, and results for each example. As we present each model, please feel free to follow along.

General Modeling Techniques –

Example I Illustrative Examples Showing Shortcuts and Productivity Features

Description

Several existing ETABS models are presented below to illustrate various important modeling techniques.

The first illustrative example shows the importance of assigning selected elements into “groups” and shows the powerful editing feature of the “copy assigns” command. The application of rigid diaphragm versus discrete floor element modeling is also demonstrated in Figure 1-1 and Figure 1-2 below.

Also discussed in this introductory chapter is the use of the “similar floors” feature of ETABS. Figure 1-3 is used to illustrate the use of more than one grid system to construct a computer model having complex geometry.

Figure 1-4 shows that a sloped diaphragm can be modeled easily. Reference planes are added to describe the building having various roof elevations.

Significant Options of ETABS Exemplified in Example 1

- Copy Assigns
- Group assignments
- Flexible Diaphragms
- Similar Floors
- Data output into Microsoft excel
- Moving through Elevations and Plans
- Import from DXF file or AutoCAD files
- Group Selection Features
- Steel Brace Design Combinations

Example 1

Computer Model Definition

Figure 1-1 depicts a structure having 25-story building of irregular geometry. The structure is base isolated at the third level above the base and has 14 grids in the X-direction and 15 grids in the Y-direction. The structure also has both vertical and horizontal irregularities.

The model is loaded with both static and dynamic loads. Gravity loads and lateral time history functions are used in various combinations in the design of the structure.

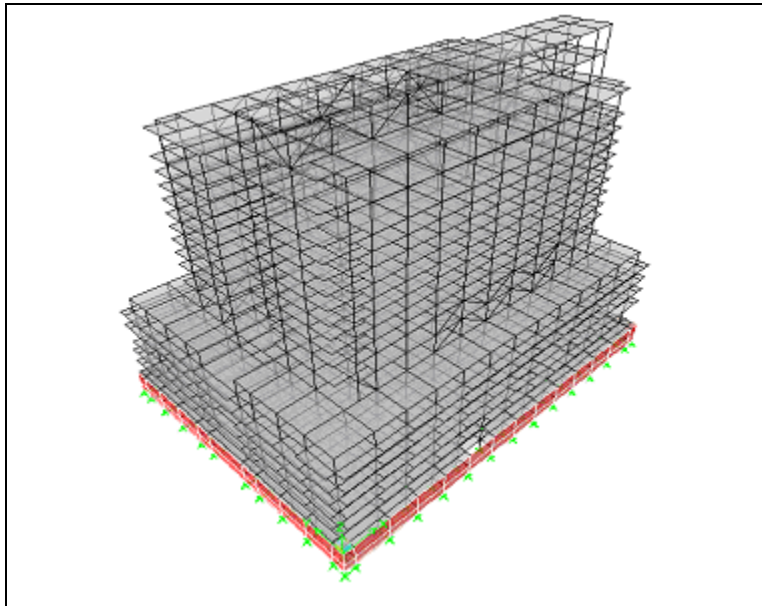


Figure 1-1 3-D view of structure

Description

Figure 1-2 shows an interior braced frame elevation. The intent of the braced frame at this location was to provide resistance to lateral loads in both shear and overturning. The horizontal truss elements at the roof and mid-height of the building are transfer trusses to help distribute overturning forces from the interior braces to the building exterior. If the top and bottom chord elements of the horizontal trusses are to be modeled properly, the joints must be disconnected from the rigid floor diaphragms to allow shortening or elongation of the horizontal truss chord elements.

To illustrate the use of the “copy assigns” command, an entire column line can be copied and assigned “pasted” anywhere within the computer model. Note that any assignment including sections, releases, offsets, output stations, local axis, property modifiers and line types can all be captured and assigned in one simple command.

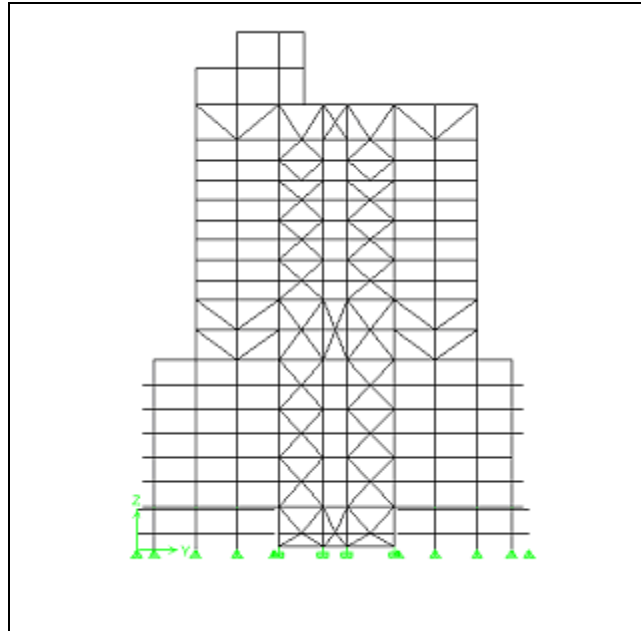


Figure 1-2 *Elevation of an interior braced frame*

Example 2

Significant Options of ETABS Exemplified in Example 2

- Multiple Grids
- Element Local Axis

Computer Model Definition

Figure 1-3 depicts a structure having two grids. The first grid consists of a Cartesian coordinate system. The second grid is a cylindrical coordinate system.

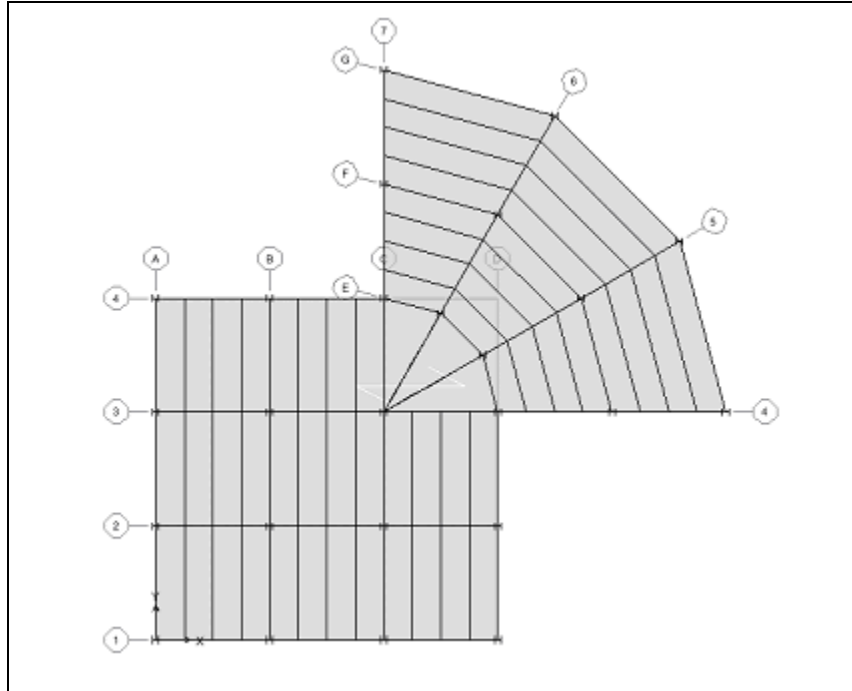


Figure 1-3 *Plan of Structure with Multiple Grids*

Example 3

Significant Options of ETABS Exemplified in Example 3

- Discrete Roof Element Modeling
- Sloped Diaphragms
- Reference Planes
- Reference Lines
- Divide Line/ Join Line Options
- Merge Points
- Align Points

Computer Model Definition

Figure 1-4 consists of a gymnasium structure that has multiple roof diaphragms. Several of the roof diaphragms are sloped between various roof elevations. The structure includes various bracing elements which have been located using the “divide line” option. Reference planes have also been introduced to aid in the development of the model.

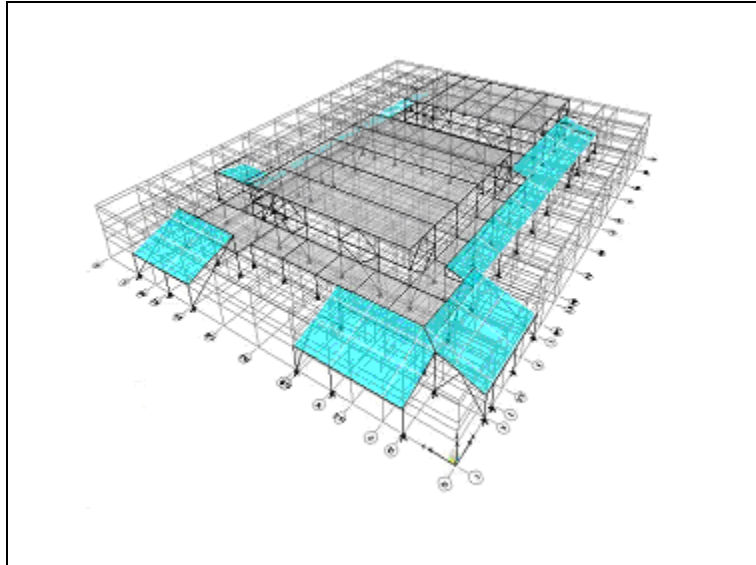


Figure 1-4 3-D image of Structure with Sloped Diaphragms

More General Modeling Techniques – EXAMPLE II Four Story C-Shaped Steel Frame Structure

Description

This is a four-story C-shaped Model, subjected to vertical static loading and computer-generated earthquake loading per the 1997 Uniform Building Code. The structure consists of steel beams and columns along with a concrete deck on every level.

Significant Options of ETABS Exemplified

- ❑ Quick-draw Beam and Column Generation
- ❑ Floor Area Generation
- ❑ Expand/Shrink Areas Option
- ❑ Steel Design Optimization
- ❑ Modes Shapes
- ❑ Lateral Displacement Targets
- ❑ Local Axis Orientation

Computer Model Definition

The structure is a C-shaped 4-column line, 3-bay system. Each bay is spaced 24 feet apart. Each story level is 12 feet in height. Kip-inch-second units are used. The modulus of elasticity is 29000 ksi. To see frame geometry, please refer to Figure 2-1.

Other parameters associated with the structure are as follows:

Coefficient of Thermal Expansion	= 6.500E-06
Poisson's ratio	= 0.3
Minimum Yield Stress, F_y	= 50 ksi
Minimum Tensile Strength, F_u	= 65 ksi

Deck Properties:

Slab Depth	= 3.25 in
Deck Depth	= 3 in
Rib Width	= 6 in
Rib Spacing	= 12 in

Service Loads:

Superimposed Dead Load	Roof	= 30 psf
	Story 3	= 30 psf
	Story 2	= 30 psf
	Story 1	= 30 psf

For the UBC97 seismic load analysis, the following code parameters associated with the structure are as follows:

UBC Seismic zone factor, Z	= 0.40
UBC Soil Profile Type	= SC
UBC Importance factor, I	= 1.0
UBC Overstrength Factor	= 8.5
UBC coefficient C_t	= 0.035
UBC Near source Factor, Seismic Source Type	= B
Distance to Source	= 15 km

Quick Draw Beam and Column Generation

Create Lines at Regions or at Clicks

The Draw menu -> Draw Line Objects -> Create Lines in Region or at Clicks (plan, elev, 3D) command works in two different ways.

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 Columns 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.

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 XY plane).

After using these functions our C-shaped model looks like this:

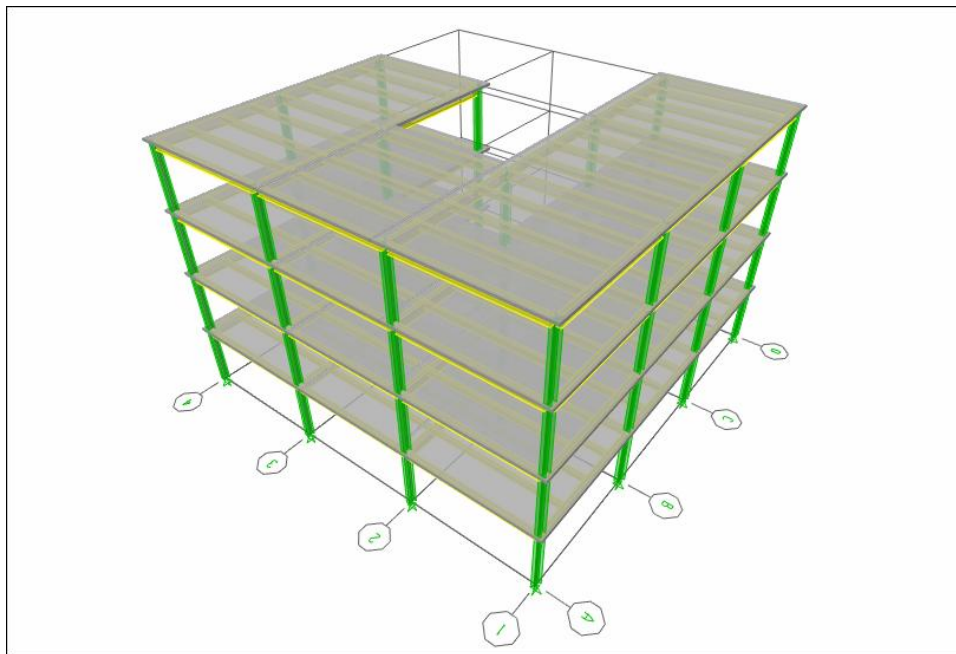


Figure 2-1 3-D View of Structure

Results

After running the analysis (under menu->Analyze-> Run Analysis), go to Design->Steel Frame Design-> Start Design Check of Structure. ETABS tells you that there are 132 steel frames whose analysis and design sections differ. Design is an iterative process. The sections that you use to run your original analysis are not typically the same sections that you end up with at the end of the design process. You always want to be sure to run a building analysis using your final frame section sizes and then run a design check based on the forces obtained from that analysis. The Design menu -> Steel Frame Design -> Verify Analysis vs. Design Section command is useful for making sure that the design sections are the same as the analysis sections.

After running the analysis and design again, we choose to see the deflection of Joint 13 shown in Figure 2-2 below:

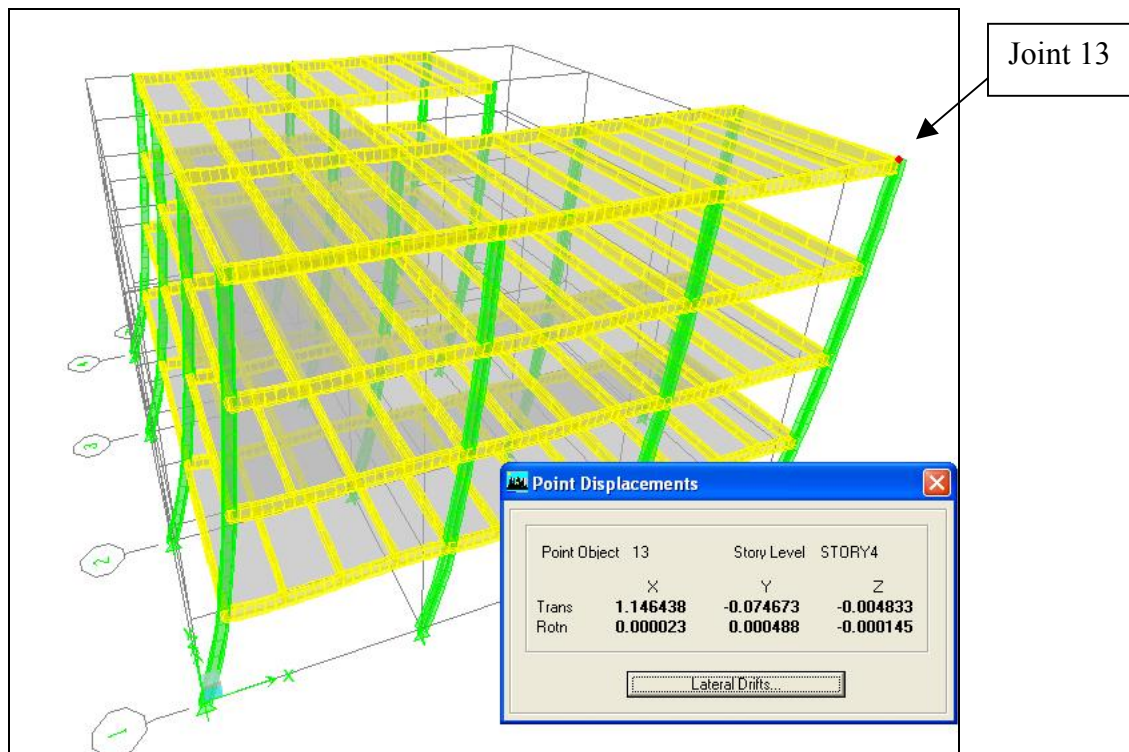


Figure 2-2 Deflected Shape Under Earthquake Load Case

Set Lateral Displacement Target

You can see that Joint 13 on Story 4 has a displacement of **1.14** inches in the X-direction. If we want to set the maximum displacement on this joint not to exceed .9 inches, we can set a lateral displacement target. From the menu, click Design -> Steel Frame Design -> Set Lateral Displacement Targets command to open the Displacement Optimization form where you can specify displacement targets for various load cases. Enter a maximum displacement of .9 inches for the lateral earthquake load in the X-direction.

The intent is to pick a point, typically at the roof level of your building, and specify a maximum displacement target (in any direction) for one or more load cases. For displacement optimization, ETABS predicts which members should be increased in size to control the displacements based on the energy per unit volume of the members. The members with more energy per unit volume are increased in size by a larger percentage than those with smaller energies per unit volume. Some members with small energy per unit volume may be decreased in size if they are still acceptable for strength considerations.

Next, run the analysis and design again. ETABS will tell you there are 86 frames whose analysis and design sections differ. This means 86 frames sizes have changed in order to control the displacement at joint 13. ETABS will then ask you whether you wish to select these sections. Click Yes. You will notice members **ONLY** parallel to the x direction have increased in size. **The deflection at Joint 13 is .81 inches. If you keep iterating, the deflection will get closer to the desired .9-inch deflection.**



Figure 2-3 *Frame Elements in X-Direction*

Mode Shapes

Click Display -> Show Mode Shape command to access the Mode Shape form. Specify the mode whose deformed shape you would like to plot. This structure has been modeled as a flexible diaphragm. This means the model will NOT rotate about an axis perpendicular to its own plane (global Z-axis) as a rigid body. The deflected shape below shows portions of the model deflecting in different directions.

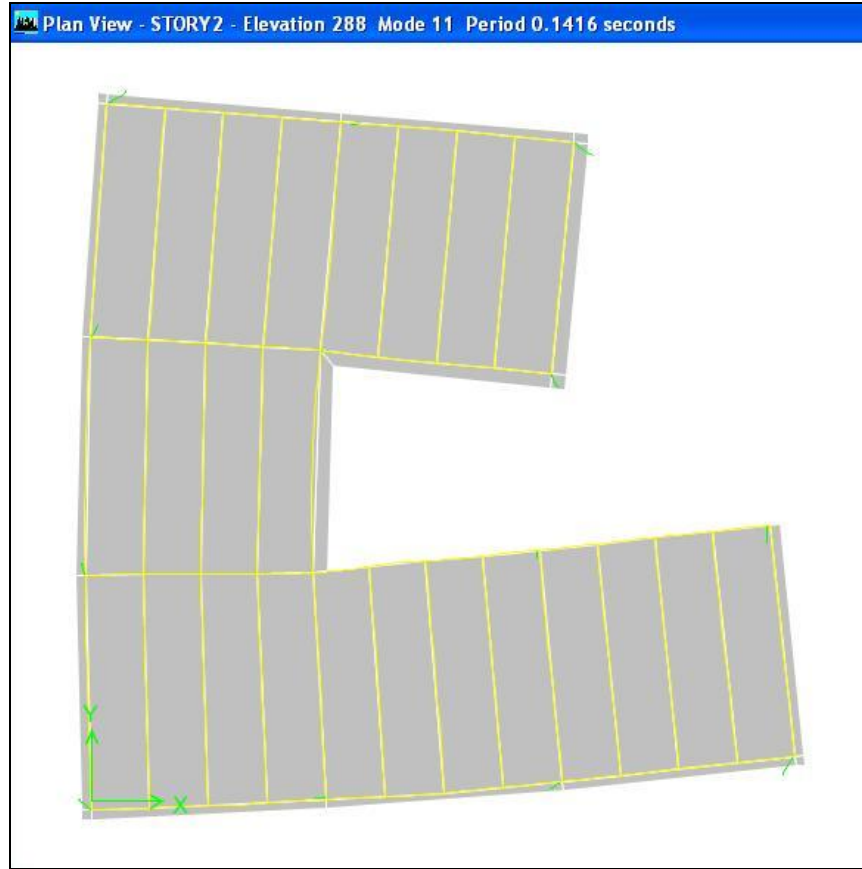


Figure 2-4 *C-Shaped Structure- Mode Shape 11*

EXAMPLE III Advanced Modeling Techniques – Eight Story Parking Garage Structure

Description

This is an eight-story parking garage structure with curved ramps located at each level. Each floor has a 12 ft story height. Each bay is 24 feet long in the X-direction and 36 feet long in the Y-direction. The structure is laterally supported by a curved shear wall system located at the edge of the ramp system.

Significant Options of ETABS Exemplified

- ❑ Replication Features
- ❑ Extrude Lines to Areas Feature
- ❑ Ramp Generation
- ❑ Curved Shear Wall Generation
- ❑ Draw Developed Elevations
- ❑ Auto Line Constraint Feature
- ❑ Creation of OpenGL Views

Computer Model Definition

Grid and Line Generation

The structure is an 11-column line, 5-bay system. Kip-inch-second units are used. Spacing in the X and Y direction is 24 and 36 feet respectively. It is an 8-story structure with 12 ft story heights.

To create this model, go to elevation line 3. Draw a line from Grid line A to Grid line C. Select this line, go to Edit-> Replicate-> Story Tab and select the EVEN story levels. (Holding down the CTRL key while selecting levels). Draw a line from Grid line I to Grid line K and replicate this line on all *odd* story levels. Next, draw a line starting at level 8 diagonally down to level 7 connecting each line at every level. (In this example, it does not matter which beam size you select). Currently, the model should look like this:

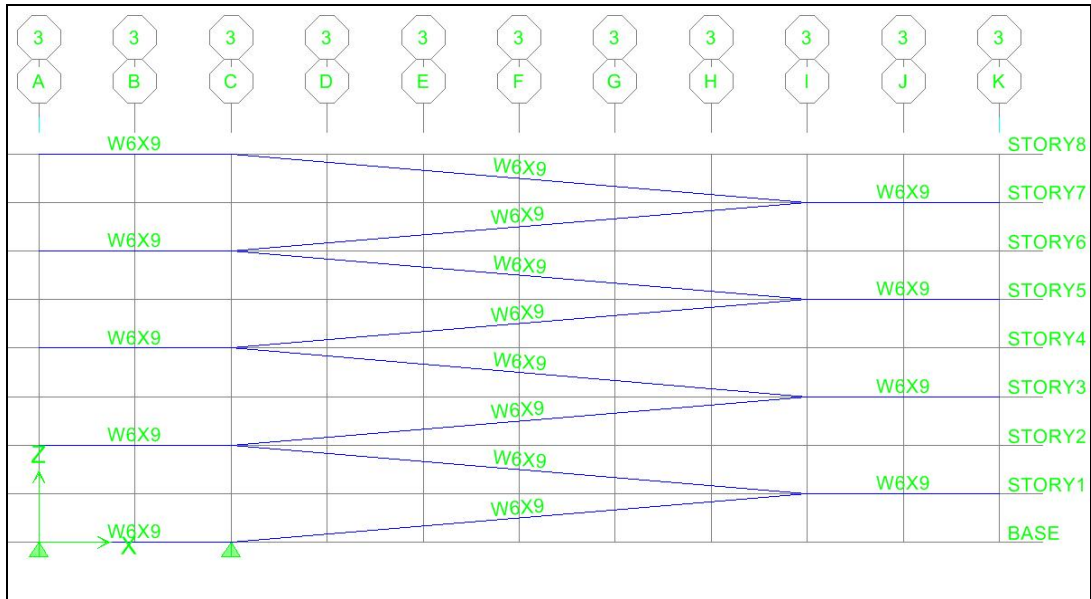


Figure 3-1 *Elevation of Ramp Structure*

Ramp Generation

Next, select all flat beams on every level and every *other* inclined beam element. Go to Edit -> Extrude Lines to Areas -> Linear. In the Increment Data box, enter 36 feet for dy and number of replications = 2. Now, select all of the flat beams again and the inclined beams that were *not* selected the first time. Perform the same extrusion application except use -36ft for dy. Your model should now look like Figure 3-2:

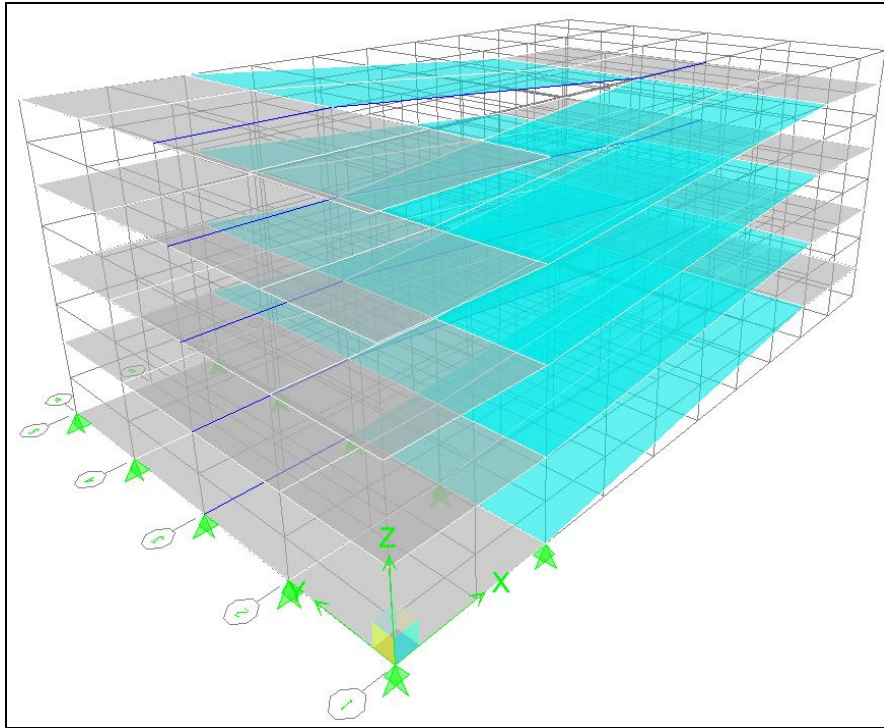
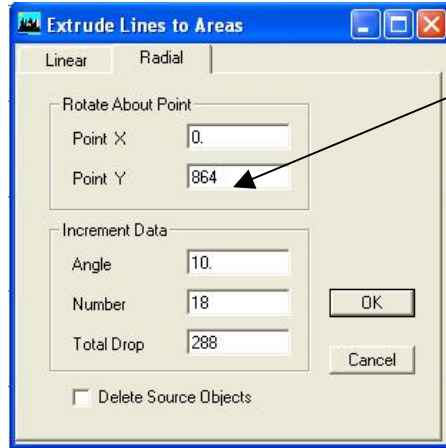


Figure 3-2 3-D View of Ramp Structure

Circular Edge Ramp Generation

Now, we are ready to create our circular ramp. Go to Elevation A, and draw a line from a point in between Grid line 3 and 4 to Grid line 5. (Make sure snap to lines and edges is activated under Draw-> Snap to -> Line and Edges). Replicate this line on all even floors. Select all lines in Elevation A, go to Edit-> Extrude lines to Areas-> Radial tab. The ramp is a semi-circle so enter 10-degree angles extruded 18 times creating a 180 degree curved surface. The total drop of the ramp is 2 stories or 288 in.



The ramp is rotated about Grid line 3 and A. Y coordinate is 864.

Figure 3-3 *Extrude Lines to Areas Box*

After completing this step, your model should look like Figure 3-4:

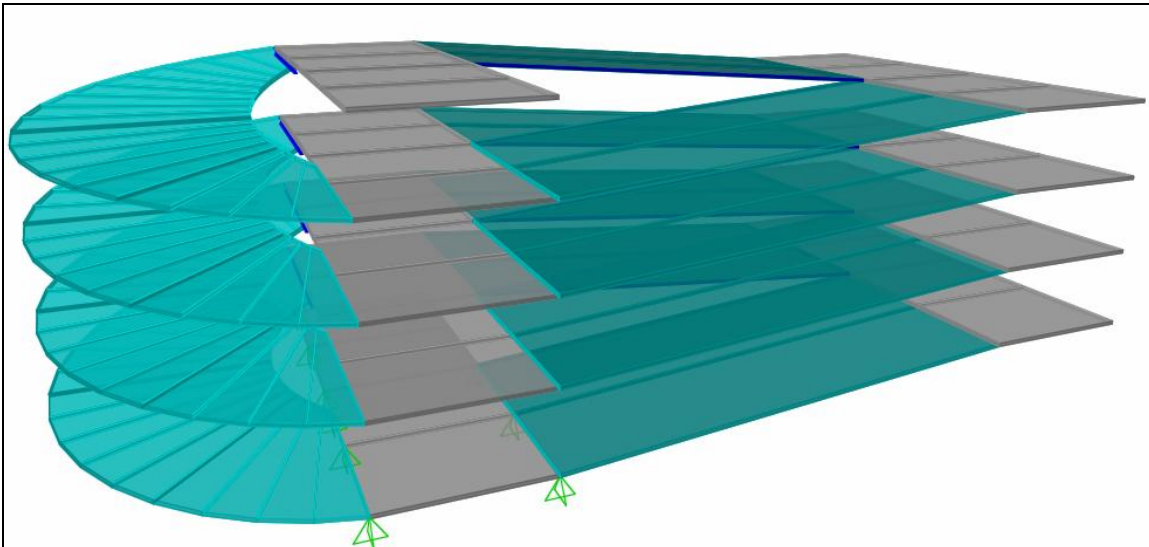


Figure 3-4 *Circular Ramp Structure*

Curved Shear Wall Generation

Now, we are interested in creating a curved shear wall surface around the edge of the circular ramp surface, which will run the full height of the structure. Go to floor plan 8 and select *all stories* the drop down box at the bottom right hand corner of the screen. This will create shear walls on all floors instead of just the 8th floor. Under View -> Set Building View Options -> de-select the toggle labeled *invisible* under point objects. This will enable the user to view and snap to joints when creating the curved shear wall. Go to the Draw -> Draw Area Objects -> Draw Walls and click on the joint located at the intersection of Grid line A and 5. Keep clicking on every joint around the edge of the ramp. When you get half way around the semi-circle, you will need to go to floor plan 7

because the rest of the joints that are located below Story 8 cannot be seen. Complete the semi circle once you get to the intersection of Grid line 1 and A.

Developed Elevation

Developed elevations can simultaneously show multiple faces of the building in a single "unfolded" elevation view. To draw a developed elevation, from the menu select Draw-> Draw Developed Elevation Definition, type a name for the developed elevation in the edit box in the Developed Elevations area of the form and click on 'Add New Name'. Next, in floor plan 8, draw an elevation the same way you drew the curved shear wall, by clicking on the edge joints of the ramp until you've created a semi-circle again. To view the elevation, click the View menu-> Set elevation view and select the name of your Developed Elevation.

Note: If the Options menu -> Window command is set to two or more windows and those windows are set to Plan or 3D View, the location of the developed elevation is illustrated with a cyan line (default) in the Plan and 3D view windows. Click on a couple of wall elements and delete them. This will create an opening in the curved shear wall element. Your model should look like Figure 3-5:

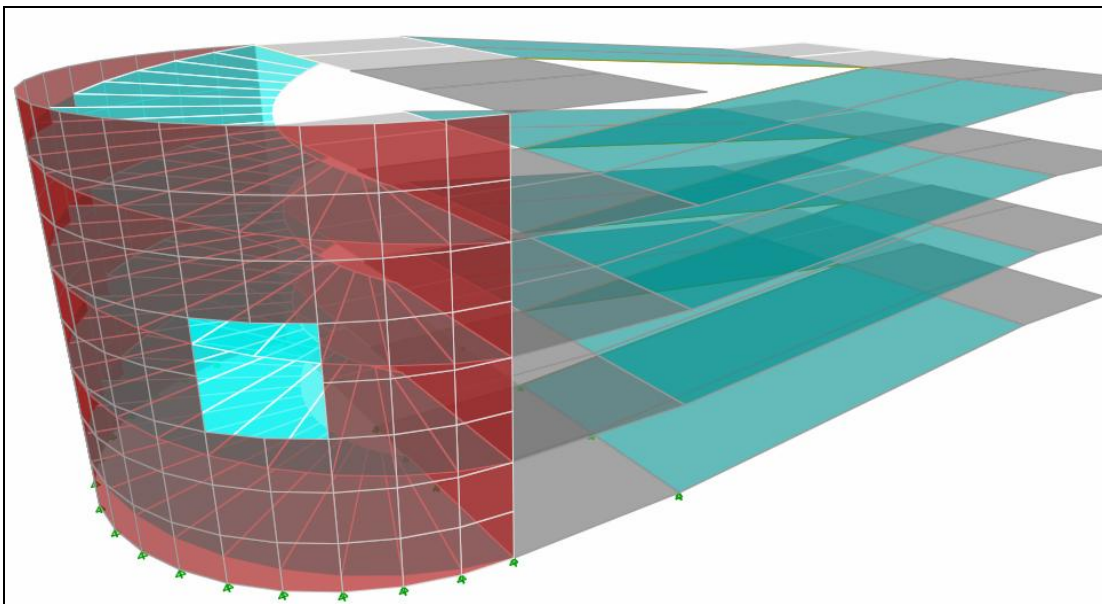


Figure 3-5 *Circular Ramp Structure with Curved Shear Wall*

Auto Line Constraint

If you look closely at Figure 3-5, you will notice that the shear wall nodes and ramp nodes do not connect. In typical finite element analysis, shell elements are connected to other elements at corner points only. 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 the capability of using 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. Additional information regarding the auto line constraint functionality in ETABS can be found in “Mesh Transitioning and Compatibility: The Automated Line Constraint in ETABS & SAP2000”. (http://www.csiberkeley.com/images/line_constraint.pdf).

EXAMPLE IV Concentric Braced 4-Story Steel Frame Structure

Description

This is a four-story special concentrically braced frame structure, subjected to vertical static loading and computer-generated earthquake loading per the 1997 Uniform Building Code. X bracing is placed in specific bays to resist the lateral forces. There are two decks per floor that span in different directions.

Significant Options of ETABS Exemplified

- ❑ Rigid Diaphragm Assignments
- ❑ Use of Section Cuts
- ❑ Shell Element Axis Rotation
- ❑ Replication Techniques
- ❑ Deck Element Options
- ❑ Quick Draw Brace Generation
- ❑ Automated Earthquake generation

Computer Model Definition

The structure is an L-shaped 13-column line, 12-bay system. Kip-inch-second units are used. The modulus of elasticity is 29000 ksi. To see frame geometry, please refer to Figures 4-1 and 4-2.

Other parameters associated with the structure are as follows:

Deck Properties:

Slab Depth	= 3.25 in
Deck Depth	= 3 in
Rib Width	= 6 in
Rib Spacing	= 12 in

Service Loads:

Live Load:	Roof	= 20 psf
	Story 3	= 80 psf
	Story 2	= 80 psf
	Story 1	= 80 psf

Superimposed Dead Load	Roof	= 73.7 psf
	Story 3	= 75.4 psf
	Story 2	= 75.4 psf
	Story 1	= 75.4 psf

In this model, the dead load self-weight multiplier has been set to zero. This can be found under Define->Static Load Case. The only dead load associated with this example is the superimposed dead loads listed above.

For the UBC97 seismic load analysis, the following code parameters associated with the structure are as follows:

UBC Seismic zone factor, Z	= 0.40
UBC Soil Profile Type	= SD
UBC Importance factor, I	= 1.0
UBC Overstrength Factor	= 6.4
UBC coefficient C_t	= 0.035
UBC Near source Factor, Seismic Source Type	= B
Distance to Source	= 8 km

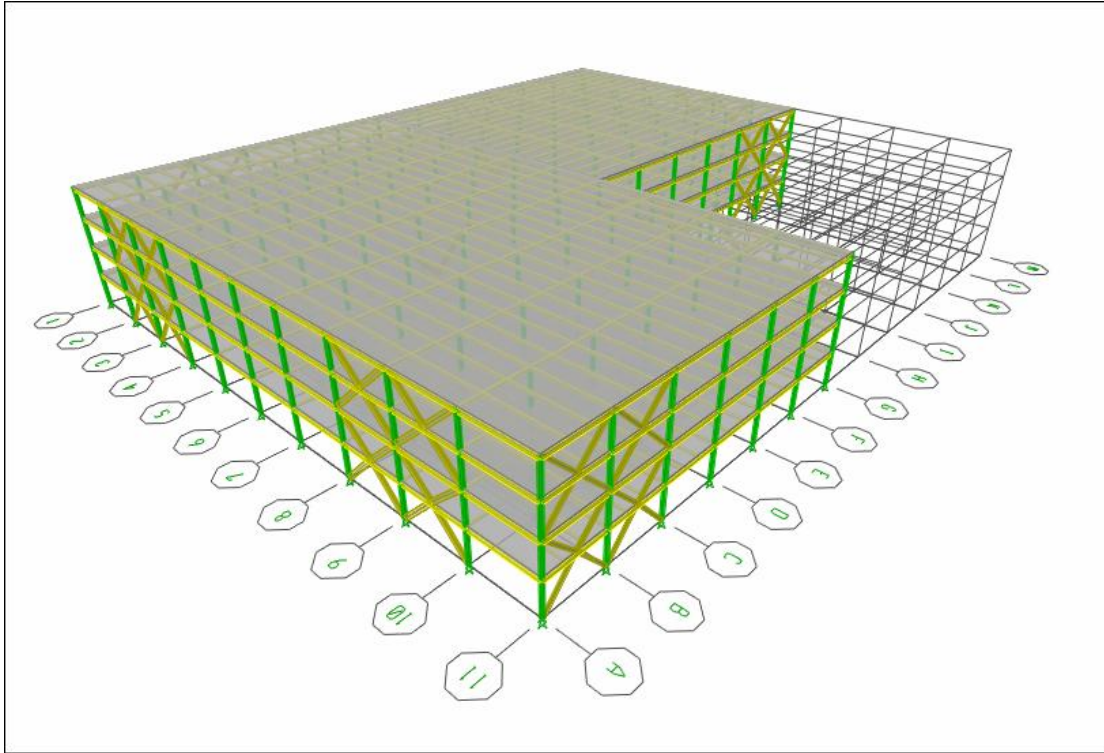


Figure 4-1 3-D view of structure

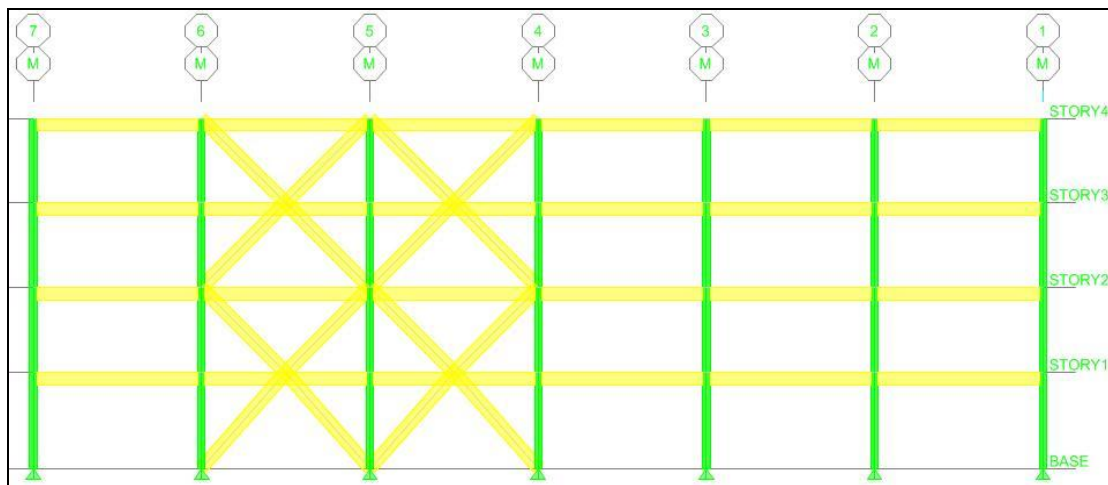


Figure 4-2 Elevation Line M

Rigid Diaphragm Assignment

Rigid diaphragms can only be horizontal. Thus rigid diaphragm assignments are not applicable to wall-type and ramp-type area objects. They are only applicable to floor type area objects and to null-type area objects that are in a horizontal plane. In ETABS, a rigid diaphragm translates within its own plane (global X-Y plane) and rotates about an axis perpendicular to its own plane (global Z-axis) as a rigid body. Designating an area object as a rigid diaphragm has no effect on the out-of-plane behavior of the area object. For example, if you specify a concrete floor slab to have plate-bending properties (i.e., out-of-plane bending capability), applying a rigid diaphragm constraint has no effect on the out-of-plane bending of the floor. It only affects in-plane behavior of the floor. Within ETABS, assigning a rigid 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 ETABS automatically meshing the area object.

Results

Section Cut Forces

There are two options available to define Section Cuts:

1. The first option is to define the location of the cut. Use the Define menu -> Section Cuts command to obtain resultant forces acting at section cuts through a model. Define section cuts before or after an analysis is run; however, it is safest to wait until after the analysis has been run. Typically do not define section cuts, and more importantly, the groups used in the section cut definition, until all manual meshing of the model (if any) has been completed. If the groups are defined before manual meshing, some of the point objects that should be in the group may not yet be created.
2. The second option is to manually draw the section cut on any portion of the model. This can be by utilizing the Draw->Draw Section Cut option. You must make sure that the model has been analyzed and you are viewing a member force/stress diagram. This can be found under Display-> Show Member Force/Stress Diagram-> select either frame or shell forces. This second option is used in this example to obtain section cuts forces.

To obtain shell stress forces on Story 4 between Grid lines L and M, go to Draw->Draw Section Cut. Draw a line parallel to Grid line M. The flashing line represents to section cut along Grid line M. Section Cut forces will now be visible on the screen. Please refer to Figure 4-3 and 4-4.

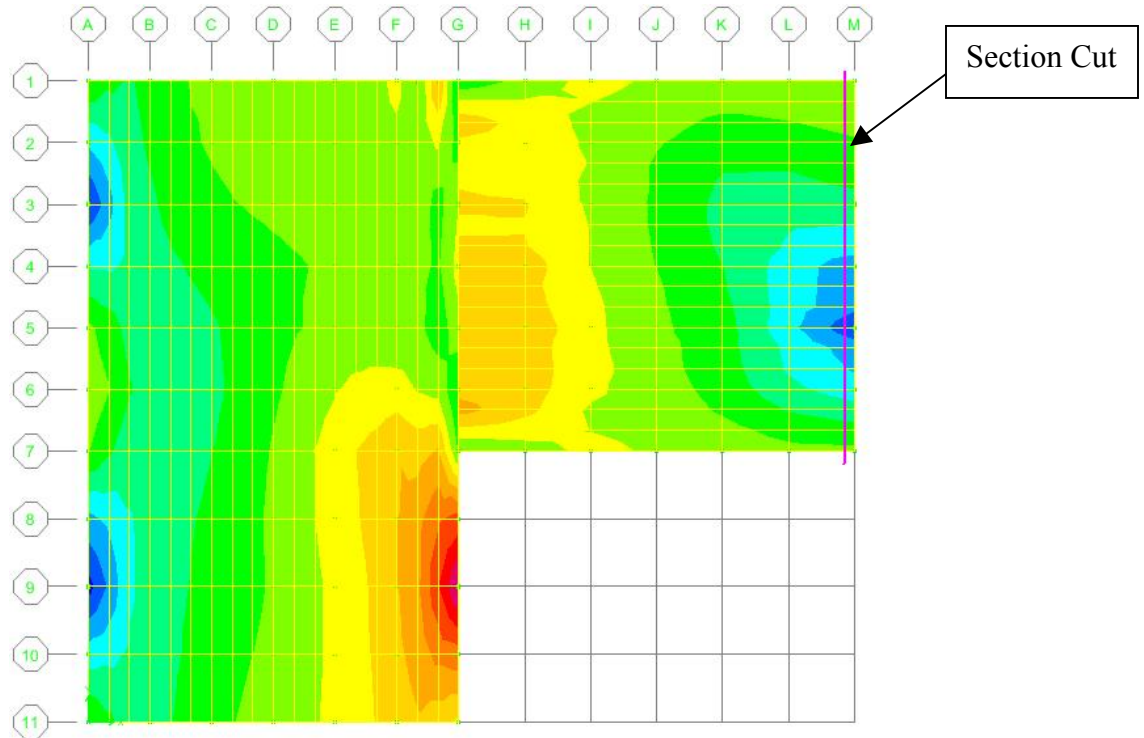


Figure 4-3 Stress Diagram (S12) for Earthquake Load Case

In Figure 4-3, you can see the high stress concentrations along line M between Grid lines 4 and 6. This is where the special concentric braced frames are located. You can see that there are also braces along Grid line A.

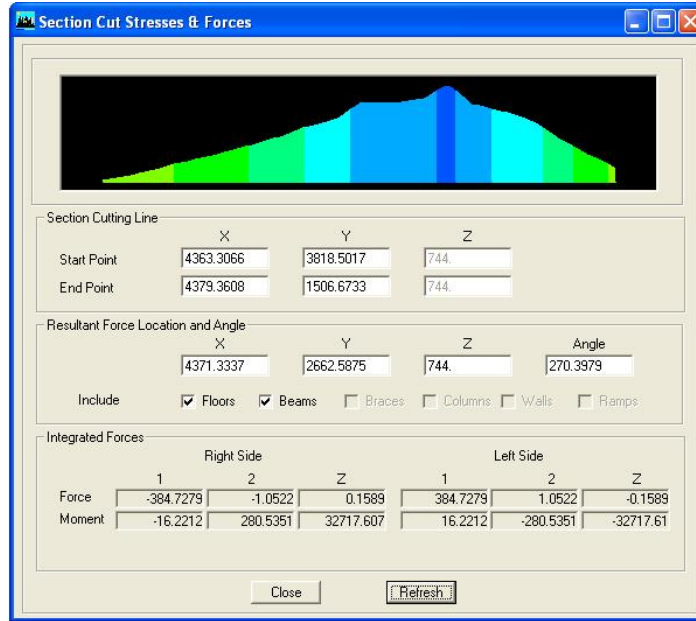


Figure 4-4 *Section Cuts force output*

EXAMPLE V Two Story Steel Braced Frame Structure – Vertical and Lateral Load Analysis

Description

This is a two story braced frame structure, subjected to vertical static loading and computer generated earthquake loading per the 1997 Uniform Building Code. The lateral forces are being resisted by concrete shear walls and steel braces. The structure contains heavy mechanical equipment loads on the Roof level.

The frame geometry is shown in Figure 5-1 and Figure 5-2.

Significant Options of ETABS Exemplified

- Automated Earthquake and Wind Load Generation
- Shell Element Options
- Use of Null floor elements
- Story and Grid Edits
- Mass Source Options
- Steel Brace Overwrite Options
- Shear Wall Generation
- Quick Draw Brace Generation
- Steel Brace Design
- Composite Beam Design

Computer Model Definition

The structure is a 4-column line, 3-bay system. Kip-inch-second units are used. The modulus of elasticity is 29000 ksi.

Other parameters associated with the structure are as follows:

Coefficient of Thermal Expansion	= 6.500E-06
Poisson's ratio	= 0.3
Minimum Yield Stress, F_y	= 50 ksi
Minimum Tensile Strength, F_u	= 65 ksi

Deck Properties:

Slab Depth	= 3.5 in
Deck Depth	= 3 in
Rib Width	= 6 in
Rib Spacing	= 12 in

Service Loads:

Live Load:	Roof	= 20 psf
	Story 1	= 125 psf
	Deck	= 100 psf
Superimposed Dead Load	Roof	= 30 psf
	Story 1	= 31 psf
	Mechanical	= 188 psf

For the UBC97 seismic load analysis, the following code parameters associated with the structure are as follows:

UBC Seismic zone factor, Z	= 0.40
UBC Soil Profile Type	= SD
UBC Importance factor, I	= 1.0
UBC Overstrength Factor	= 5.5
UBC coefficient C_t	= 0.035
UBC Near source Factor, Seismic Source Type	= B
Distance to Source	= 10 km

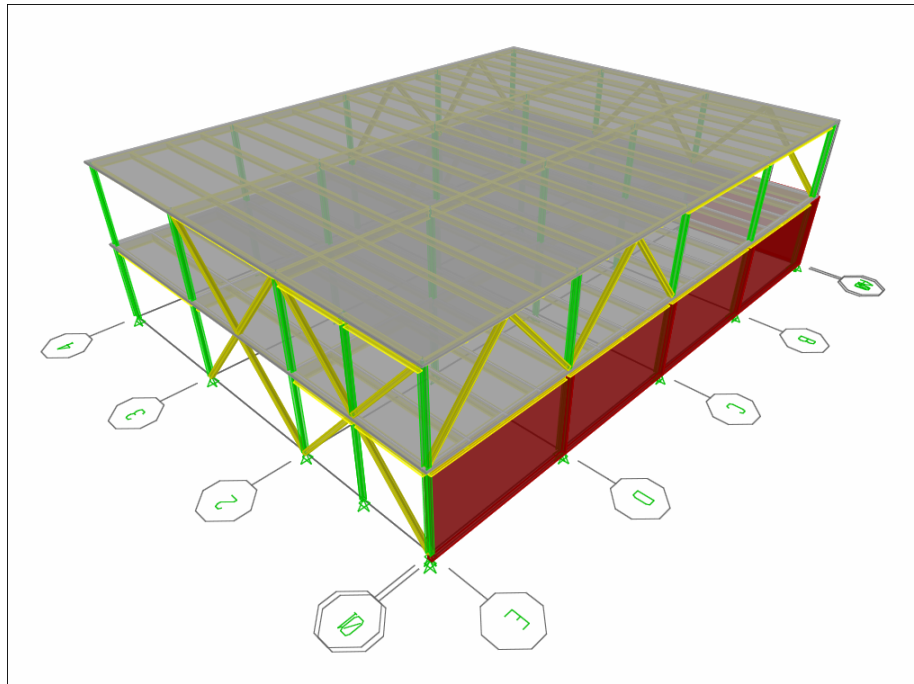


Figure 5-1 3-D view of structure



Figure 5-2 Plan view of Story 1

Mass Source Option

In ETABS, the user has the option of choosing one of three options for defining the source of the mass of a structure. Click the Define menu -> Mass Source command to bring up the Define Mass Source form. The following options appear on the form:

1. From Self and Specified Mass:

Each structural element has a material property associated with it; one of the items specified in the material properties is a mass per unit volume. When the 'From Self and Specified Mass' box is checked, ETABS determines the building mass associated with the element mass by multiplying the volume of each structural element times its specified mass per unit volume. This is the default. It is also possible to assign additional mass to account for partitions and cladding, etc. ETABS adds any additional mass assignments to the element mass to derive a total mass. You cannot have a negative mass in ETABS.

2. From Loads:

This specifies a load combination that defines the mass of the structure. The mass is equal to the weight defined by the load combination divided by the gravitational multiplier, g . This mass is applied to each joint in the structure on a tributary area basis in all three translational directions.

3. From Self and Specified Mass and Loads:

This option combines the first two options, allowing you to consider self-weight, specified mass, and loads in the same analysis.

It is important to remember when using the 'From Self and Specified Mass and Loads' option, NOT to include the Dead Load Case in the 'Define Mass Multiplier for Loads' box. This will account for the dead load of the structure TWICE.

Results

Base Shear Calculation

This information shown below can be printed using:

File -> Print Tables -> Input -> Auto Seismic Loads.

AUTO SEISMIC INPUT DATA

Direction: X
 Typical Eccentricity = 5%
 Eccentricity Overrides: No

Period Calculation: Program Calculated
 $C_t = 0.035$ (in feet units)

Top Story: STORY2
 Bottom Story: BASE

$$V = (C_v I W) / (R T) \quad (\text{Eqn. 1})$$

$$V \leq 2.5 C_a I W / R \quad (\text{Eqn. 2})$$

$$V \geq 0.11 C_a I W \quad (\text{Eqn. 3})$$

$$V \geq 0.8 Z N_v I W / R, \quad (\text{Eqn. 4})$$

If $T \leq 0.7$ sec, then $F_t = 0$

If $T > 0.7$ sec, then $F_t = 0.07 T V \leq 0.25 V$

AUTO SEISMIC CALCULATION RESULTS

$T_a = 0.4260$ sec
 $T_{\text{Used}} = 0.1865$ sec
 $W_{\text{Used}} = 3910.77$

$$V (\text{Eqn 1}) = 0.8446W$$

$$V (\text{Eqn 2}) = 0.2031W$$

$$V (\text{Eqn 3}) = 0.0581W$$

$$V (\text{Eqn 4}) = 0.0788W$$

$$V_{\text{Used}} = 0.2031W = 794.19 \text{ Kips}$$

Steel Frame Design

After running the Steel Frame Design, (See Figure 5-3) the user can right-click on any steel frame element to view the Steel Stress Check Information. By clicking the Details button, the design details for the selected design combination and station location are shown. (See Figure 5-4):

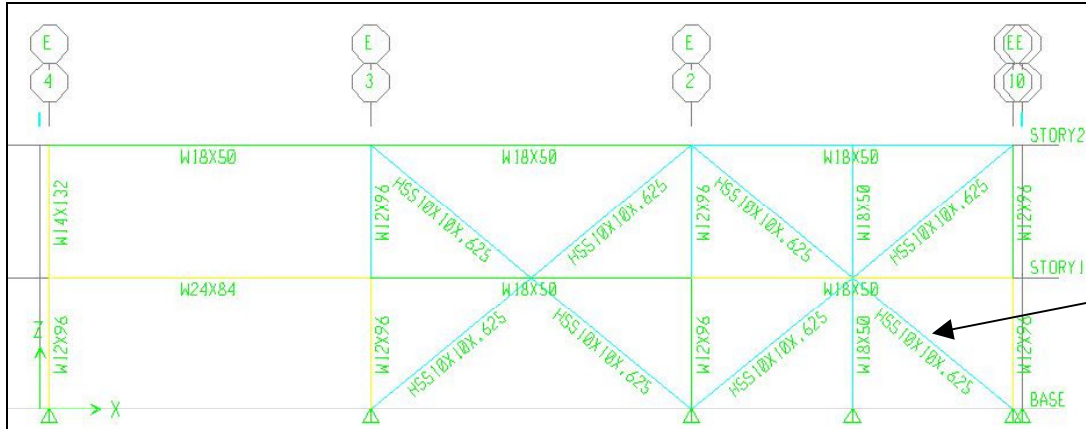


Figure 5-3 Elevation E

Steel Stress Check Information AISC-LRFD99

File Drawing

AISC-LRFD99 STEEL SECTION CHECK Units: Kip-in (Summary for Combo and Station)
 Level: STORY1 Element: D20 Station Loc: 134.466 Section ID: HSS10X10X.625
 Element Type: Special Centrically Braced Frame Classification: Seismic

L=268.931
 A=21.000 i22=304.000 i33=304.000 z22=73.200 z33=73.200
 s22=60.800 s33=60.800 r22=3.805 r33=3.805
 E=29000.000 Fy=50.000 Ry=1.300
 RLLF=1.000 Omega=3.000

P-M33-M22 Demand/Capacity Ratio is 0.227 = 0.212 + 0.015 + 0.000

STRESS CHECK FORCES & MOMENTS						
Combo	DSTLS9	P	M33	M22	U2	U3
		-131.059	56.500	0.000	0.000	0.000

AXIAL FORCE & BIAXIAL MOMENT DESIGN (H1-1a)								
	Pu Load	phi*Pnc Strength	phi*Pnt Strength					
Axial	131.059	619.385	945.000					

SHEAR DESIGN								
	Uu Force	Phi*Un Strength	Stress Ratio					
Major Shear	0.000	313.740	0.000					
Minor Shear	0.000	313.740	0.000					

END REACTION AXIAL FORCES				
	Left End Reaction	Load Combo	Right End Reaction	Load Combo
	-315.708	DSTLD2	-314.510	DSTLD2

Figure 5-4 Steel Stress Check Details Output

Composite Beam Design

After running the Composite beam design, the user can view the results using two different methods:

1. Text Format: File -> Print Tables -> Composite Beam Design
2. Graphically: Design -> Composite Beam Design -> Display Design Info.

Shown below in Figure 5-5, is an example of how the composite beam design results can be displayed graphically. You can see that the beams sizes chosen between Grid Lines D and E are larger because this is where the deck/mechanical loads were assigned. **A more complete discussion of Composite Beam Design can be found in Example VI.**

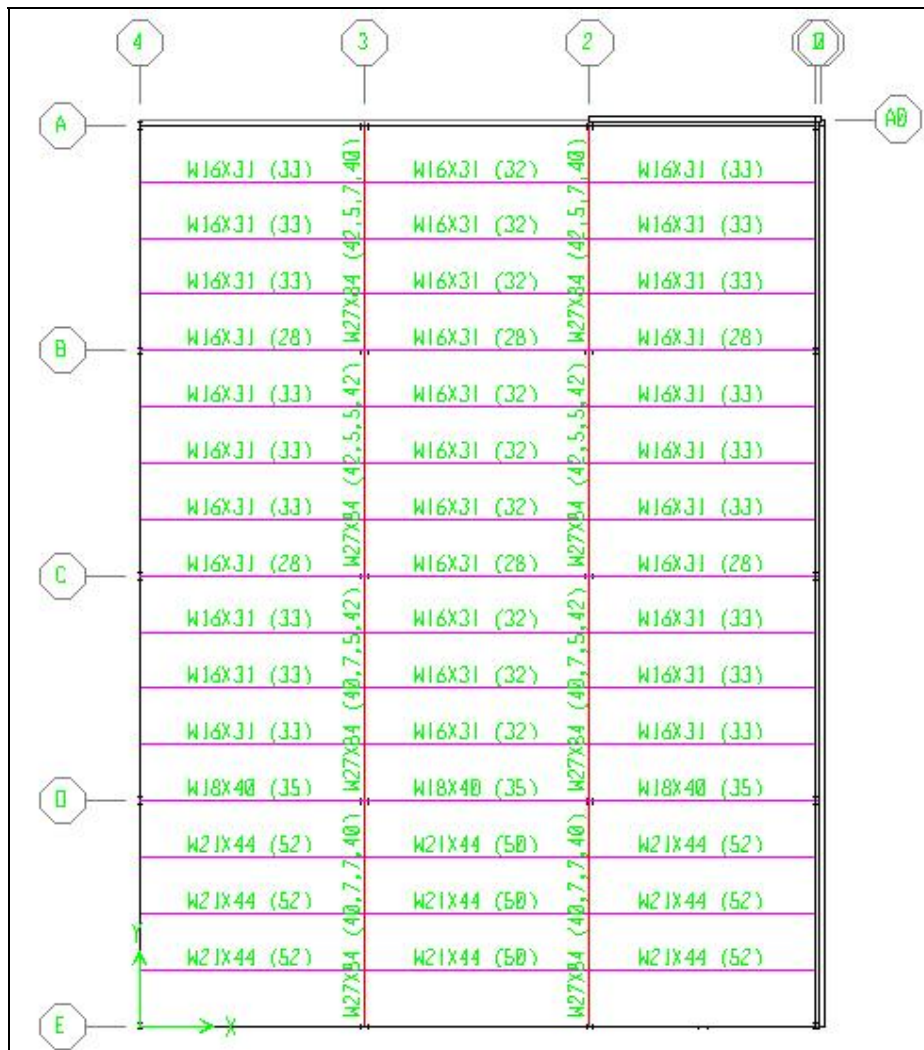


Figure 5-5 Composite Design Results Output

EXAMPLE VI Composite Beam Design

Description

Composite beam design is a powerful feature in ETABS. In this section, we will be discussing 2 composite beam design examples. The first model will be using the AISC LRFD 99 code. The second model will be using the AISC-ASD 89 code.

Significant Options of ETABS Exemplified

- ❑ Deck Properties
- ❑ Composite Beam Properties
- ❑ Construction Loading
- ❑ Auto Load Combinations
- ❑ Stud Spacing Options
- ❑ Partial Composite Limits
- ❑ Vibration

Example 1

This is an example of composite flexural member design using the LRFD 99 code. This model can be found in the AISC-LRFD 99 3rd Edition example 5.6. The model is a one-story 3-D steel frame structure, subjected to dead and live loads. We will look at the unshored and shored condition of this model.

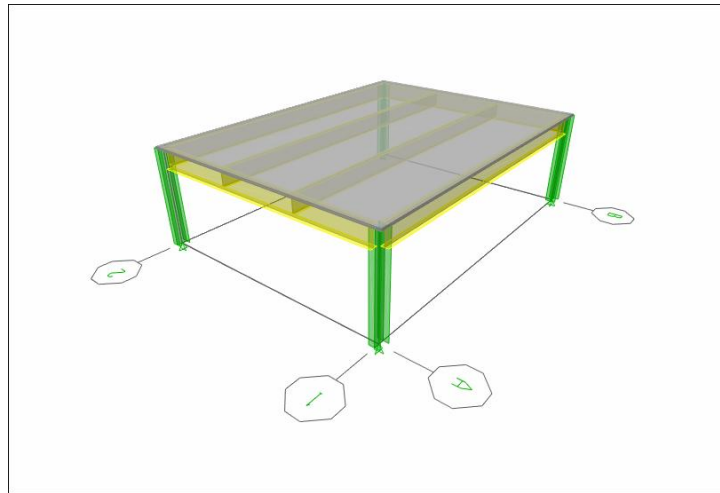


Figure 6-1 3-D view of Structure

Computer Model Definition

The beam span is 40 feet and the beam spacing is 10 feet. The modulus of elasticity is 29000 ksi. To see frame geometry, please refer to Figure 6-1.

Other parameters associated with the structure are as follows:

Coefficient of Thermal Expansion	= 6.500E-06
Poisson's ratio	= 0.3
Minimum Yield Stress, F_y	= 50 ksi
Minimum Tensile Strength, F_u	= 65 ksi

Deck Properties:

Slab Depth	= 3 in
Deck Depth	= 3 in
Rib Width	= 6 in
Rib Spacing	= 12 in

Service Loads:

Load	Service Load	Load Factor	Factored Load
Dead	.9 k/ft	1.2	1.1 k/ft
Live	2.5 k/ft	1.6	4.0 k/ft
Total	3.4 k/ft	-----	5.1 k/ft

Composite Beam Design Preferences

Click the Options menu -> Preferences -> Composite Beam Design command to bring up the Composite Beam Design Preferences form. On the Composite Beam Design Preferences form, click in the drop-down boxes or edit boxes and select or enter the desired value. Preferences control a variety of items that affect the look and feel of the program, the default behavior of the design postprocessors, and how the program considers live load reduction. Composite Beam Design Preferences are code specific. The Vibration tab gives the user an option to state the percent of live load plus reduced live load considered (in addition to full dead load) when computing weight supported by the beam for use in calculating the first natural frequency of the beam. Also, ETABS can consider Murray's minimum damping requirement as one of the criteria to be used for determining if a beam section is acceptable. For an explanation of the code-specific preference items and their associated possible values, default values, and a brief description of the items, click the **Help menu -> Documentation and Tutorials** command. Click **Technical Notes - Composite Beam Design**. Click your selected code and the **Preferences Technical Note** for that code.

Composite Beam Property Overwrites

Use the Design menu -> Composite Beam Design -> View/Revise Overwrites command to review or change the composite beam overwrites. You may not need to assign any composite beam overwrites; however the option is always available. If your design includes cover plates or user-defined shear connector patterns, assign them using the overwrites option. The composite beam design overwrites are basic properties that apply only to the beams to which they are specifically assigned. Some of the default overwrite values are based on composite beam preferences. Thus, define the preferences before defining the overwrites (and before designing or checking any composite beam).

Shear Studs is one of the tabs available in the composite beam overwrites. You can indicate if a user-defined shear connector pattern is defined, specify uniform spacing of shear studs along the beam and overwrite maximum/minimum longitudinal spacing of shear studs along the length of the beam. The shear stud form is shown in Figure 6-2:

<input type="checkbox"/>	User Pattern?	Yes
<input type="checkbox"/>	Uniform Spacing	13.5
<input type="checkbox"/>	No. Additional Sections	0
<input type="checkbox"/>	Single Segment?	
<input type="checkbox"/>	Stud Increase Factor	
<input type="checkbox"/>	Minimum Extra Studs	
<input type="checkbox"/>	Min Long Spacing	4.5
<input type="checkbox"/>	Max Long Spacing	36
<input type="checkbox"/>	Min Tran Spacing	3
<input type="checkbox"/>	Max Studs Per Row	3
<input type="checkbox"/>	Qn	0

Figure 6-2 Composite Beam Overwrites- Shear Studs

Under the Miscellaneous tab shown in Figure 6-2, the user can set the minimum and maximum value of percent composite connection considered for the beam. In the Results section of this example, we will show you where the Minimum and Maximum PCC is calculated. (See arrow #2 in Figure 6-3). Under the Vibration Tab, you have the option to indicate whether N_{eff} is user-specified or calculated by the program based on specified beam spacing. You can also state the effective number of beams resisting a heel drop impact.

Composite Beam Design Combinations

It is not necessary to run the analysis or select an element before using the Design menu -> Composite Beam Design -> Select Design Combo command.

Click the Design menu -> Composite Beam Design -> Select Design Combo command to open the Design Load Combinations Selection form. Review the default composite beam design load combinations defined by ETABS or designate your own design load combinations. Note that for composite beam design, separate design load combinations are specified for construction loading, final loading considering strength, and final loading considering deflection. Each of these types of design load combinations is specified in a separate tab in the form.

The default composite beam design load combinations have names such as DCMPC1, and so forth, as follows:

- **DCMPCn:** The D stands for Design. The CMP stands for composite. The last C stands for construction. The n item is a number. Design load combinations with this type of designation are the ETABS default for construction loading in composite design.
- **DCMPSn:** The D stands for Design. The CMP stands for composite. The S stands for strength. The n item is a number. Design load combinations with this type of designation are the ETABS default for strength considerations under final loading in composite design.
- **DCMPDn:** The D stands for Design. The CMP stands for composite. The last D stands for deflection. The n item is a number. Design load combinations with this type of designation are the ETABS default for deflection considerations under final loading in composite design.

Results

The resulting flexural required strengths for the total factored load are:

$$\begin{aligned}
 M_u &= \frac{wL^2}{8} \\
 &= \frac{(5.1 \text{ k/ft})(40\text{ft})^2}{8} \\
 &= \mathbf{1020 \text{ k-ft}}
 \end{aligned}$$

$$\begin{aligned}
 V_u &= \frac{wL}{2} \\
 &= \frac{(5.1 \text{ k/ft})(40\text{ft})}{2} \\
 &= \mathbf{102 \text{ k}}
 \end{aligned}$$

From Table 5-13, $Q_n = 26.1 \text{ k/stud}$ and the number of shear studs required is:

$$\frac{2\Sigma Q_n}{Q_n} = \frac{2(810 \text{ k})}{26.1 \text{ k/stud}} = \mathbf{62.1 \rightarrow 64 \text{ studs}}$$

After you run the analysis and composite beam design, right-click on a beam element to activate the interactive composite beam design box. Click on Details to see a summary of the beam design. The AISC-LRFD example calculated 64 shear stud connectors required. If you look below under the Summary in Figure 6-3, you can see ETABS calculates 65 studs required. (See Arrow #1):

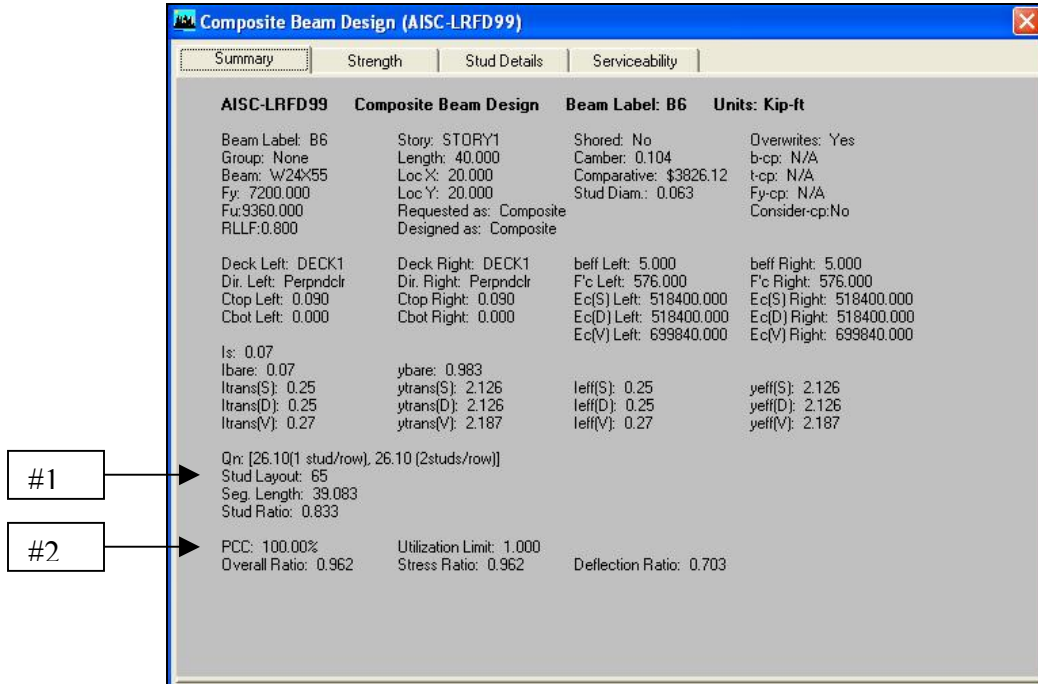


Figure 6-3 Composite Beam Design Details

Example 2

This is an example of composite flexural member design using the AISC-ASD 89 code. It is a fully composite interior floor of an office building. This model can be found in the AISC-ASD 89 9th Edition, example 16. The model is a one-story 3-D steel frame structure, subjected to dead and live loads.

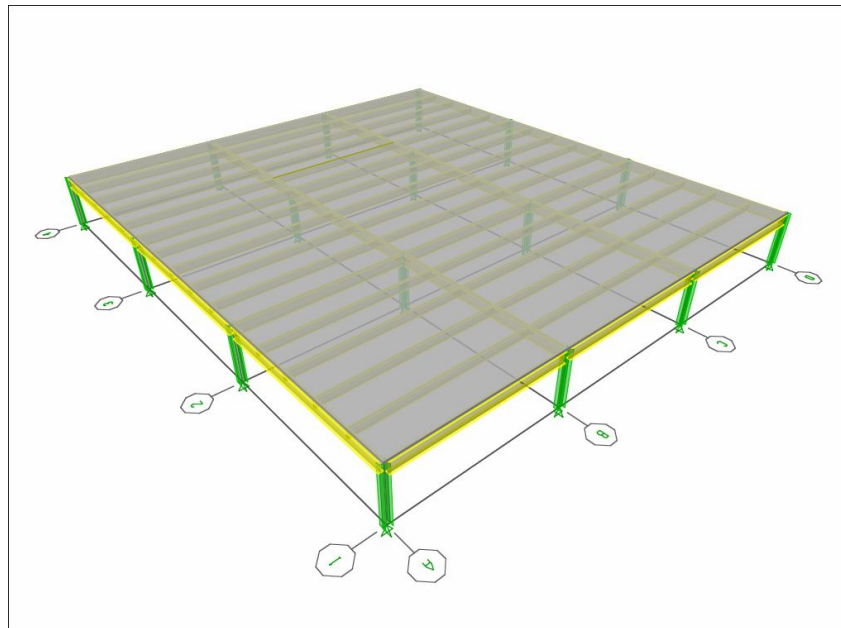


Figure 6-4 3-D View of Structure

Computer Model Definition

The one story structure has a span length of 36 feet with beam spacing of 8 feet. . Kip-inch-second units are used. The modulus of elasticity is 29000 ksi. To see frame geometry, please refer to Figure 6-4.

Other parameters associated with the structure are as follows:

Weight of Concrete	= 145 pcf
Poisson's ratio	= 0.3
Minimum Yield Stress, F_y	= 36 ksi
Minimum Tensile Strength, F_u	= 65 ksi
Concrete Compressive Strength, f'_c	= 3 ksi

Deck Properties:

Slab Depth	= 2 in
Deck Depth	= 2 in
Rib Width	= 2.5 in
Rib Spacing	= 6 in
Stud Diameter	= .75 in.
Stud Height	= 3.5 in.

Building Loading

Live Load	= 100 psf
Partition Load	= 20 psf
Ceiling Load	= 8 psf

Results

After you run the analysis and composite beam design, right-click on a beam element to activate the interactive composite beam design box. Click on *details* to see a summary of the beam design. The AISC-ASD example calculated a W21x44 beam size with 30 shear stud connectors. Figure 6-5 shows that ETABS calculated the same W21x44 beam size with 31 shear stud connectors. This can be displayed by selecting Design->Composite Beam Design->Display design info->select beam sections and shear stud layout.

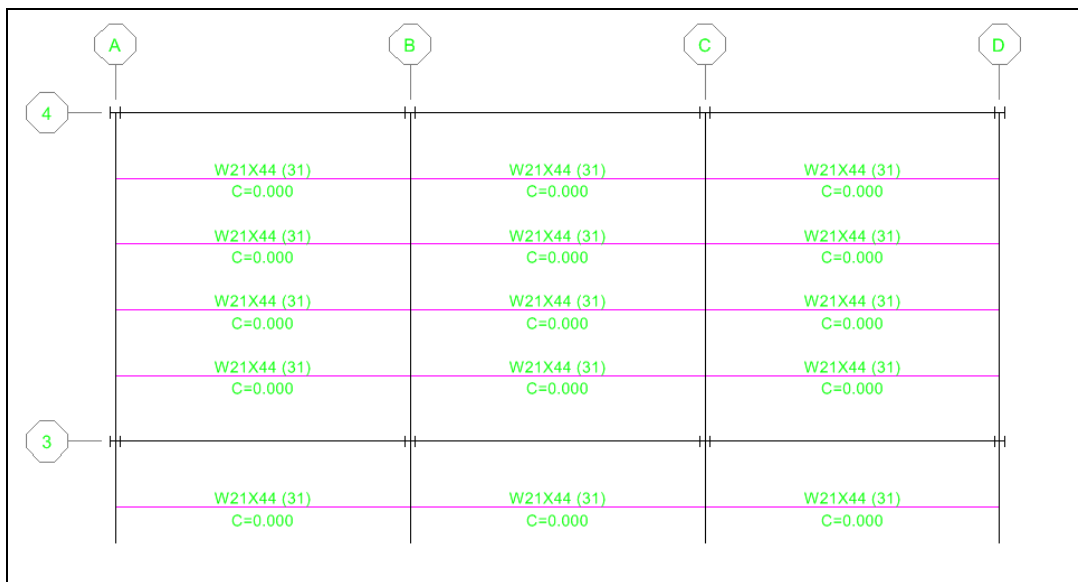


Figure 6-5 Partial Plan-Composite Beam Design Results

EXAMPLE VII

Nonlinear Time History Analysis

Description

This example consists of 3 simple 2-D frames, subjected to a nonlinear Time History Analysis with the use of link elements. The first frame has no link properties assigned to it. The second frame contains damper link properties, as it's lateral force resisting system. The third frame contains isolator link elements. The frame geometry is shown in Figure 7-1.

Significant Options of ETABS Exemplified

- Time History Function Definition
- Time History Case Definition
- Damper/ Isolator Link Properties
- Creation of .AVI Files
- Time History Traces
- Response Spectrum Curves

Computer Model Definition

These 3 frames contain one bay that is spaced 24 feet apart. Each frame is 3 stories high. Each story level is 12 feet in height. Kip-inch-second units are used. The modulus of elasticity is 29000 ksi. A point mass of .25 is assigned to each joint in the X and Y-directions.

Link Element Definition

Damper and Isolator Link Element

Click the Define menu -> Link Properties command to bring up the Define Link Properties form. Click the Add New Property button to access the NL link property data form. Select the Damper or Isolator link element from the pull-down menu. Enter the desired values for Mass, Weight, and Rotational Inertia. Note that the mass is for the entire length of the element and is not mass per unit volume. Typically, link elements can have two sets of properties assigned to them:

- **Linear properties**, can be assigned to linear link elements only. The linear property that you specify for each of the six degrees of freedom of a linear link element is an effective stiffness. This is simply a spring stiffness.
- **Nonlinear dynamic properties**, which are used for nonlinear dynamic (time history) analysis. Note that you must have the nonlinear version of ETABS to use the nonlinear dynamic link properties.

The linear properties that you specify for each of the six degrees of freedom of all types of link elements other than linear link elements are an effective stiffness and effective damping. In such cases, the effective stiffness is, again, simply a spring stiffness. The effective damping specifies dashpot-type damping; it is not a specification of percent critical damping.

In a linear analysis ETABS converts the specified effective damping for a link element to modal damping. It then adds the modal damping calculated for all link elements in the model that have effective damping specified to any modal damping already specified for the structure as a whole to get the final modal damping. ETABS reports this final modal damping in the printed analysis output for building

modes. To get this output, click the File menu-> Print Tables -> Analysis Output command and check the Building Modal Info check box.

Link Element Assignment

Damper Link Element

Go to Draw->Draw Line Object->Draw Line and in the Properties of Object Box, select a NONE Property. Draw diagonal line on every level of the middle frame. Select these lines and go to Assign->Frame/Line-> Link Property and select the Damper property previous defined.

Isolator Link Element

Select the bottom joints on the 3rd frame and go to Assign->Joint/Point->Restraints and select all restraints in all 6 directions. Select these same joints and go to Assign->Joint/Point-> Link Properties. Select the Isolator property previously defined.

The model should look like Figure 7-1:

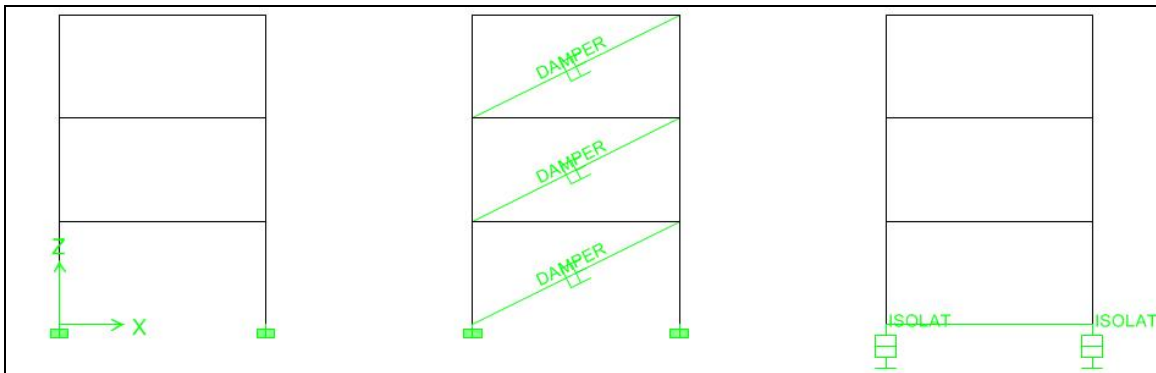


Figure 7-1 *Three Frame Time History Model*

Time History Function and Case Definition

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. You can view time history graph by clicking on the Display Graph button. Fill in the Function File information:

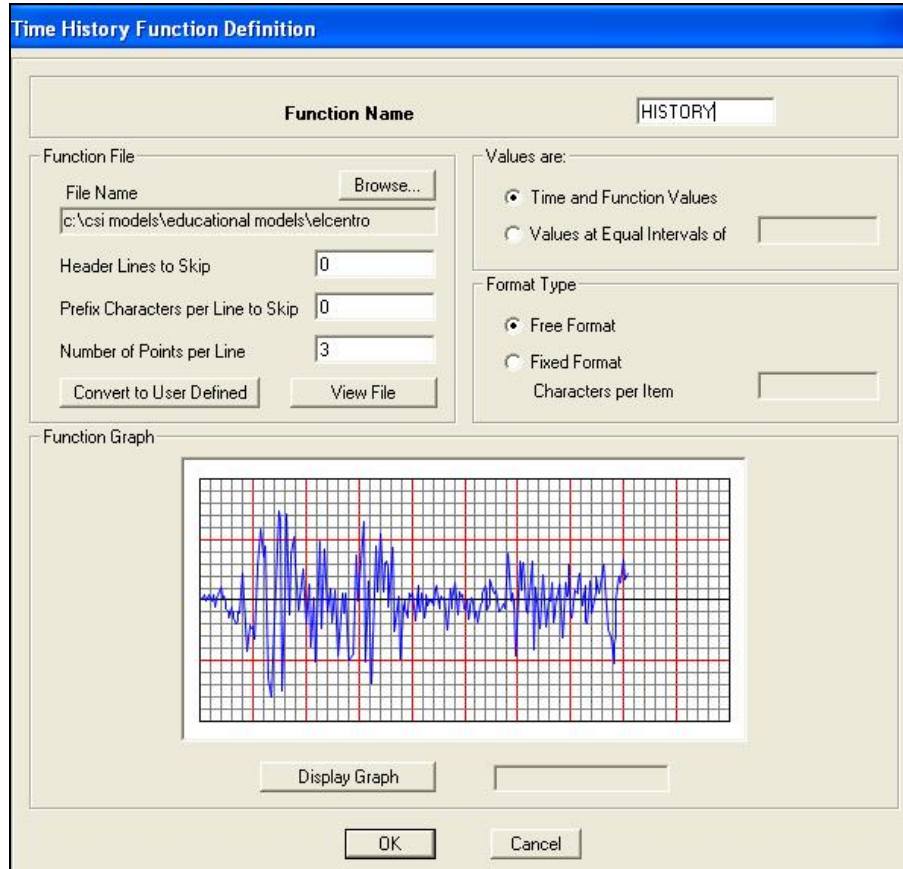


Figure 7-2 *Time History Function Definition*

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 .002 seconds.

Results

Creation of .AVI video files

After running the analysis (under menu->Analyze-> Run Analysis), ETABS has the capability of creating an .AVI animation file. Use the File menu -> Create Video-> Time History Animation command to create videos in ETABS showing the movement of the structure during any time history analysis that has been run. Enter the input data for the video file:

The Start Time, End Time, Time Increment boxes must be filled to create the video file. Use the default or define start and end times and time increments. The time increment controls how many different pictures (frames) of the deformed shape of the structure are created. For example, a time increment of 0.1 means a picture (frame) of the deformed shape is created for every one-tenth second of the time history.

Time History Traces

After running a time history analysis, click the Display menu -> Show Time History Traces command to access the Time History Display Definition form. Use the form to specify the appropriate data to plot various time history curves.

A time history trace is simply a plot of a vertical time history function versus a horizontal time history function. The vertical time history function can be any defined time history function. Although the horizontal time history function defaults to Time, it can be any defined time history function.

When you click the Display menu -> Show Time History Traces command, ETABS automatically creates time history display functions for all selected objects. Use the Time History Display Definition form to define additional time history display functions as desired. To display the time history trace for a particular point, for example, it is easiest to select that joint before selecting the Display menu -> Show Time History Traces command. ETABS creates the time history display function automatically for that point. You can easily modify the component of displacement displayed if ETABS did not default to the one you want.

We are interested in the deflection values if joints 2,4 and 6 in this example. Add these joints to the Vertical function column in the Time History Traces dialogue box. Click Define functions and select joint 2 and press Modify/Show TH function. Make sure the Displacement option is selected under the Vector Type. Repeat this step for joints 4 and 6. We are now ready to view the time history traces. See Figure 7-3:

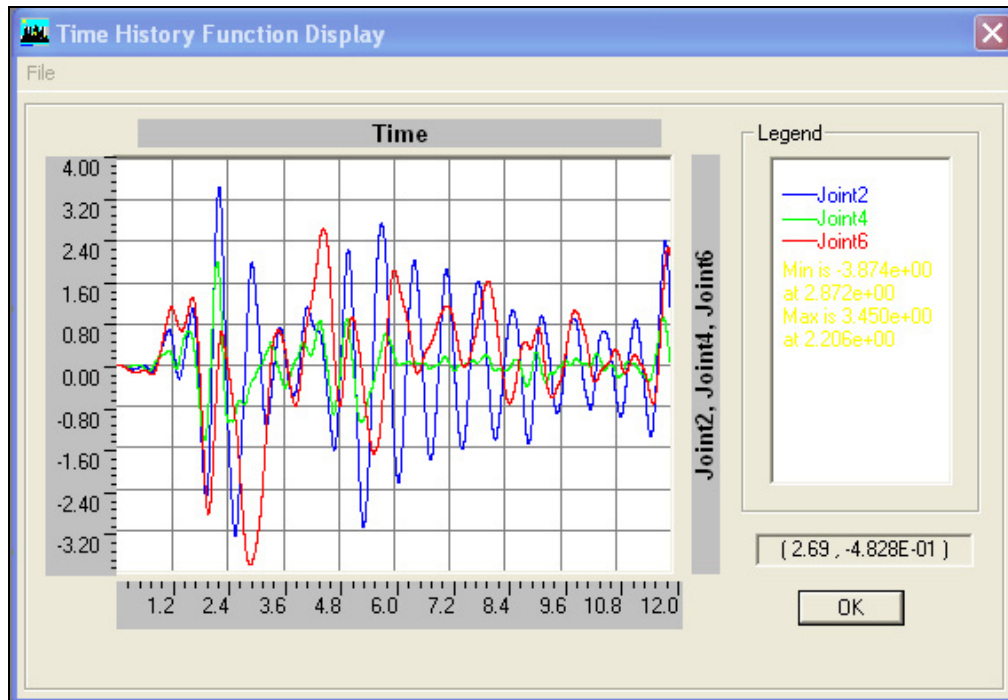


Figure 7-3 Time History Trace Plot

You can see that joint 2 has the most deflection, followed by joint 4 (the isolator frame system) and joint 6 (damped frame system).

Response Spectrum Curves

After running a time history analysis, select one or more point objects and click the Display menu -> Show Response Spectrum Curves command to access the Response Spectrum Generation form. Use the form to specify the appropriate data to plot the various response spectra.

Important Note: The response spectrum that you plot using this command is based on a time history that you have previously run. It has *nothing* to do with any response spectrum analysis that you may have run.

There are many options available when viewing Response Spectrum Curves. These include Axes, Damping and Period/Frequency definition. See Figure 7-3 for a sample Response Spectrum Curve.

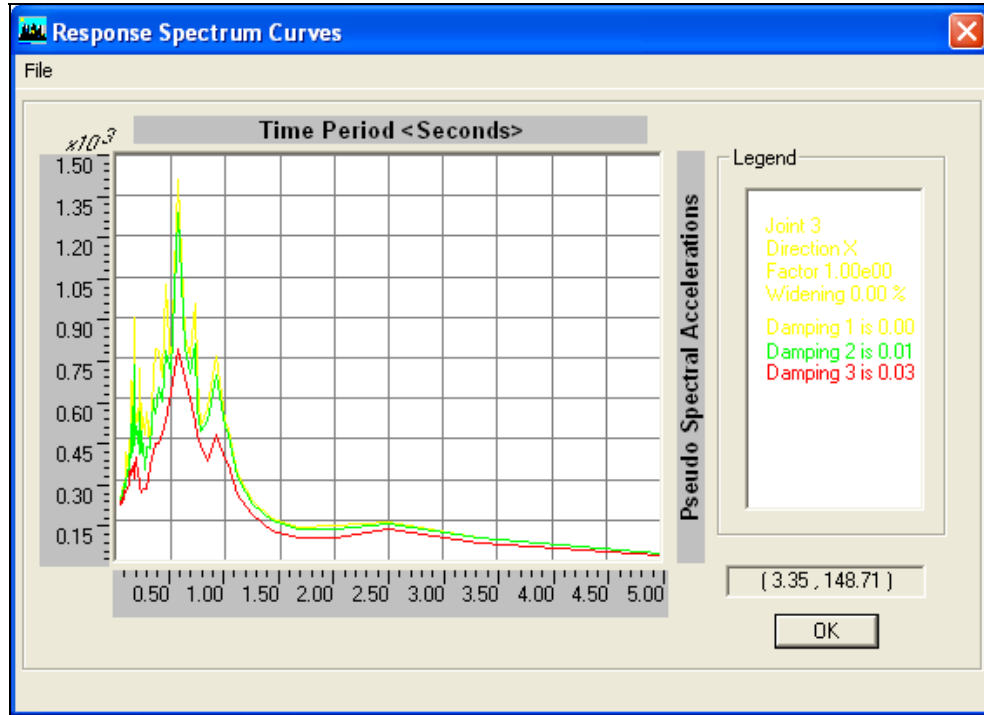


Figure 7-4 *Response Spectrum Curves Plot*

EXAMPLE VIII Pushover Analysis Example

Description

This is a 2-D four-story steel frame structure, subjected to a static nonlinear pushover analysis. Each story is 12 feet high and each bay is 24 feet long. Nonlinear frame hinges are assigned to every frame element. The frame geometry is shown in Figure 8-1.

Significant Options of ETABS Exemplified

- ❑ Static Frame Hinge Definition
- ❑ Static Nonlinear Pushover Case Definition
- ❑ Pushover Curves

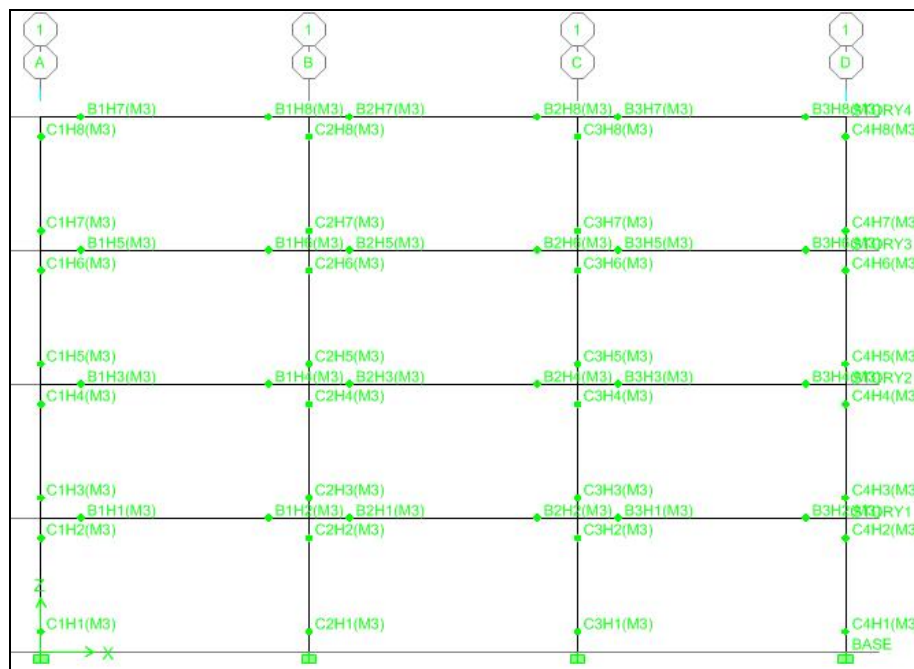


Figure 8-1 3-D view of Pushover Structure

Computer Model Definition

Static nonlinear (including pushover) analysis is a very powerful feature offered only in the nonlinear version of ETABS. In addition to performing pushover analyses for performance-based seismic design, this feature can be used to perform general static nonlinear analysis and the analysis of staged (incremental) construction. Performing any nonlinear analysis takes time and patience.

Hinge Property Definition

Select the Define menu -> Frame Nonlinear Hinge Properties command to display the Define Frame Hinge Properties form. Frame Nonlinear Hinge Properties are used to define nonlinear force-displacement and/or moment-rotation behavior that can be assigned to discrete locations along the length of frame (line) elements. These nonlinear hinges are only used during static nonlinear (pushover) analysis. For all other types of analysis, the hinges are rigid and have no effect on the linear behavior of the member.

The built-in default hinge properties for steel members are generally based on Tables 5.4 and 5.8 in FEMA-273. The built-in default hinge properties for concrete members are generally based on Tables 9.6, 9.7 and 9.12 in ATC-40. **You should review any generated properties for their applicability to your specific project.**

Default hinge properties cannot be modified. They also cannot be viewed because the default properties are section dependent. The default properties cannot be fully defined by the program until the section to which they apply is identified. Thus, to see the effect of the default properties, the default property should be assigned to a frame element, and then the resulting generated hinge property should be viewed.

For this model, we will use user defined hinge properties. User-defined hinge properties can be based on default properties or they can be fully user-defined. When user-defined properties are based on default properties, the hinge properties cannot be viewed because the default properties are section dependent. When user-defined properties are not based on default properties, the properties can be viewed and modified.

Hinge Property Assignment

Select the Assign menu -> Frame/Line -> Frame Nonlinear Hinges command to bring up the Assign Frame Hinges (Pushover) form. Choose a defined hinge property from the Hinge Property drop-down box. Type in a distance in the Relative Distance box (note the value must be less than 1). Click the Add button.

A hinge assignment consists of a hinge property and a location for that hinge along the frame element. The location is specified as a relative distance along the full length of the element measured from the i-end. The relative distance is equal to the distance from the i-end of the element to the hinge location, divided by the length of the frame element. Hinges may not be located within the end offsets. If the specified relative distance places a hinge within an end offset, the hinge is automatically moved to the inside face of that offset.

Static Nonlinear Pushover Case Definition

Select the TDefine menu -> Static Nonlinear/Pushover Cases command to display the Define Static Nonlinear Cases form. Static nonlinear/pushover analysis is a very powerful feature available only in the nonlinear version of ETABS. In addition to performing pushover analyses for performance-based seismic design, this feature can be used to perform general static nonlinear analysis and the analysis of staged (incremental) construction. Before an analysis can be run, the static nonlinear case must be defined.

There are many options available when defining pushover case data. In this model we will be using most of the default values supplied by the program. We will enter a value of 25 inches in the Push to Displacement Magnitude box. Check the Push To Display. Magnitude check box to perform a displacement-controlled analysis. The load combination specified in the Load Pattern area of the form is applied, but its magnitude is increased or decreased as necessary to keep the control displacement increasing in magnitude. This option is useful for applying lateral load to the structure, or for any case where the magnitude of the applied load is not known in advance, or when the structure can be expected to lose strength or become unstable. If you want more specific information about each section of this data form, go to the Help section in ETABS and look under Static Nonlinear Case Data.

Results

Pushover Curves

After you have run a static nonlinear analysis, click the Display menu -> Show Static Pushover Curve command to access the Pushover Curve form. The static pushover curve is a single force-displacement curve obtained from a static nonlinear analysis. View the pushover curve in the Acceleration-Displacement Response Spectrum (ADRS) format and overlay it with various response spectra, thus allowing you to perform capacity-spectrum analysis in ETABS.

In our example, we view the deflected shape of the structure by selecting Display->Show Deformed Shape->select the pushover case-> Step 1. By using the arrows on the bottom right hand corner of the screen, we can toggle step by step to see

the frames hinging. The colors of dots on the hinges change with each step. Each color represents a different stage in the pushover analysis. For example, a blue dot represents that the frame has reached the Immediate Occupancy Stage, a green dot represents Collapse Prevention and a red dot represents Failure of the structure. Please refer to Figure 8-2 and 8-3.

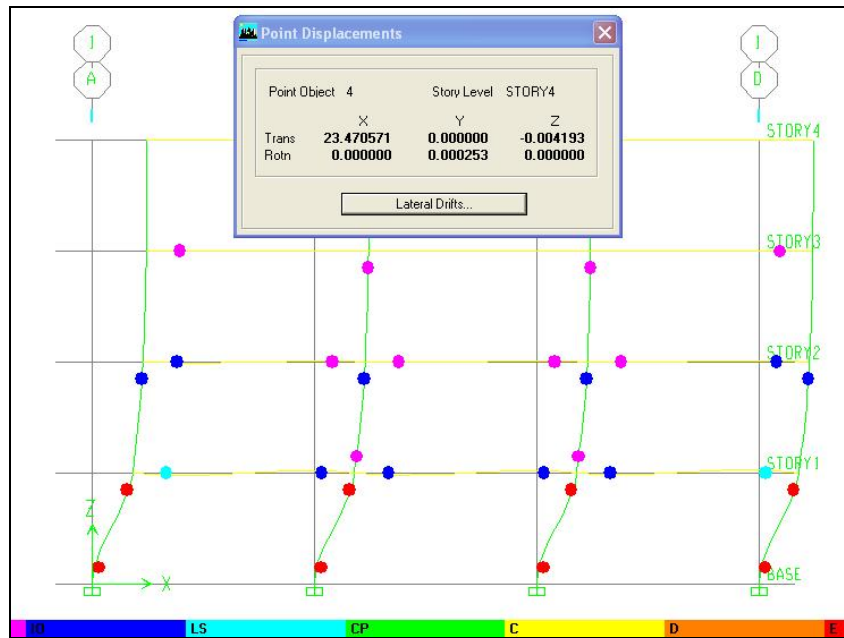


Figure 8-2 *Deflected Shape of Structure*

As seen in Figure 8-2, the columns on the bottom level show red dots on the I and J ends which signifies they have reached their capacity and the structure has collapsed. The deflection at the top level is 23.47 inches. The pushover curve in Figure 8-3 displays the collapse occurring at the specified displacement.

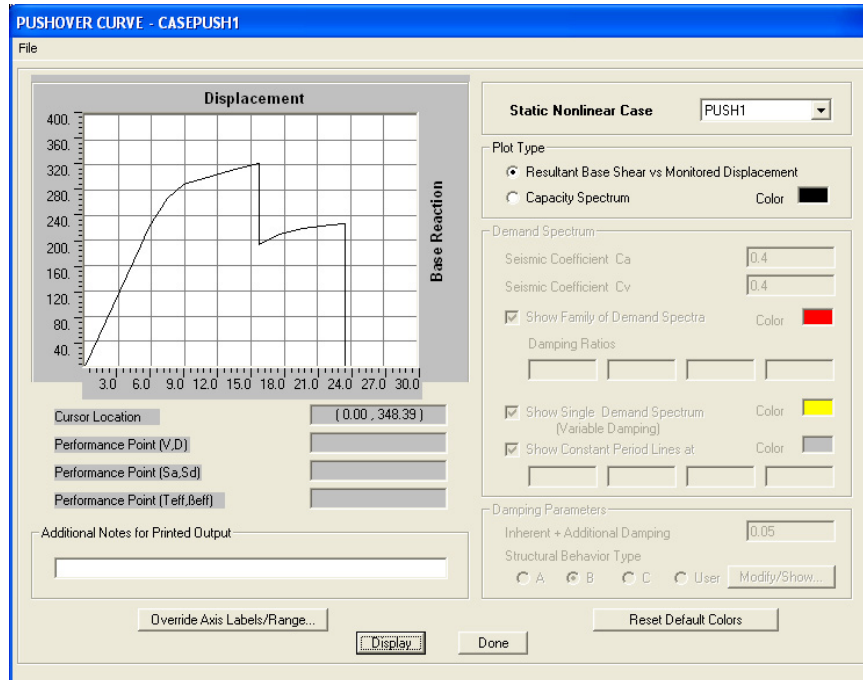


Figure 8-3 Pushover Curve

About the Speakers

ROBERT TOVANI

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 a Principal at Degenkolb Engineers.

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 with CSI on example models and documentation.

ATIF HABIBULLAH

Atif Habibullah has worked as a Software Support Engineer at CSI for the past 4 years. He has extensive experience in modeling of a variety of structural systems, solving special modeling problems and interpretation of analysis results. Prior to working at CSI, Atif has worked in a design office for 6 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.