

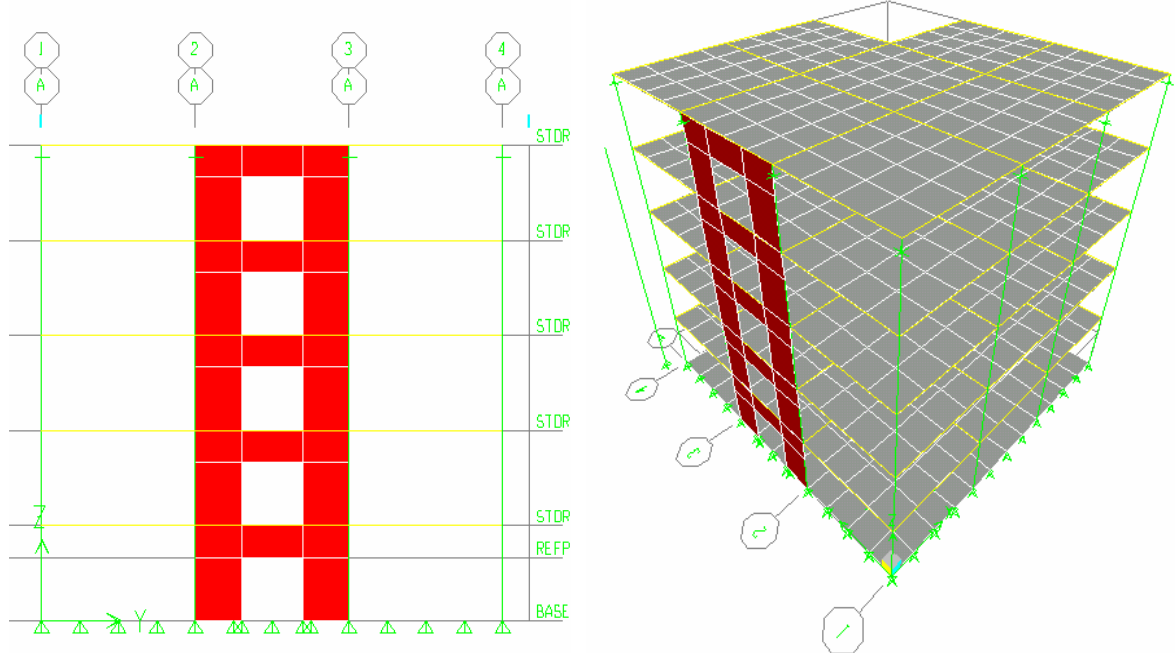
---

# ETABS Example

---

**Static, Dynamic Analysis**  
*and*

**Design of RC Building with Shear Wall**  
(5 Story Building, US Units)



---

ACECOMS, AIT

---

## Table of Content for Example

Objective.....	4
Problem .....	4
Part A: Modeling, Static Analysis and Design .....	7
Step-by-Step Solution .....	7
1. Define Plan Grids and Story Data.....	7
2. Define Material Properties.....	9
3. Define Frame Sections.....	10
4. Define Slab Sections.....	16
5. Define Load Cases .....	17
6. Draw Beam Objects (Frame Members) .....	17
7. Draw Column Objects (Frame Members).....	19
8. Assign Slab Sections.....	21
9. Assign Restraints .....	23
10. Assign Slab Loads .....	24
11. View Input Data in Tabular Form.....	27
12. Run the Analysis.....	28
13. View Analysis Results Graphically .....	29
14. Design Concrete Frame Elements.....	32
Part B: Dynamic Analysis and Design .....	35
Step-by-Step Solution .....	35
1. Unlock the Model .....	35
2. Define Response Spectrum Function.....	35
3. Define Response Spectrum Cases.....	37
4. Run Analysis.....	38
5. View dynamic analysis results.....	39
6. Design Concrete Frame .....	40
Part C: Design of Shear Wall .....	42

Define Wall Section .....	45
7. Draw Wall Sections.....	46
8. Define Reference Planes and Lines .....	47
9. Add Wall Opening.....	48
10. Assign Piers Labels .....	50
11. Assign Spandrels Labels.....	51
12. Run Analysis .....	52
13. View Shear Wall Results.....	53
14. Design Shear Walls .....	54

## Objective

To demonstrate and practice step-by-step on the modeling, static analysis and design of 5 story reinforced concrete building.

## Problem

### Part A:

Carry out the modeling, **static analysis and design** of 5 story reinforced concrete building subjected to static loads.

### Part B:

Use the model from part A to **analyze and design for dynamic load** (UBC97 response spectrum load in Y direction).

### Part C:

Model, analyze and design **Shear Wall**.

### Material Properties

Strength of concrete ( $F_c'$ )	=	4 ksi
Yield strength of main reinforcement ( $F_y$ )	=	60 ksi
Yield strength of shear reinforcement ( $F_{ys}$ )	=	60 ksi
Young Modulus of concrete ( $E_c$ )	=	3600 ksi

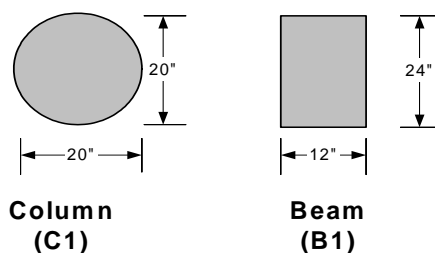
### Loading

Load Cases	Type	Details
DEAD	Dead load	Use Self Weight Multiplier
SUPERDL	Dead load	Slabs: 35 psf Perimeter Beams: 250 plf
LIVE	Live load	Slab: 100 psf
SPEC1 (for Part B)	Dynamic load	UBC97 response spectrum in Y direction $C_a = 0.1$ , $C_v = 0.1$

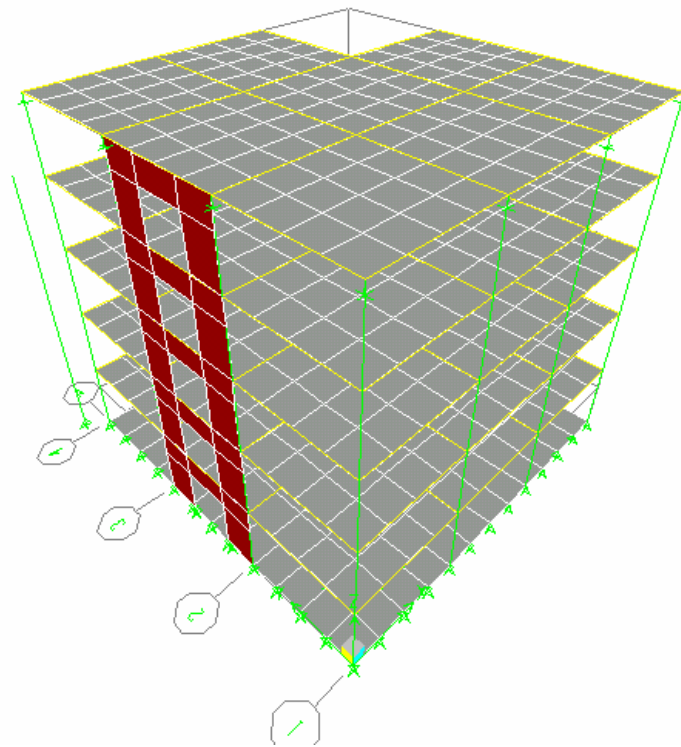
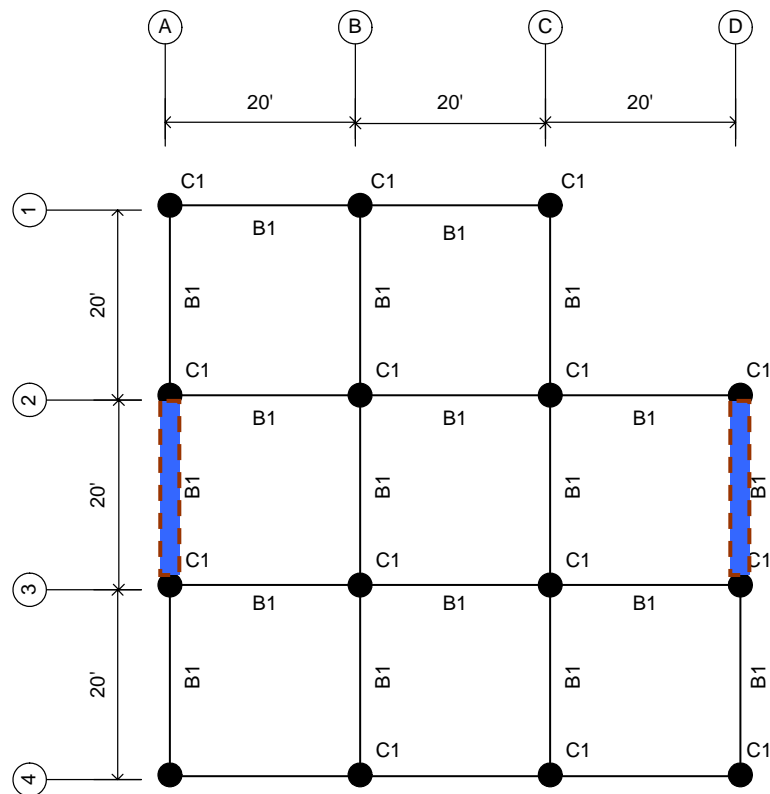
### Slab Section

Reinforced concrete solid slab, thickness = 6 inch

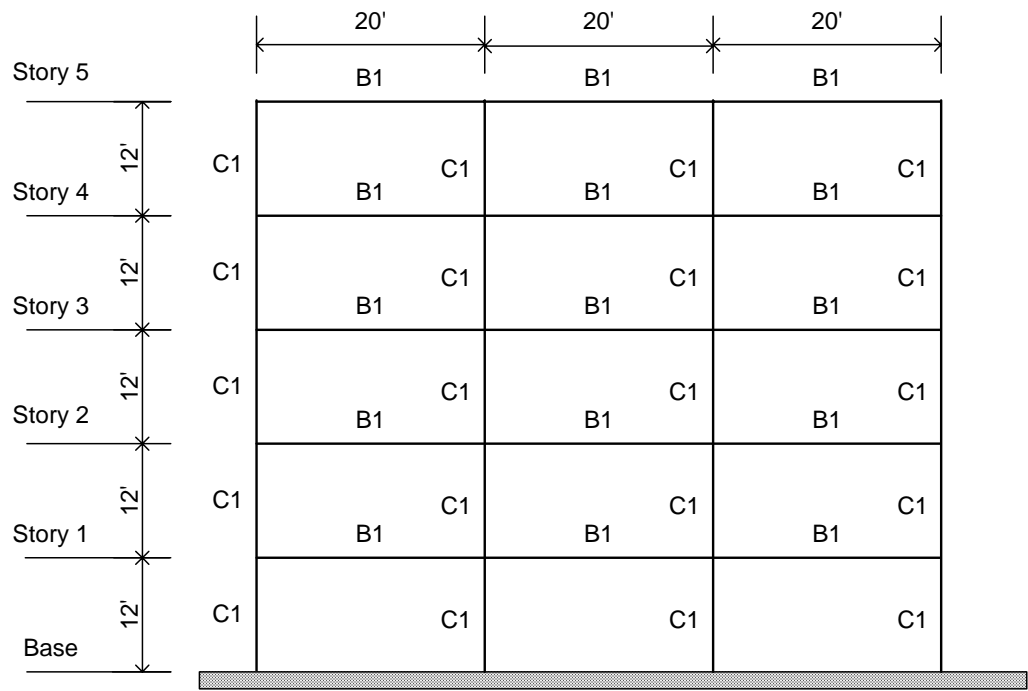
### Frame Sections



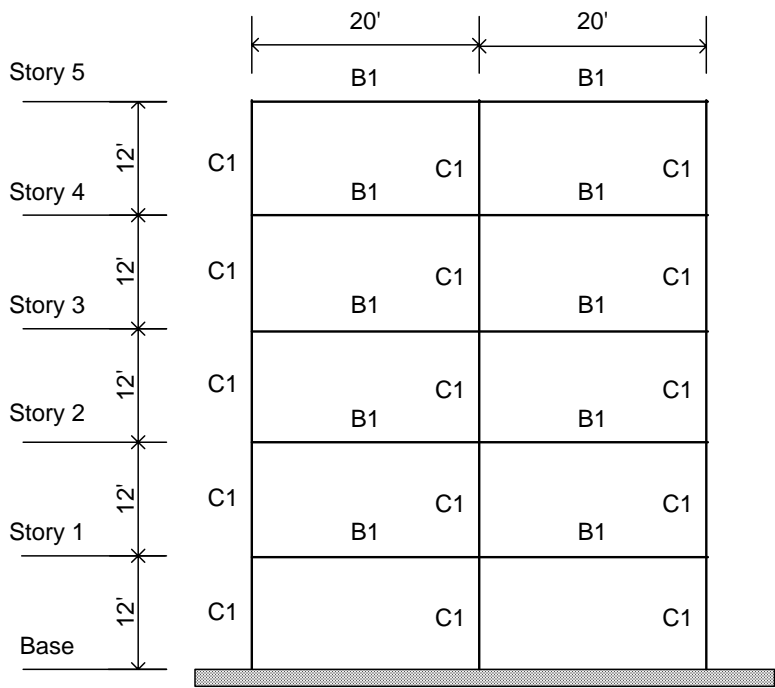
3D View of the building

Plan and Elevation View  
Plan View at Roof Floor

Elevation At Grid Lines 2, 3, 4, A, B and C (Shear Wall Not Shown)



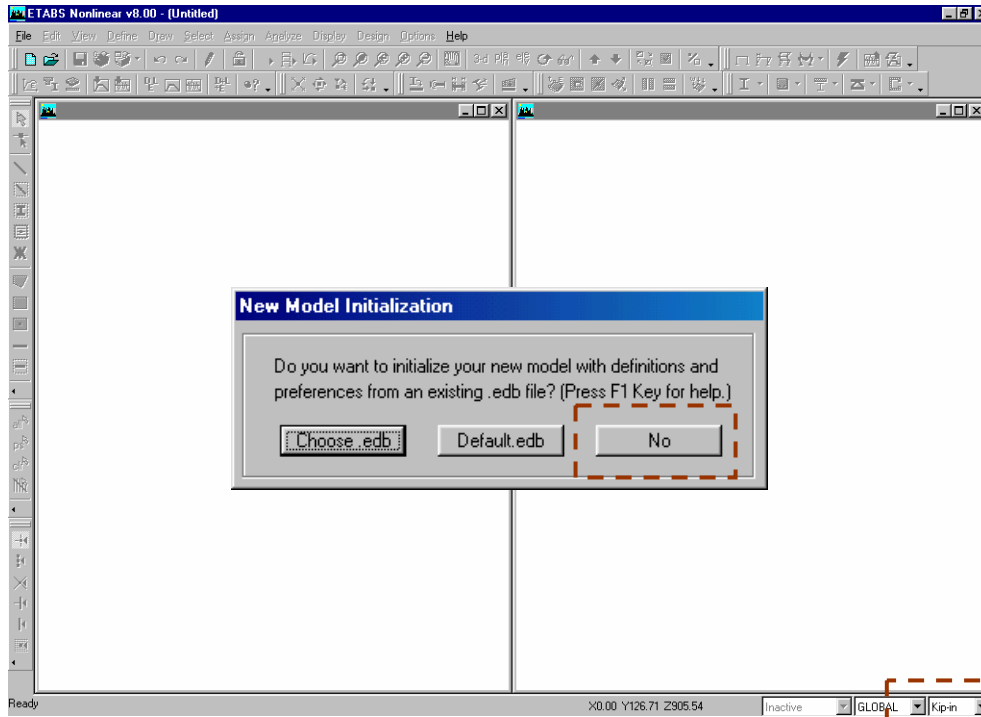
Elevation at Grid Line 1 and D (Shear Wall Not Shown)




# Part A: Modeling, Static Analysis and Design

## Step-by-Step Solution

### 1. Define Plan Grids and Story Data



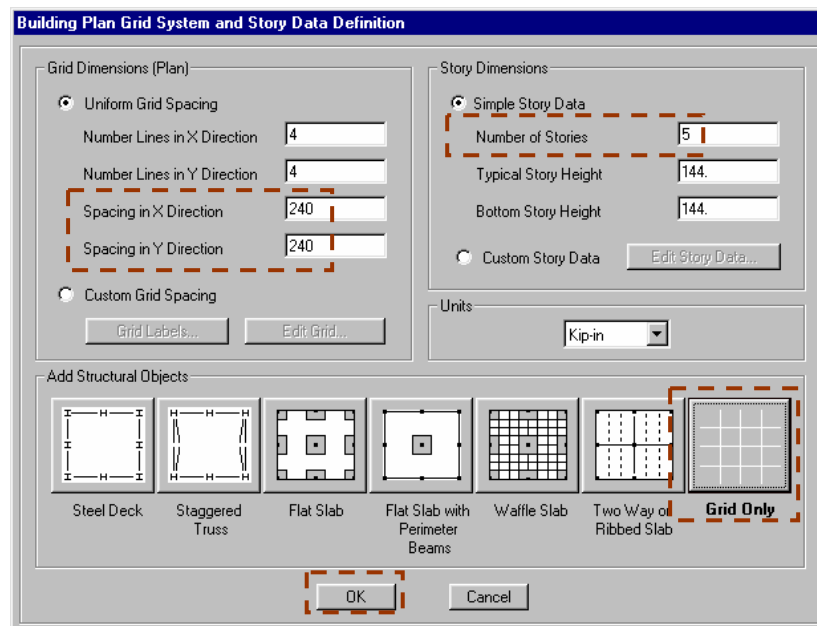
**Step 1-1:** Start ETABS by clicking on the appropriate desktop shortcut or by selecting ETABS from your Windows Start menu. Select "Kip-in" from drop-down menu box in bottom-right screen and click on  in top tool bar or go to **File > New Model** in main menu. Click **No** to start new model without opening any existing file.

- ✓ Click the **Default.edb** button. This means that the definitions and preferences will be initialized (get their initial values) from the Default.edb file that is in the same directory as your ETABS.exe file. If the Default.edb file does not exist in this directory then the definitions and preferences are initialized using ETABS built-in defaults.

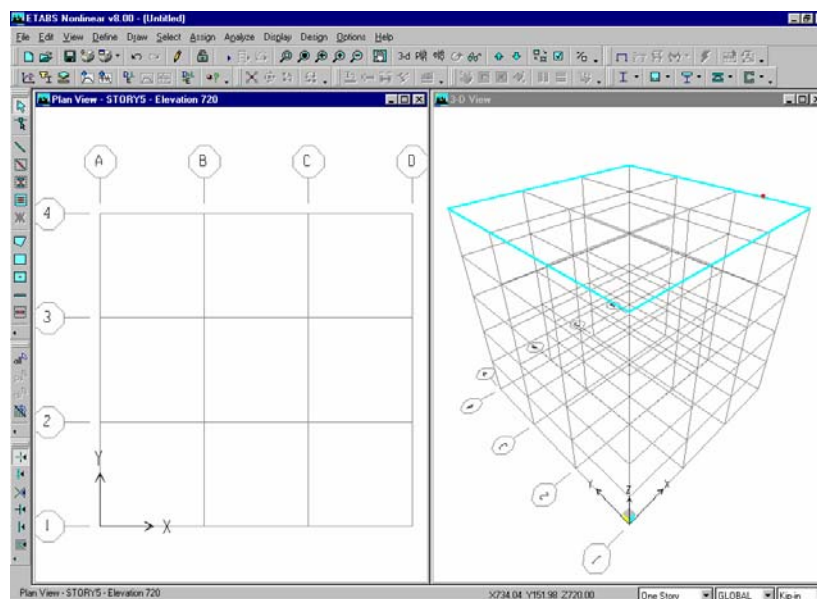
You should create your Default.edb file such that you most commonly click this button.

- ✓ In some cases you may want to click the **Choose .edb** button and specify a different file from which the definitions and preferences are to be initialized. For example, a certain client or project may require certain things in your model to be done in a certain way that is different from your typical office standards. You could have a specific .edb file set up for this client or project which could then be used to initialize all models for the client or project.

Click the **No** button if you just want to use the built-in ETABS defaults.

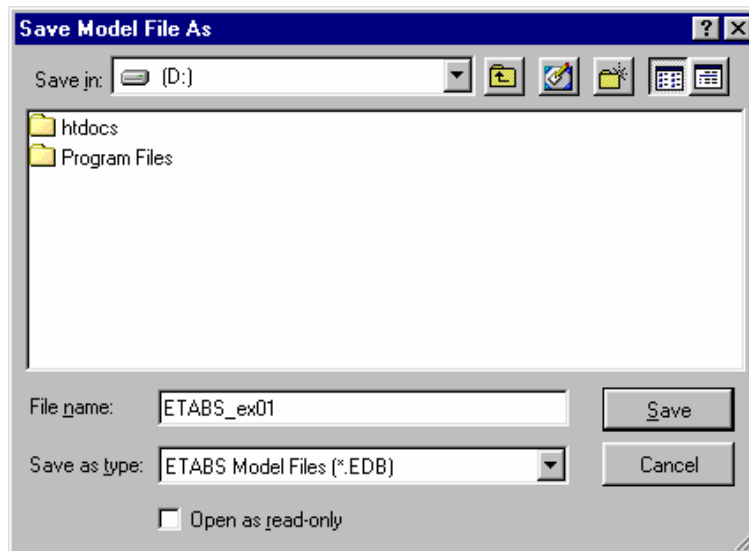



**Step 1-2:** Enter “240” or “20 ft” into **Spacing in X Direction** and **Spacing in Y Direction**, “5” into **Number of Stores**, select **Grid Only** from **Add Structural Objects** and click **OK**.



**Step 1-3:** After clicking **OK**, ETABS creates Grid System based on the parameters specified in the previous step and displays in “Plan View” and “3-D View” window.



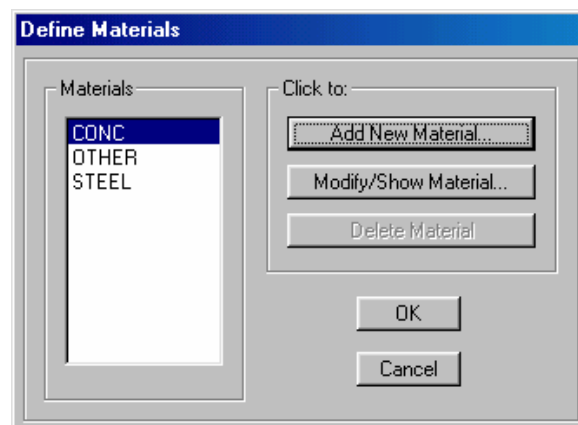



**Step 1-4:** Save the project by clicking on  or **File > Save** from main menu, enter **File name** = "ETABS\_ex01" and click Save.

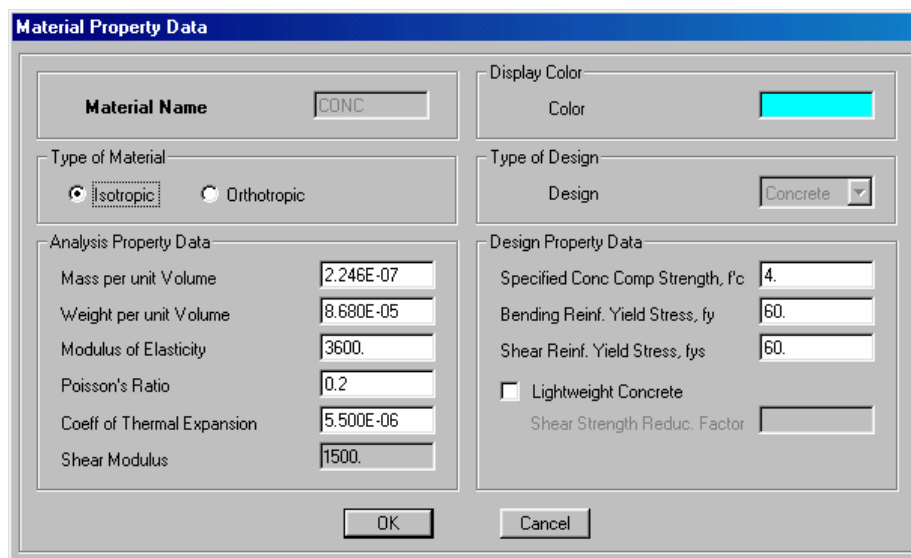
## 2. Define Material Properties

Default concrete material ("CONC") has been used for this example.

Strength of concrete ( $F_c'$ )	=	4 ksi
Yield strength of main reinforcement ( $F_y$ )	=	60 ksi
Yield strength of shear reinforcement ( $F_{ys}$ )	=	60 ksi
Young Modulus of concrete ( $E_c$ )	=	3600 ksi



**Step 2-1:** Click on  in tool bar or from **Define > Material Properties** in main menu. Select **CONC** and click on **Modify/Show Material** to view/revise the material properties.



**Material Property Data**

**Material Name**: CONC

**Type of Material**: ☒ Isotropic ☐ Orthotropic

**Analysis Property Data**

Mass per unit Volume	2.24E-07
Weight per unit Volume	8.680E-05
Modulus of Elasticity	3600.
Poisson's Ratio	0.2
Coeff of Thermal Expansion	5.500E-06
Shear Modulus	1500.

**Design Property Data**

**Display Color**: Color [Blue]

**Type of Design**: Design [Concrete]

Specified Conc Comp Strength,  $f'_c$ : 4.

Bending Reinf. Yield Stress,  $f_y$ : 60.

Shear Reinf. Yield Stress,  $f_{ys}$ : 60.

☐ Lightweight Concrete

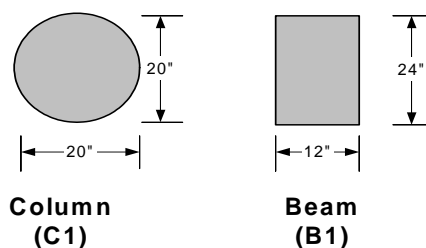
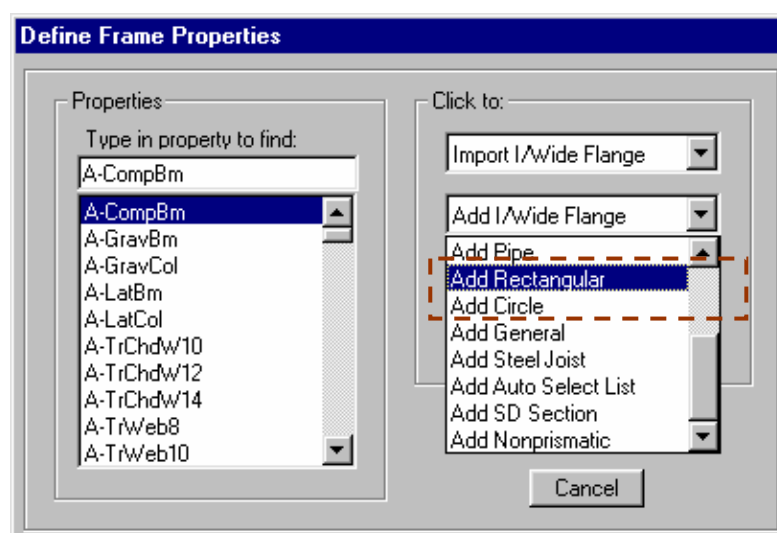
Shear Strength Reduc. Factor: [ ]

OK Cancel

**Step 2-2:** Accept the default properties by clicking **Cancel** or **OK** 2 times

### 3. Define Frame Sections

2 concrete frame sections ("B1" for rectangular beams and "C1" for circle columns) need to be defined for this example.

**Define Frame Properties**

**Properties**


Type in property to find:

- A-CompBm
- A-GravBm
- A-GravCol
- A-LatBm
- A-LatCol
- A-TrChdw10
- A-TrChdw12
- A-TrChdw14
- A-TrWeb8
- A-TrWeb10

**Click to:**

- Import I/Wide Flange
- Add I/Wide Flange
- Add Pipe
- Add Rectangular
- Add Circle
- Add General
- Add Steel Joist
- Add Auto Select List
- Add SD Section
- Add Nonprismatic

Cancel

**Step 3-1:** Click on  in tool bar or from **Define > Frame Sections** in main menu to start frame section definition properties editor. Select "Add Rectangular" from second drop-down menu to add rectangular beam.

## Reinforcing Information for Beams

For concrete beams there are two types of reinforcing information that you specify. They are rebar cover and reinforcement overrides. Rebar cover is specified at the top and bottom of the beam. The top cover is measured from the top of the beam to the centroid of the top longitudinal reinforcing. The bottom cover is measured from the bottom of the beam to the centroid of the bottom longitudinal reinforcing.

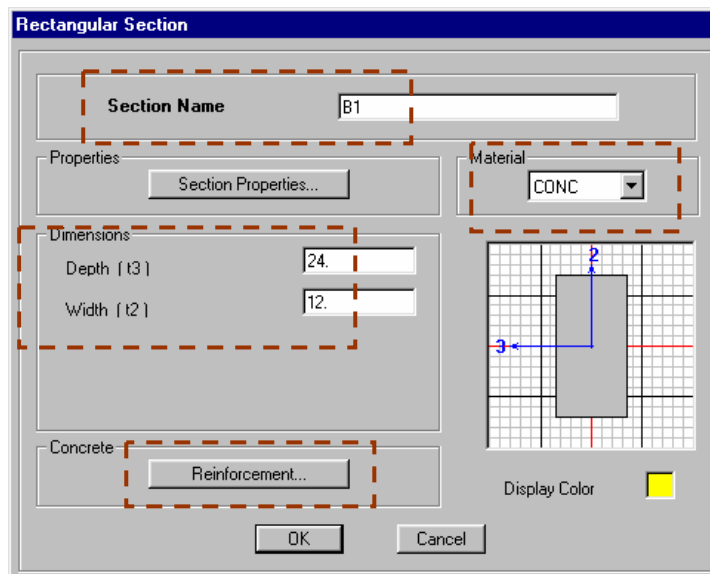
The reinforcement overrides are specified areas of longitudinal reinforcing steel that occur at the top and bottom of the left and right ends of the beam. These overrides are used by ETABS as follows:

In the Concrete Frame Design postprocessor when the design shear in a concrete beam is to be based on provided longitudinal reinforcement (that is, the shear design is based on the moment capacity of the beam) ETABS compares the calculated required reinforcement with that specified in the reinforcement overrides and uses the larger value to determine the moment capacity on which the shear design is based.

In the Concrete Frame Design postprocessor when the minimum reinforcing in the middle of a beam is to be based on some percentage of the reinforcing at the ends of the beam ETABS compares the calculated required reinforcement at the ends of the beam with that specified in the reinforcement overrides and uses the larger value to determine the minimum reinforcing in the middle of the beam.

In the Concrete Frame Design postprocessor when the shear design of columns is to be based on the maximum moment that the beams can deliver to the columns ETABS compares the calculated required reinforcement with that specified in the reinforcement overrides and uses the larger value to determine the moment capacity of the beam.

For any degree of freedom in the frame nonlinear hinge properties assigned to a concrete member that is specified as default ETABS calculates the hinge force-deformation properties based on the larger of the calculated required reinforcement at the ends of the beam (assuming you have run the design through the Concrete Frame Design postprocessor) and the specified reinforcement overrides.

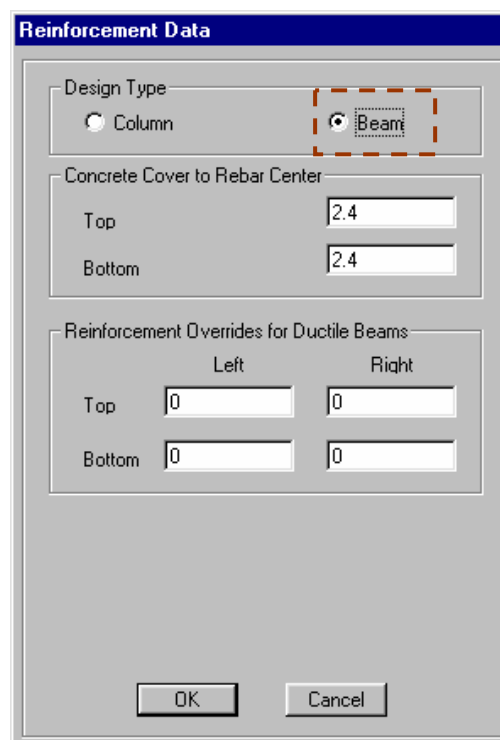


The "Rectangular Section" dialog box is shown with the following fields and controls:

- Section Name:** B1
- Properties:** Section Properties...
- Material:** CONC
- Dimensions:**
  - Depth (t3): 24.
  - Width (t2): 12.
- Concrete:** Reinforcement...
- Display Color:** Yellow
- OK** and **Cancel** buttons.

A grid diagram on the right shows a rectangular section with dimensions 12 (width) and 24 (depth) indicated by blue arrows.

**Step 3-2:** Enter "B1" as Section Name as shown in the above figure and accept the other data. Click on **Reinforcement** to specify reinforcement data.

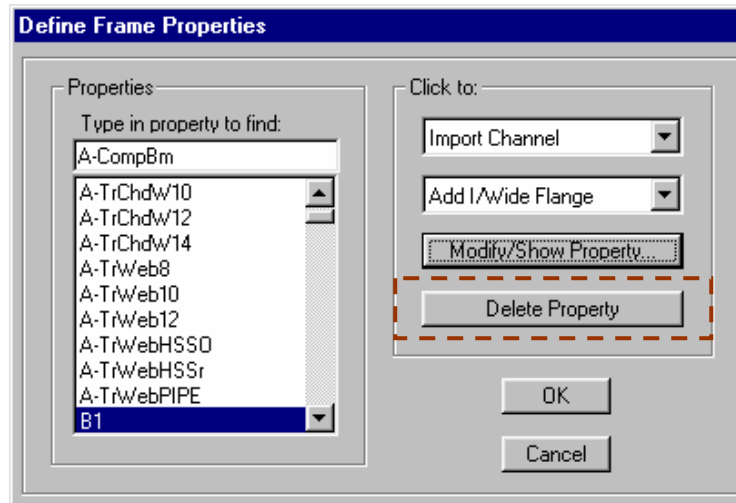


The "Reinforcement Data" dialog box is shown with the following fields and controls:

- Design Type:** ☐ Column, ☒ Beam
- Concrete Cover to Rebar Center:**
  - Top: 2.4
  - Bottom: 2.4
- Reinforcement Overrides for Ductile Beams:**

	Left	Right
Top	0	0
Bottom	0	0
- OK** and **Cancel** buttons.

**Step 3-3:** Select **Design Type** = "Beam" and click **OK** 2 times to go back to "Define Frame Properties" window.



**Step 3-4:** The “Define Frame Properties” window shows “B1” section added in the list. Select “Add Circle” from second drop-down menu to add column section.

## Reinforcing Information for Columns

For columns the following areas are provided in the Reinforcement Data dialog box:

**Configuration of Reinforcement:** Here you can specify rectangular or circular reinforcement. You can if desired put circular reinforcement in a rectangular beam or put rectangular reinforcement in a circular beam.

**Lateral Reinforcement:** If you have specified a rectangular configuration of reinforcement then the only choice available to you here is ties. If you have specified a circular configuration of reinforcement then you have an option of either ties or spiral for the lateral (transverse) reinforcement.

**Rectangular Reinforcement:** This area is visible if you have chosen a rectangular configuration of reinforcement. The following options are available in this area.

**Cover to Rebar Center:** This is the distance from the edge of the column to the center of a longitudinal bar. In the special case of rectangular reinforcement in a circular column the cover is taken to be the minimum distance from the edge of the column to the center of a corner bar of the rectangular reinforcement pattern.

**Number of bars in 3-dir:** This is the number of longitudinal reinforcing bars (including corner rebar) on the two faces of the column that are parallel to the local 3-axis of the section.

**Number of bars in 2-dir:** This is the number of longitudinal reinforcing bars (including corner rebar) on the two faces of the column that are parallel to the local 2-axis of the section.

**Bar size:** This is the specified size of reinforcing steel for the section. You can only specify one bar size for a given concrete frame section property.

**Circular Reinforcement:** This area is visible if you have chosen a circular configuration of reinforcement. The following options are available in this area.

**Cover to Rebar Center:** This is the distance from the edge of the column to the center of a longitudinal bar. In the special case of circular reinforcement in a rectangular column the cover is taken to be the minimum distance from the edge of

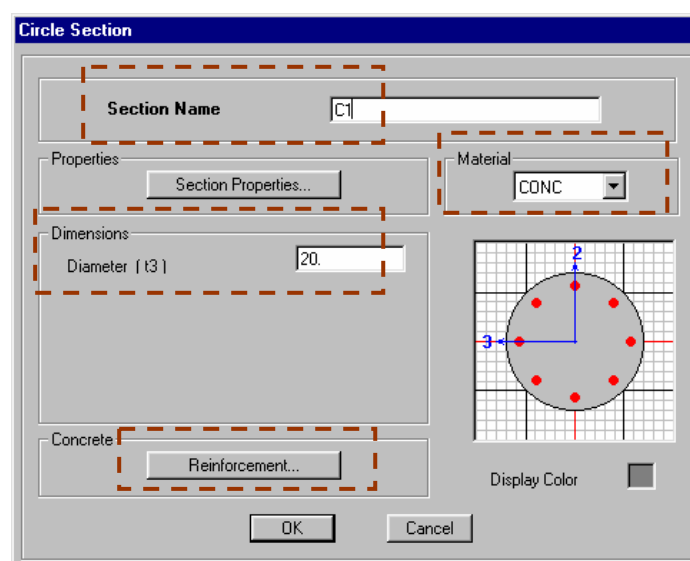
the column to a circle drawn through the center of all the rebar in the circular reinforcement pattern.

**Number of bars:** This is the number of longitudinal reinforcing bars in the section.

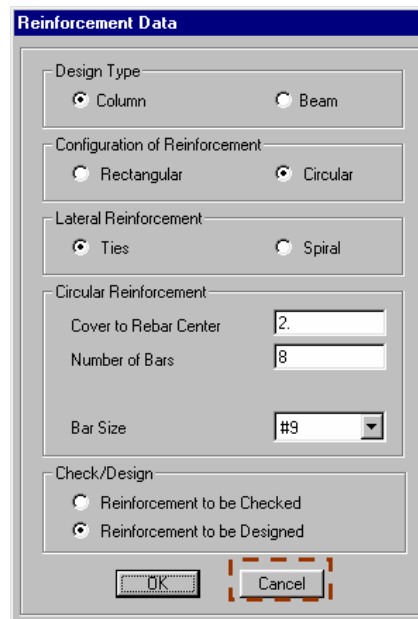
**Bar size:** This is the specified size of reinforcing steel for the section. You can only specify one bar size for a given concrete frame section property.

**Check/Design:** In this area you specify that when a member with this frame section property is run through the Concrete Frame Design postprocessor the reinforcement is either to be checked or to be designed. If the reinforcement is to be checked then all information in the Reinforcement Data dialog box is used. If the reinforcement is to be *designed* then all information in the Reinforcement Data dialog box is used except the bar size is ignored and the total required steel area is calculated. For design the configuration of reinforcement, lateral reinforcement and cover is used.

If you specify reinforcing in a concrete column frame section property that is specified using the section designer utility then the Concrete Frame Design postprocessor either checks the column for the specified reinforcing or designs new reinforcing depending on the option you selected when you specified the section.



**Step 3-5:** Enter “C1” section properties as shown in above figure, accept other data and click on **Reinforcement** button to specify reinforcement data.



**Reinforcement Data**

Design Type  
☒ Column ☐ Beam

Configuration of Reinforcement  
☐ Rectangular ☒ Circular

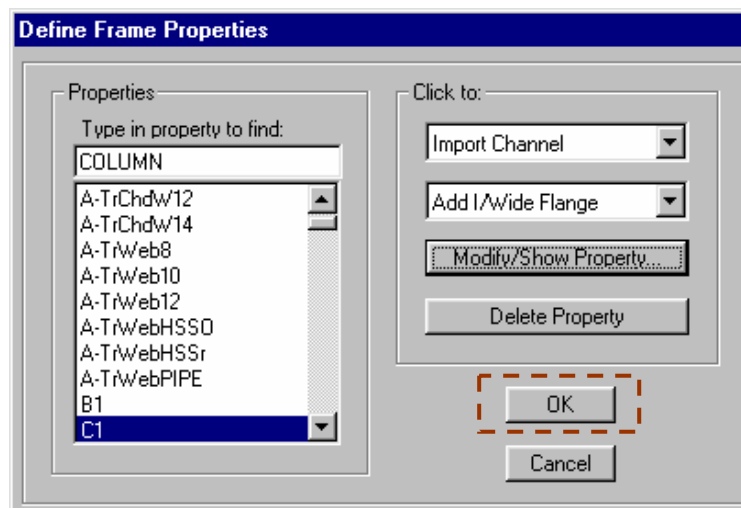
Lateral Reinforcement  
☒ Ties ☐ Spiral

Circular Reinforcement  
 Cover to Rebar Center: 2  
 Number of Bars: 8  
 Bar Size: #9

Check/Design  
☐ Reinforcement to be Checked  
☒ Reinforcement to be Designed

OK Cancel

**Step 3-6:** In **Design Type** area select **Design Type** = “Column”. Clicking **OK** 2 times to finish this step.



**Define Frame Properties**

Properties  
 Type in property to find:  
 COLUMN

A-TrChdw12  
 A-TrChdw14  
 A-TrWeb8  
 A-TrWeb10  
 A-TrWeb12  
 A-TrWebHSS0  
 A-TrWebHSSr  
 A-TrWebPIPE  
 B1  
 C1

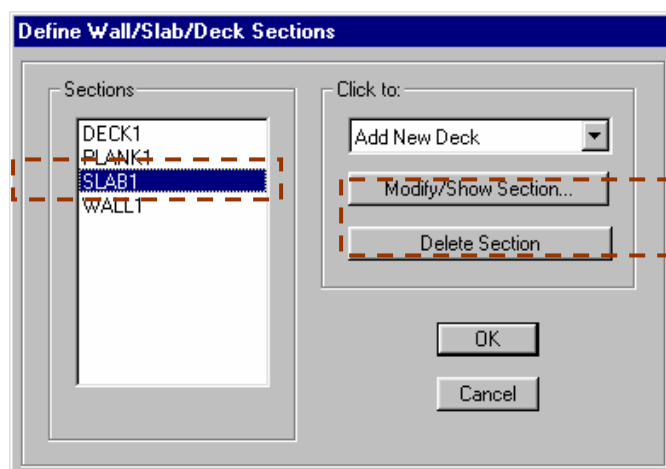
Click to:  
 Import Channel  
 Add I/Wide Flange  
 Modify/Show Property  
 Delete Property


OK Cancel

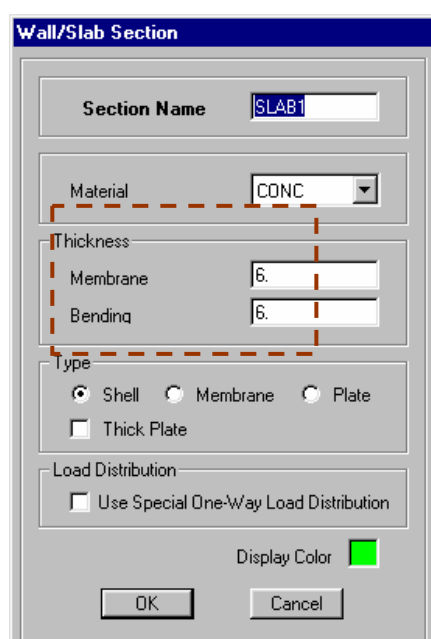
**Step 3-7:** **Define Frame Properties** dialogue shows “C1” section in list. Click **OK** to go back to main screen, which completes the Frame Section Definition steps.

## 4. Define Slab Sections

Default slab section ("SLAB1") will be modified by changing only the thickness value.



**Step 4-1:** Click on  in tool bar or **Define > Wall/Slab/Deck Sections** menu command. Select "SLAB1" from list and click on **Modify/Show Section**.



**Step 4-2:** Specify **Thickness** = "6" in **Membrane** and **Bending** and click **OK** 2 times to finish Define Slab Section.

**Thickness:** Two thicknesses are specified: membrane and bending. Typically these thicknesses are the same but they can be different. For instance they may be different if you are trying to model full shell behavior for a corrugated metal deck.

The membrane thickness is used for calculating:

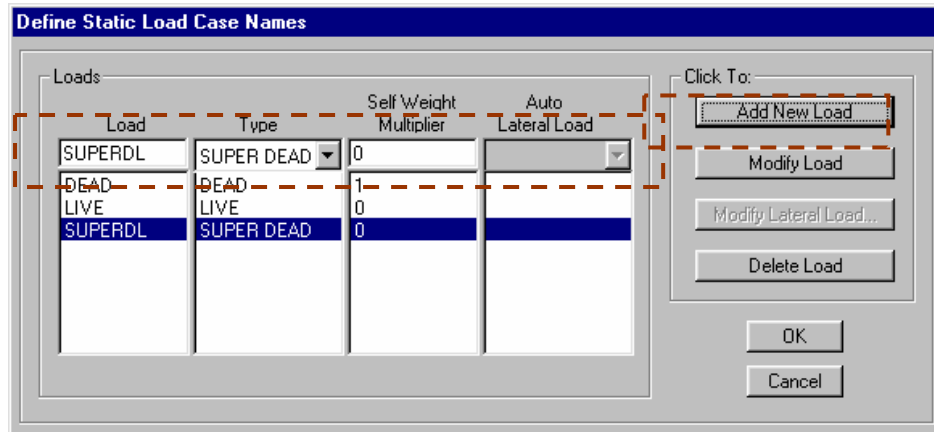
- ✓ The membrane stiffness for full shell and pure membrane sections.
- ✓ The element volume for element self-mass and self-weight calculations.

The bending thickness is used for calculating the plate-bending and transverse-shearing stiffnesses for full shell and pure plate sections.



## 5. Define Load Cases

2 basic load cases "DEAD", "LIVE" will be defined first and one more load cases ("SUPERDL") will be defined for superimposed dead load on perimeter beams.

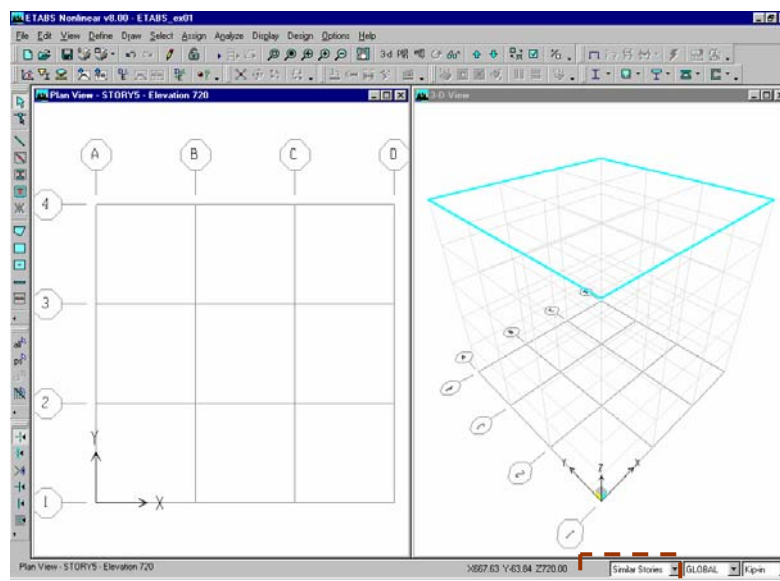


**Step 5-1:** Click on  in tool bar or select **Define > Load Cases** menu. 2 basic load cases are already added to the list.

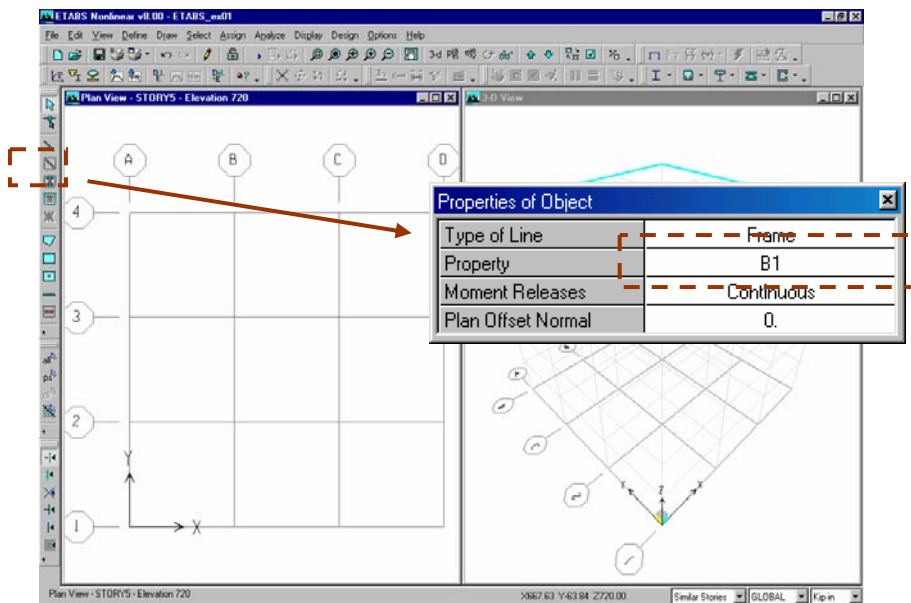
Add "SUPERDL" load case by entering "Load" = "SUPERDL", "Type" = "SUPER DEAD" and "Self Weight Multiplier" = "0". Click **Add New Load** and click **OK** to finish this step.


## 6. Draw Beam Objects (Frame Members)

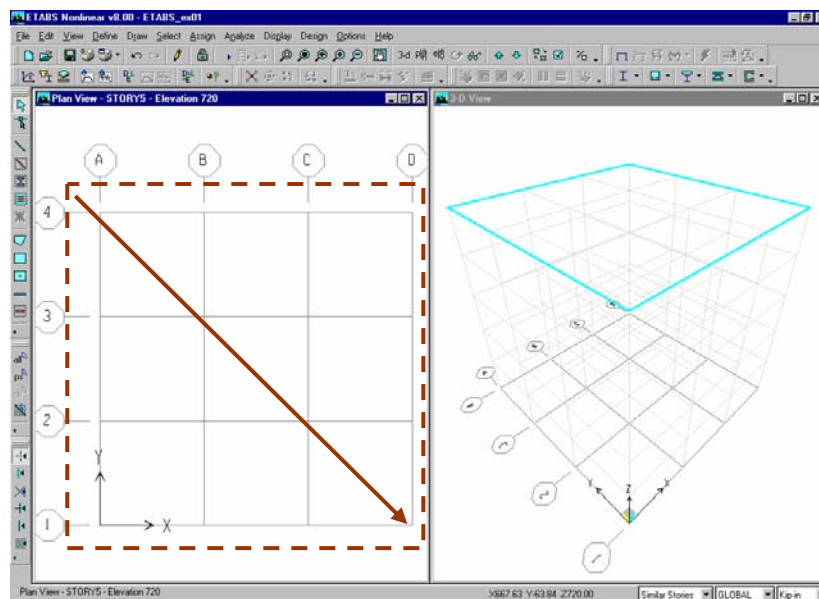
Select "Similar Stories" from drop-down menu in bottom-right of screen to apply all assignments in one floor (current floor) in 'Plan View' to all similar stories. Similar floor can be defined from **Edit > Edit Story Data > Edit Story** menu



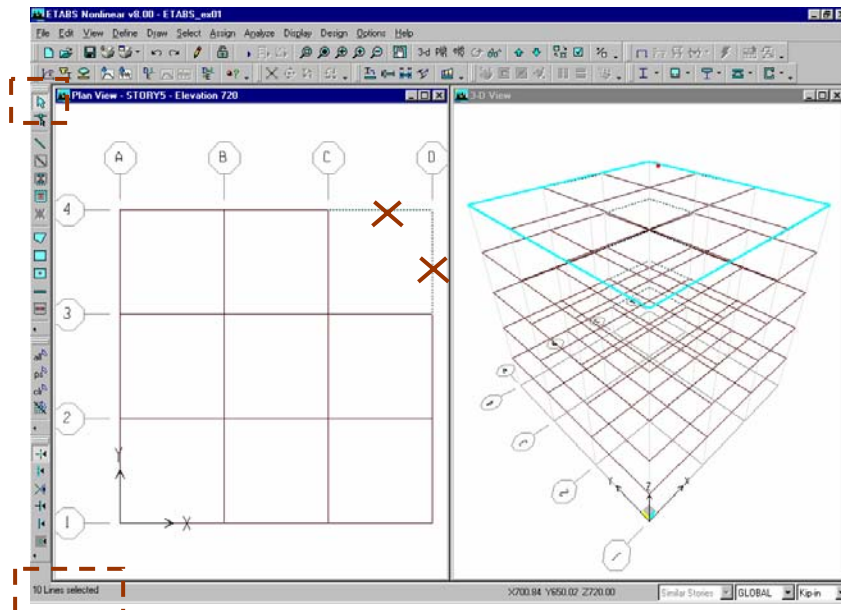
**Step 6-1:** Back to main screen, activate "Plan View" window by clicking anywhere in left window. Change "One Story" to "Similar Stories" from drop-down menu in bottom-right screen (to edit multiple stories simultaneously).





**Step 6-2:** Click on  in tool bar. “The properties of Object” window will appear. Select **Property** = “B1”.



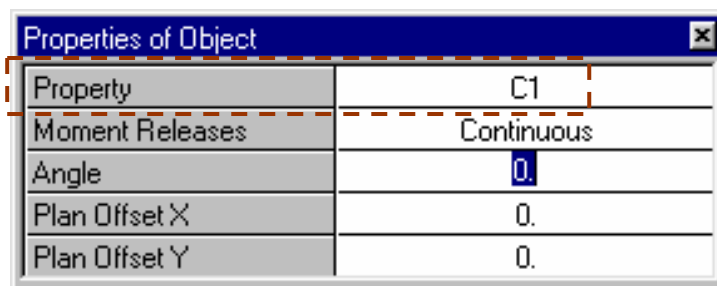
**Step 6-3:** To add “B1” in all spans in model, draw rectangular selection to cover all plan in Plan View. “B1” will be added in every span of the entire model.




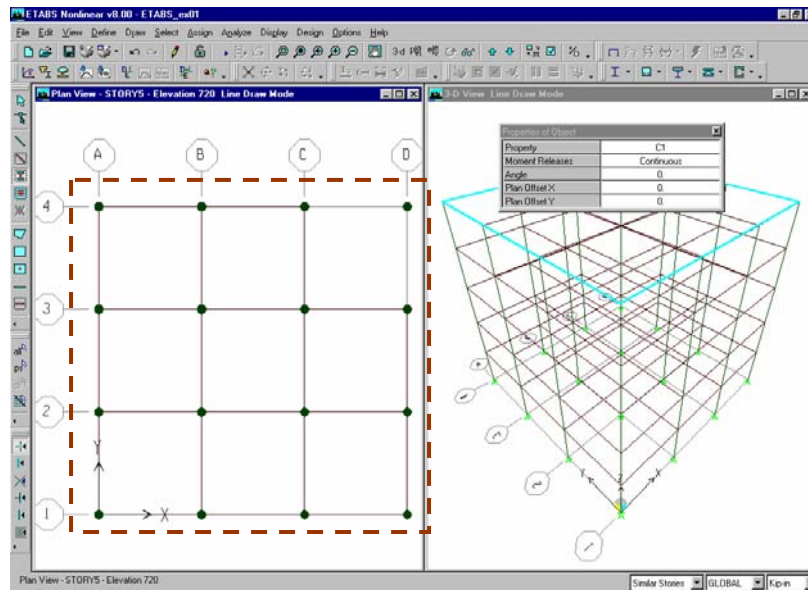
**Step 6-4:** Change back to selection mode by clicking on  in tool bar. To delete all beams in top-right corner, click on these 2 bays, the selection status in bottom-left of main screen reports "10 Lines selected" (2 bays x 5 floors). If this report is different, press  and repeat this step again. Press **Delete** key on keyboard.

## 7. Draw Column Objects (Frame Members)

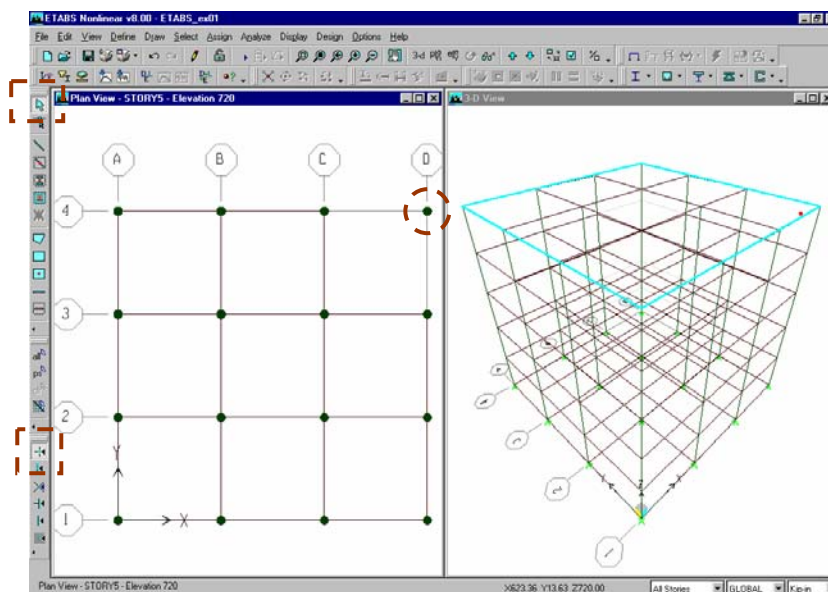
The procedure for adding column is similar to adding beams as described in previous section. Make sure that "Similar Stories" is selected from drop-down menu in bottom-right of main screen.





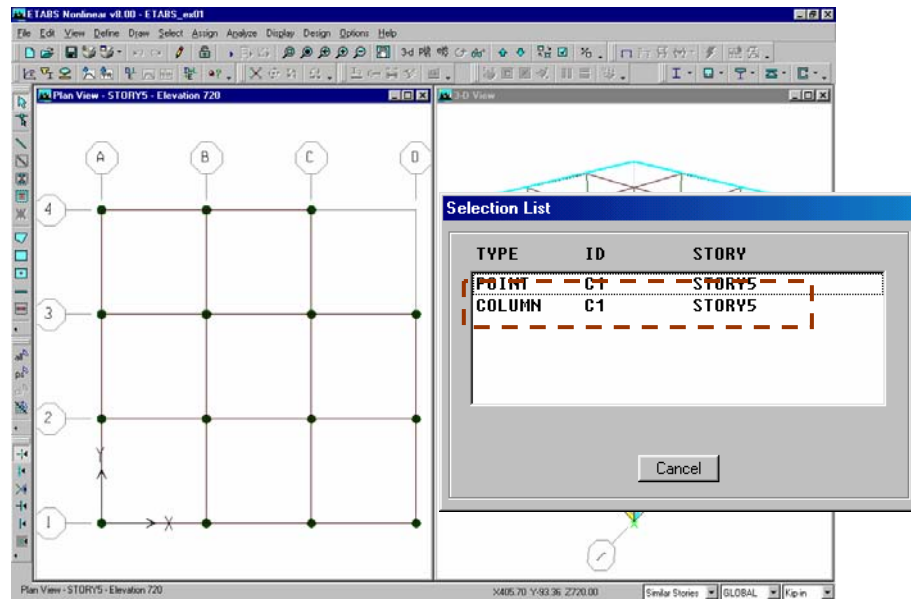
**Step 7-1:** Click on  in left tool bar. Select **Property** = "C1" from "Properties of Object" window.



**Step 7-2:** To add "C1" in all column locations in the model, draw rectangular selection to cover all plan in the Plan View. Note the change in 3D view.



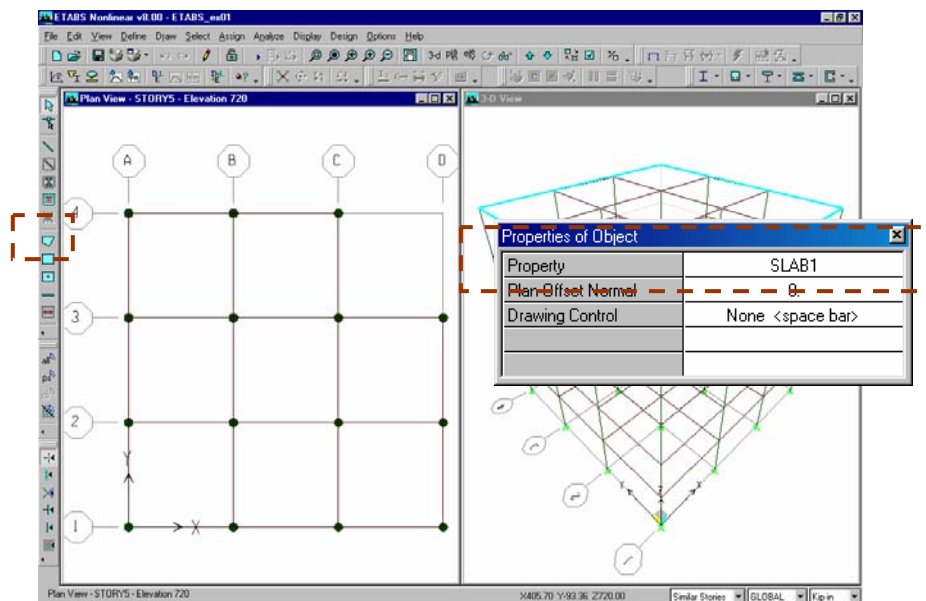
**Step 7-3:** Change back to selection mode by clicking on  in left tool bar. Make sure that "Snap to Grid Intersections and Points" is active . Click on intersection point between grid line D and 4 while holding **Ctrl** on the keyboard.




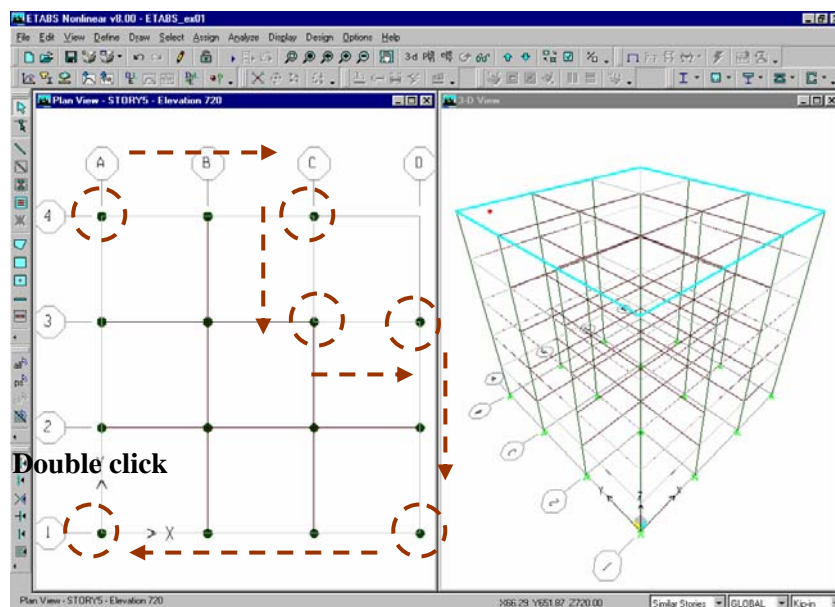
**Step 7-4:** “Selection List” window will open showing that points and a columns object are in the same location. Click on “COLUMN” to select only the columns. “5 Lines Selected” (1 column line x 5 floors) message is displayed on the status bar at bottom-left of screen. Press **Delete** key on keyboard or go to select **Edit > Delete** menu to delete the selected columns.


## 8. Assign Slab Sections

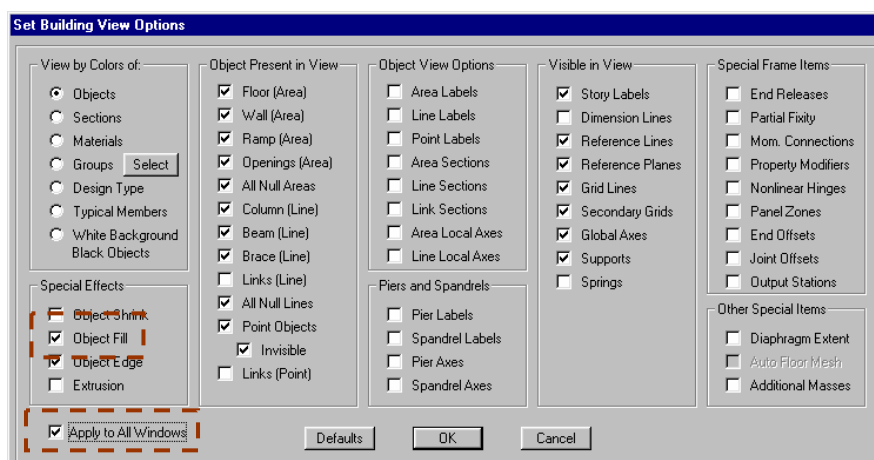
Slab section will be assigned to all floors in “Plan View” window in one step by taking advantage of “Similar Stories” feature.




**Step 8-1:** Click on  in tool bar or select **Draw > Draw Areas Objects > Draw Areas** menu. Select **Property** = “SLAB1” from Properties of Object window.

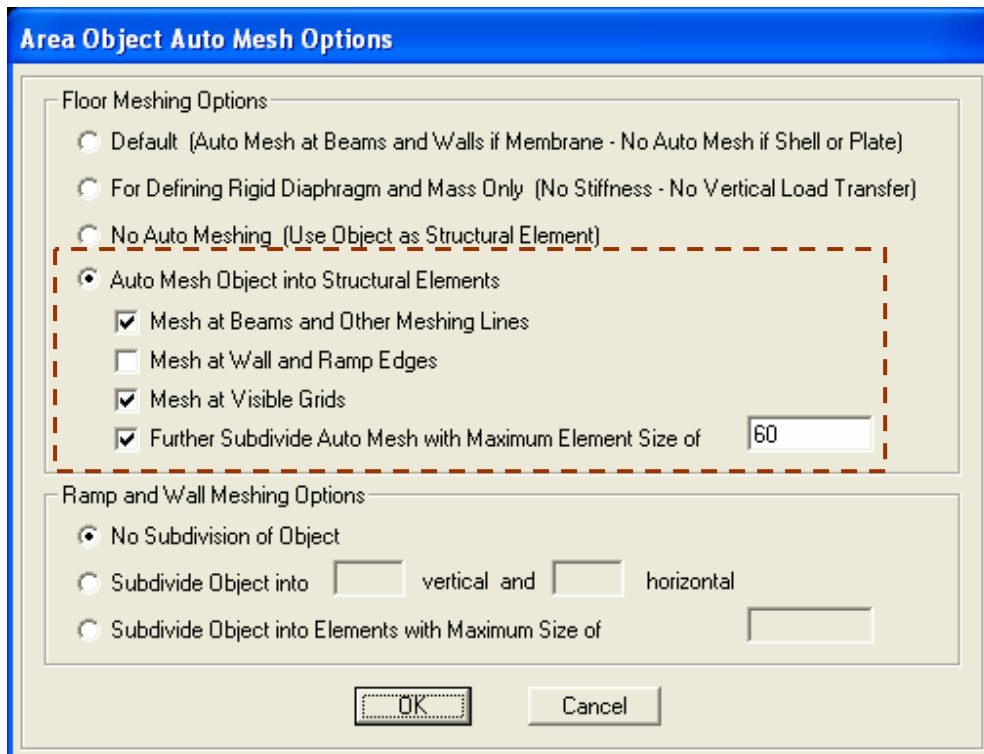


**Step 8-2:** Make sure that “Snap to Grid Intersections and Points” is active . To draw Slab Object, one click a series of column locations as shown above and double click on the column to finish the operation.




**Step 8-3:** Slab sections are applied in all floors. To view slab more clearly, click on  or go to **View > Set Building View Options**, select “Object Fill” to fill area objects and select “Apply to All Windows” to applied selected options to all windows. Click **OK**.

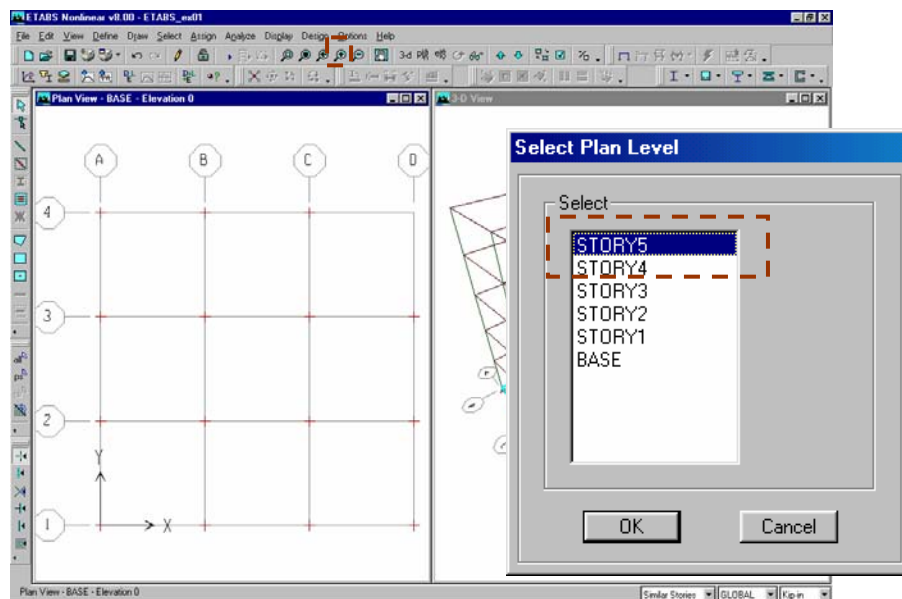





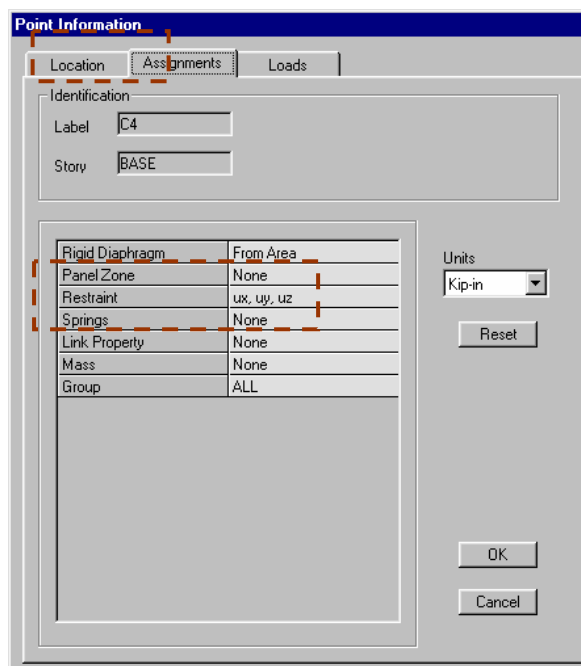
**Step 8-4:** Click on , select all Slabs by clicking on them go to **Assign >> Shell/Area >> Area Object Mesh Options** and specify parameter as shown in above figure

## 9. Assign Restrains

All nodes at Basement Floor will be assigned to be Hinge automatically. To change support restraint properties, select support nodes, click on  or go to **Assign > Joint/Point > Restraint** menu and select support type.



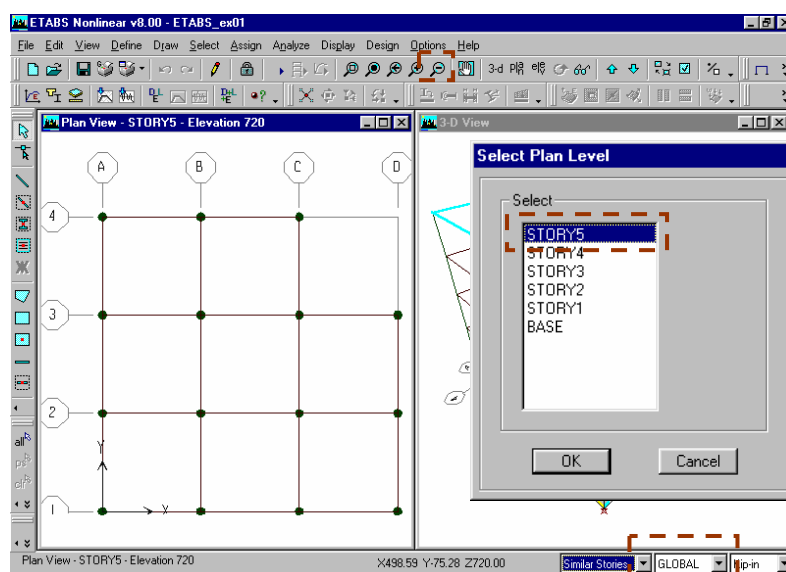
**Step 9-1:** Change Elevation View to Plan View by clicking  and selecting "BASE" from list. Click on tight button of the mouse at any joints on this Plan View to open "Point Information" window.



**Step 9-2:** Click on "Assignments" tab. In "Restraint" shows "ux, uy, uz" that means all deformation in x, y and z direction are fixed. Click **Cancel**.

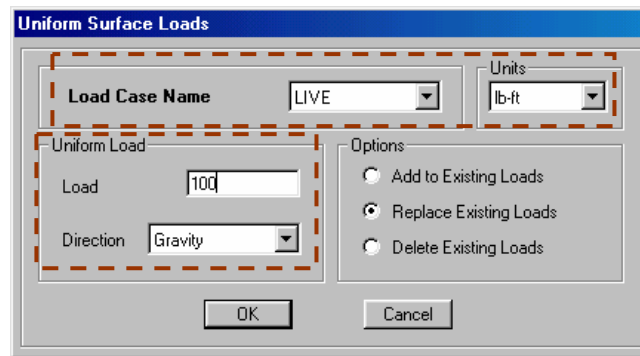
## 10. Assign Slab Loads


Self weight for slabs and beams are computed automatically which can be specified while defining load cases and setting "Self Weight Multiplier" = "1" for "DEAD" load case" (see Step 5-1). Live load (100 psf) will be assigned in "LIVE" load case. Wall load along perimeter beams (250 plf) and super-imposed dead load (35 psf) in slabs will be assigned to "SUPERDL" load case. All steps will be done in Plan View keeping "Similar Stories" selected.

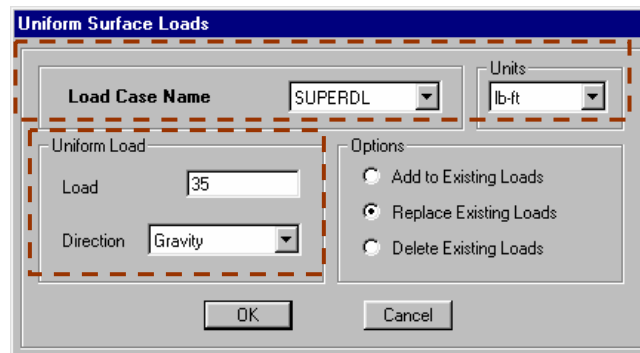


**Step 10-1:** Change "Elevation View" to "Plan View" by clicking **Pla** and selecting "STORY5" from list. Select "Similar Stories" in drop-down menu in bottom-right of screen.

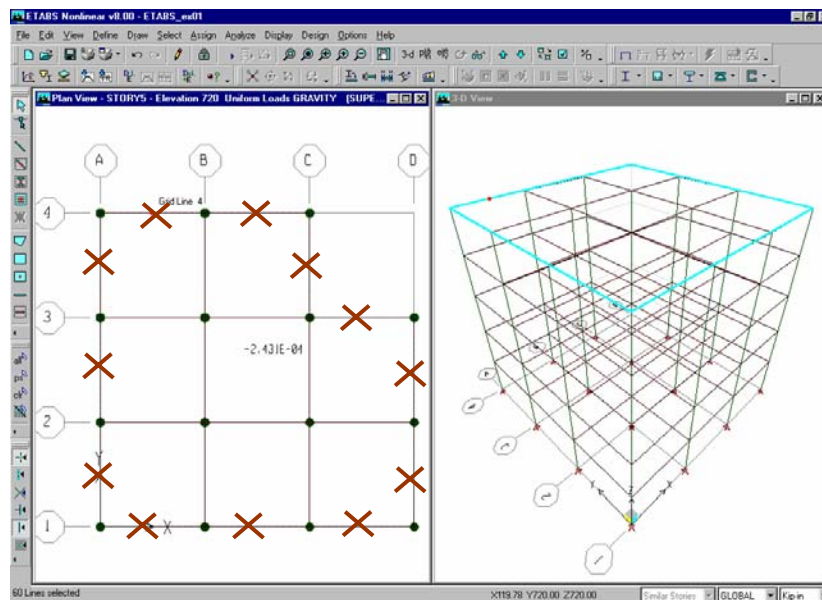





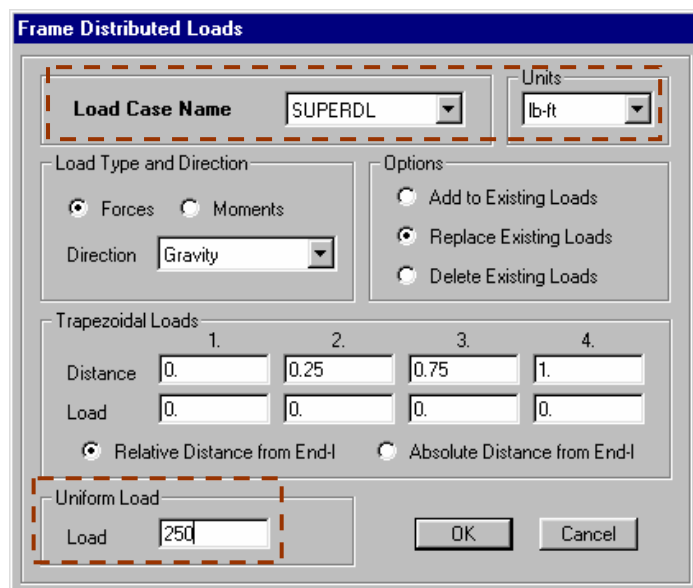
**Step 10-2:** Click any area in slab from Plan View. The status bar shows "5 Areas, 30 Edges selected" (1 areas x 5 floors). Click on  or select **Assign > Shell/Area Loads > Uniform** menu. In "Uniform Surface Loads" window, select **Load Case Name** = "LIVE", select **Unit** = "lb-ft", select **Direction** = "Gravity", enter **Load** = "100" (positive for downward in "Gravity" direction) and click **OK**.



**Step 10-3:** Repeat Step 10-1 and Step 10-2 to assign "35 psf" to "SUPERDL" load case in all slabs.



**Step 10-4:** Click on  or go to **Draw > Snap to > Lines and Edges**. Move mouse over perimeter beam wait until red dot appears on that beam then right mouse click. Repeat this step to select perimeter beam one by one until bottom-left screen displays "60 Lines selected" (12 bays x 5 floors).



**Frame Distributed Loads**

Load Case Name: SUPERDL Units: lb-ft

Load Type and Direction: ☒ Forces ☐ Moments Direction: Gravity

Options: ☐ Add to Existing Loads ☒ Replace Existing Loads ☐ Delete Existing Loads

Trapezoidal Loads:

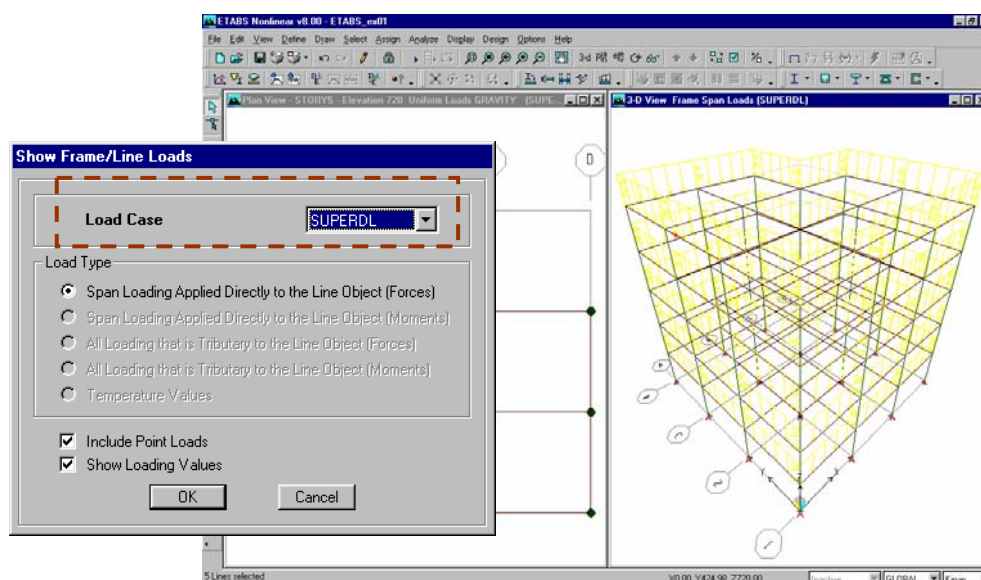
	1.	2.	3.	4.
Distance	0.	0.25	0.75	1.
Load	0.	0.	0.	0.


☒ Relative Distance from End-I ☐ Absolute Distance from End-I

Uniform Load: Load: 250

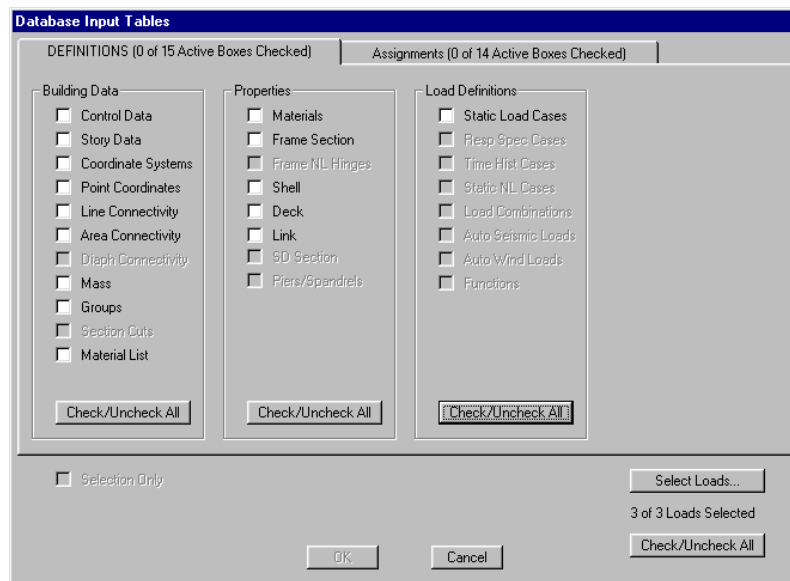
OK Cancel

**Step 10-5:** Click on  or select **Assign > Frame/Line Loads > Distributed**. Menu to specify **Load Case Name** = "SUPERDL", **Units** = "lb-ft" and **Uniform Load** = "250" and click **OK**.

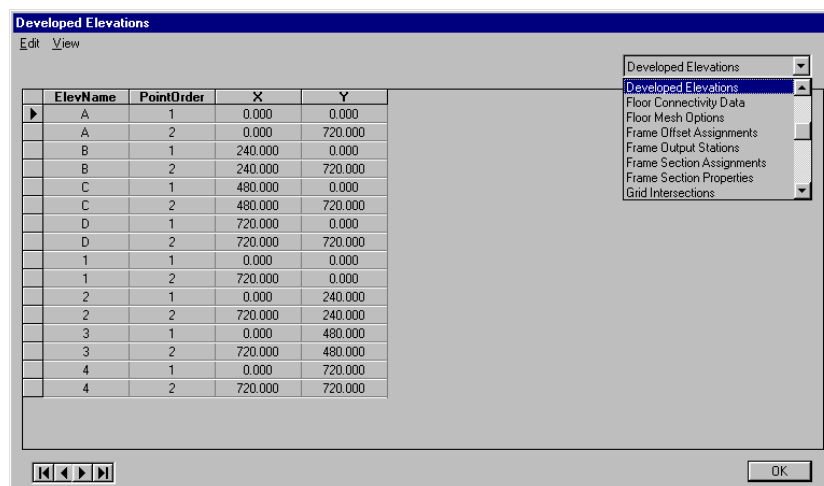


**Step 10-6:** To display distributed load that just assigned in perimeter beams from previous step. Activate "3D View" window, select **Display > Show Loads > Frame/Line** menu, select Load case = "SUPERDL" and click OK. Click on  to change back to normal display mode.

## 11. View Input Data in Tabular Form

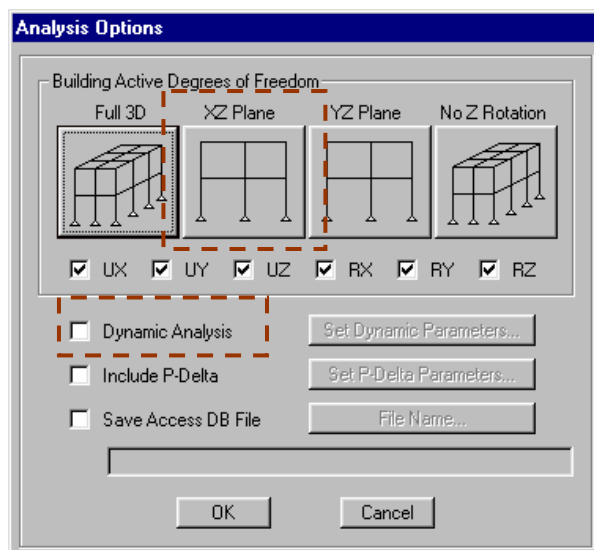


**Step 11-1:** Go to **Display > Set Input Table Mode**, select items to be displayed and click **OK**.

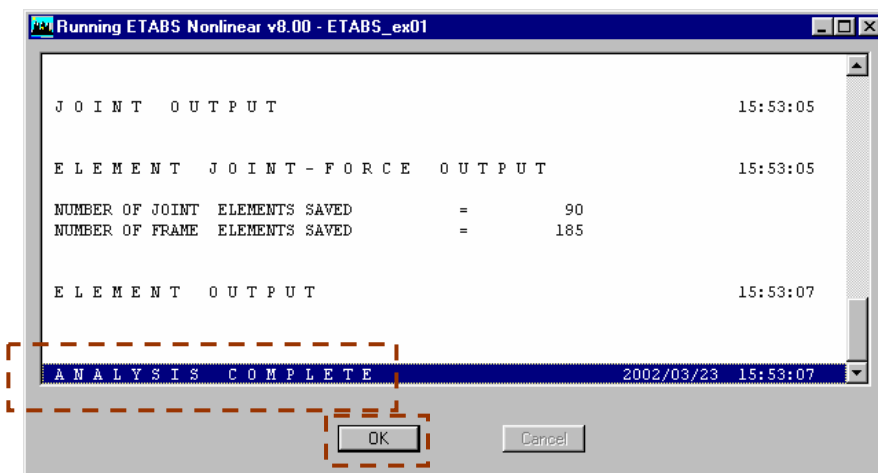



**Step 11-2:** To view each input data, select items from drop-down menu. Click **OK** to finish this step.

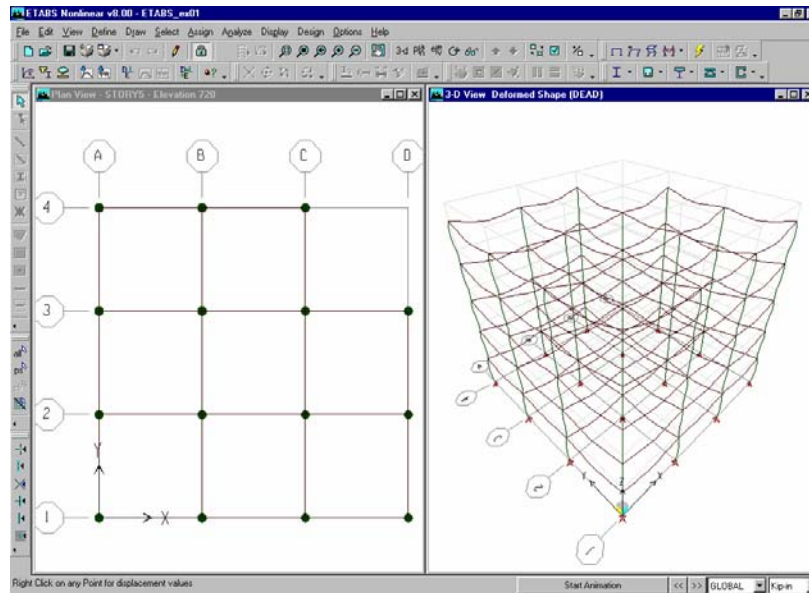
## 12. Run the Analysis



**Step 12-1:** Go to **Analysis > Set Analysis Options**, select "Full 3D", deselect "Dynamic Analysis" (not required for this part of example) and click **OK**.

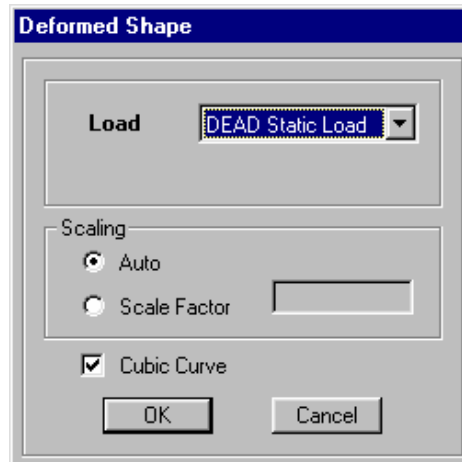


**Step 12-2:** Click on  or go to **Analysis > Run Analysis**, click on **Run** from Analysis Options "Window" and wait until ETABS displays "ANALYSIS COMPLETE" and click **OK**.

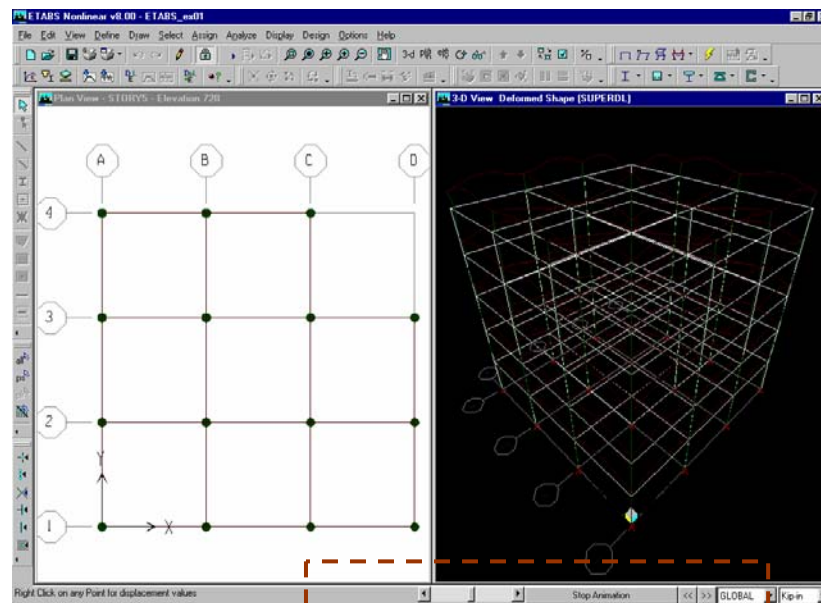



**Step 12-3:** When Analysis process has been completed, ETABS will display deformed shape of the model in active window and model is locked automatically (🔒 is pressed). When the model is locked, model cannot be modified unless unlock the model by depressing 🔓 button. After model is unlocked, all output data will be deleted and reanalysis is required to obtain output data again.

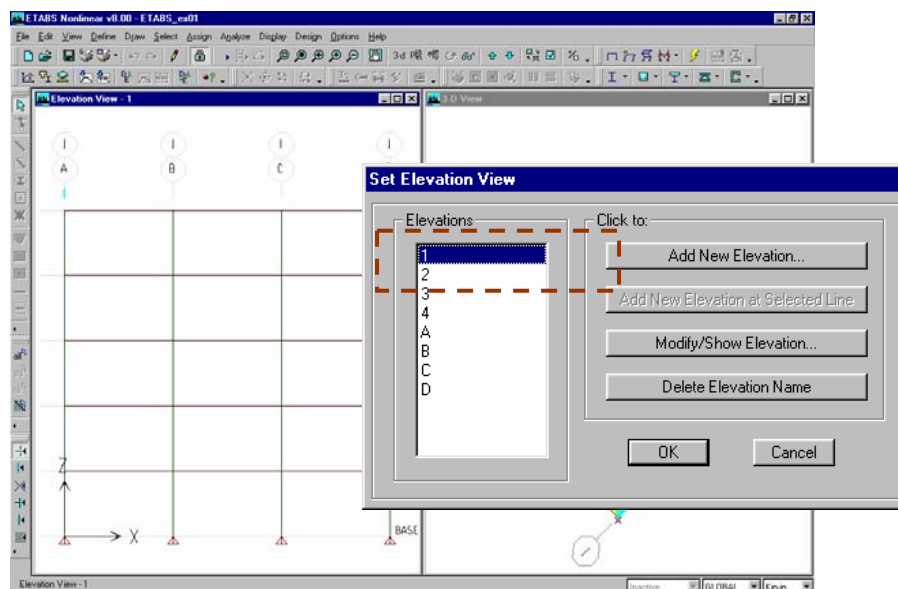
### 13. View Analysis Results Graphically




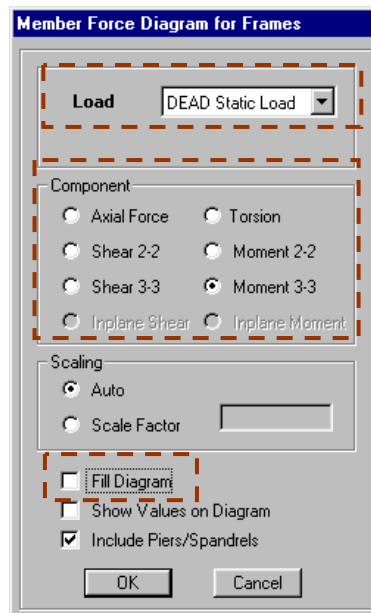
**Step 13-1:** Deformed shape can be displayed by clicking on 📐 and select load case from drop-down menu.




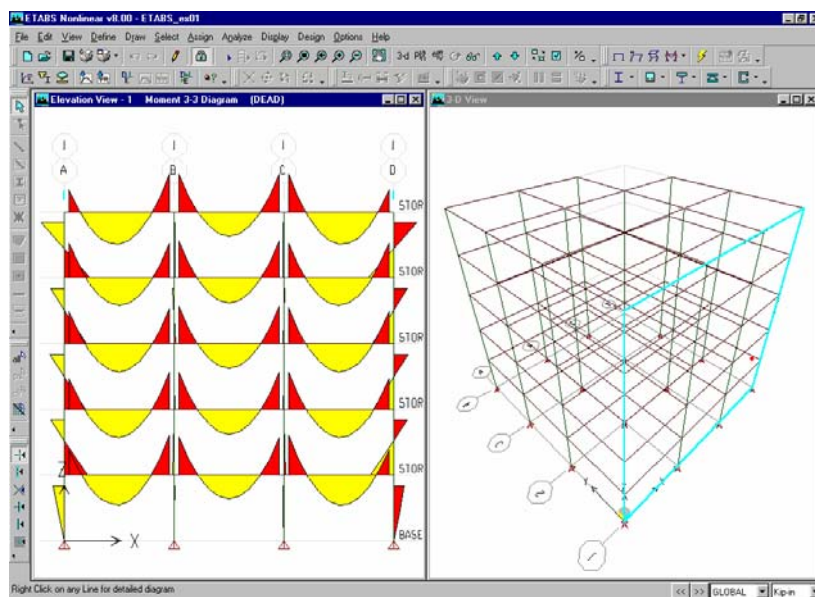
**Step 13-2:** Deformed shape can be displayed in animation by clicking on **Start Animation**. Speed can be adjusted by scroll bar at the bottom of this window. Click on **Stop Animation** and click on  to change back to normal display mode.



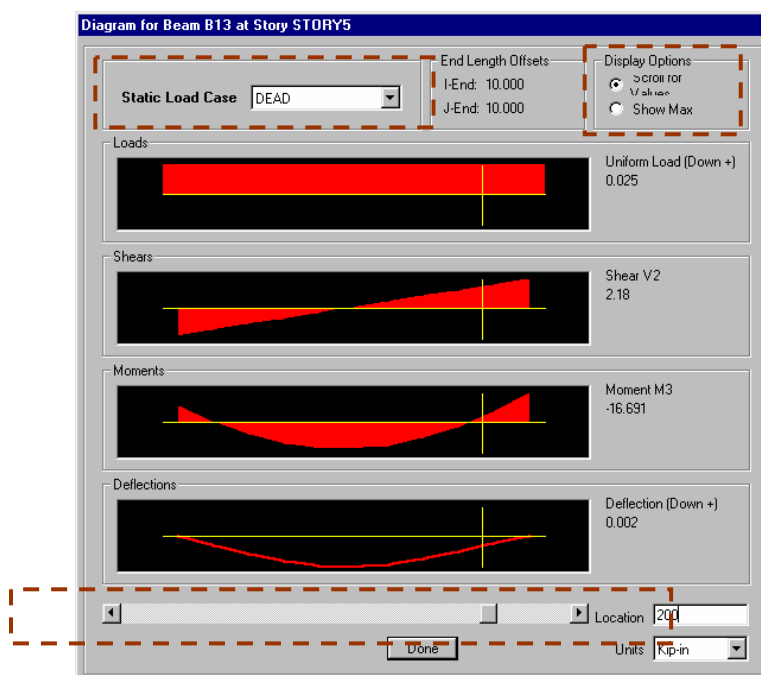
**Step 13-3:** Active "Plan View" window and change to "Elevation View" by clicking on  and selecting **Elevation = "1"**.




**Step 13-4:** Analysis results can be displayed by clicking on . Select load cases from drop-down menu, select "Component" (shear, moment or torsion) and select "Fill Diagram".



**Step 13-5:** Now moment diagrams (3-3) are displayed with positive moment on the tension side. To display tension in negative side, go to **Options** and deselect **Moment Diagrams on Tension Side**. Right click on the beam between grid line A and B at roof floor to display analysis result details.



**Step 13-6:** Loads, shears, moments and deflections for the right clicked beam are displayed. Select **Display Options** = "Show Max" to see maximum and location for each result. Select **Display Options** = "Scroll for Values" result values at particular location can be shown by moving scroll bar or entering location in text box. Click **Done** to close this window and click on  to change back to normal display mode..

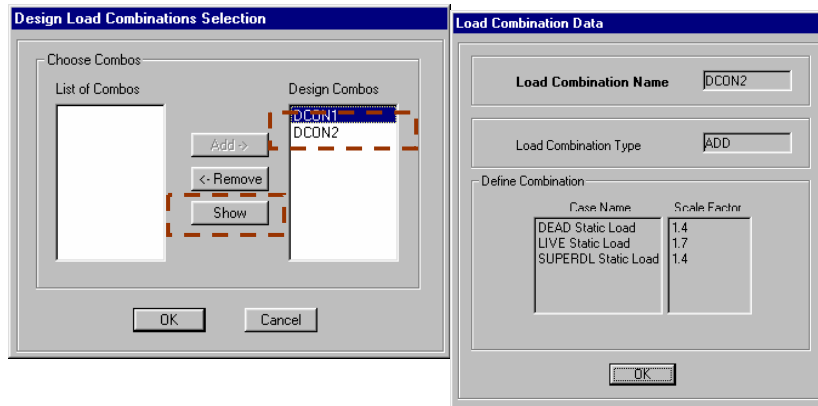
## 14. Design Concrete Frame Elements

There are many design codes available for concrete frame design. ETABS will define load combinations according to specified design code automatically after specifying the design codes. Additional load combinations can be defined and added to the design combo list if required.

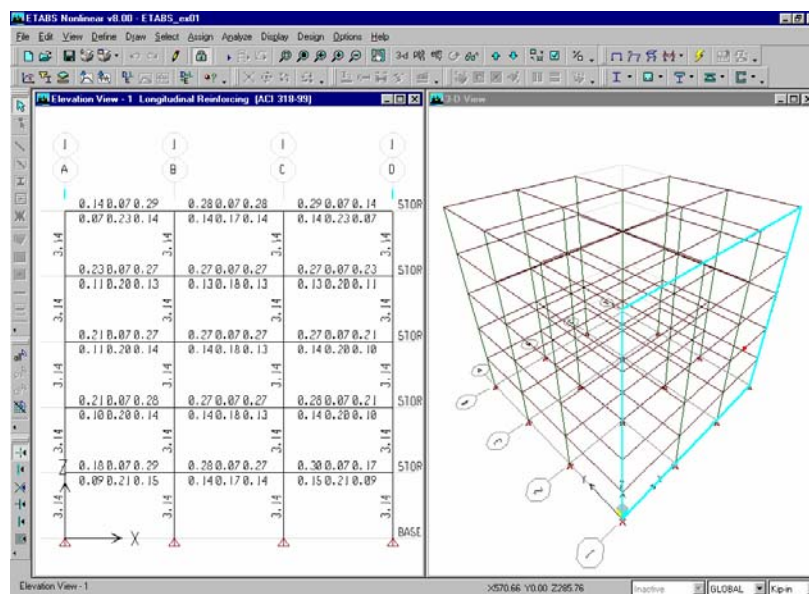
Concrete Frame Design Preferences	
Design Code	ACI 318-99
PhiBendingTension	0.9
PhiCompressionTied	0.7
PhiCompressionSpiral	0.75
PhiShear	0.85
NumberInteractionCurves	24
NumberInteractionPoints	11
Time History Design	Envelopes
EuroNu	0.01
EuroGammaC	1.5
EuroGammaS	1.15
Utilization Factor Limit	0.95

**Step 14-1:** To select design code, go to **Options > Preferences > Concrete Frame Design** menu, select "ACI 318-89" and click **OK**.





**Step 14-2:** To see load combination for design, go to **Design > Concrete Frame Design > Design Combo**. 2 load combinations are defined and selected for design automatically, select "DCON2" and click **Show**. Load factors are defined according to specified design code from previous step. Click **OK** 2 times.



**Step 14-3:** Start concrete frame design by clicking on . When ETABS finishes the design process, required reinforcement in each element will be displayed.

Concrete Beam Design Information				
Story	STORY5		Section Name	
Beam	B13		B1	
COMBO ID	STATION LOC	TOP STEEL	BOTTOM STEEL	SHEAR STEEL
DCON2	98.00	0.071	0.227	0.000
DCON2	120.00	0.071	0.231	0.000
DCON2	142.00	0.071	0.198	0.000
DCON2	164.00	0.071	0.130	0.000
DCON2	186.00	0.071	0.071	0.000
DCON2	208.00	0.111	0.071	0.000
DCON2	230.00	0.287	0.143	0.000
<div> Overwrites Summary Flex. Details Shear Details </div>				
<div> OK Cancel </div>				

**Step 14-4:** To see beam design details, right mouse click on particular beam (example beam between grid line A and B at roof floor). ETABS highlights the critical location along the element length (maximum required reinforcement). More details can be displayed by clicking on button below. Click **OK** to close this dialogue.

**Concrete Column Design Information**

Store:  Section Name:   
 Column:

COMBO ID	STATION LOC	LONGITUDINAL REINFORCEMENT	MAJOR SHEAR REINFORCEMENT	MINOR SHEAR REINFORCEMENT
DCON1	0.00	3.142	0.000	0.000
DCON1	60.00	3.142	0.000	0.000
DCON1	120.00	3.142	0.000	0.000
DCON2	0.00	3.142	0.000	0.000
DCON2	60.00	3.142	0.000	0.000
DCON2	120.00	3.142	0.000	0.000

Overwrites Interaction Summary Flex. Details Shear Details Joint Shear B/C Details

OK Cancel

**Step 14-5:** As beam design details, column design details can be displayed by right mouse clicking on particular column (example column along grid line A between roof and 4 floor). ETABS highlights the critical location along element length (maximum required reinforcement). More design details can be displayed by clicking on button below. Click **OK** to close this dialogue.

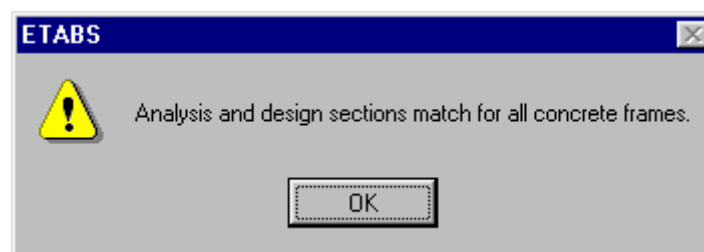
**Display Design Results**

☒ Design Output

☐ Design Input

OK Cancel

**Step 14-6:** To display design input or output in graphic view, go to **Design > Concrete Frame Design > Display Design Results**.

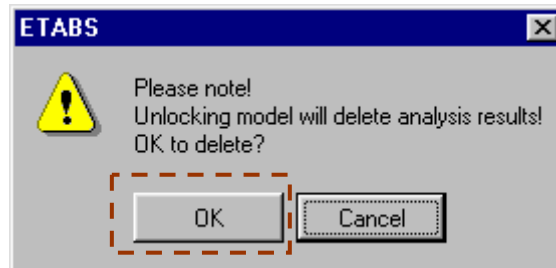



**Step 14-7:** To display the status on the concrete frame members that passed the design, go to **Design > Concrete Frame Design > Verify Analysis VS Design Section**.

## Part B: Dynamic Analysis and Design

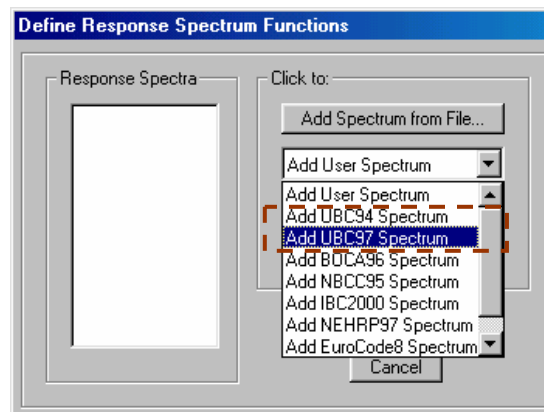
### Step-by-Step Solution


#### 1. Unlock the Model

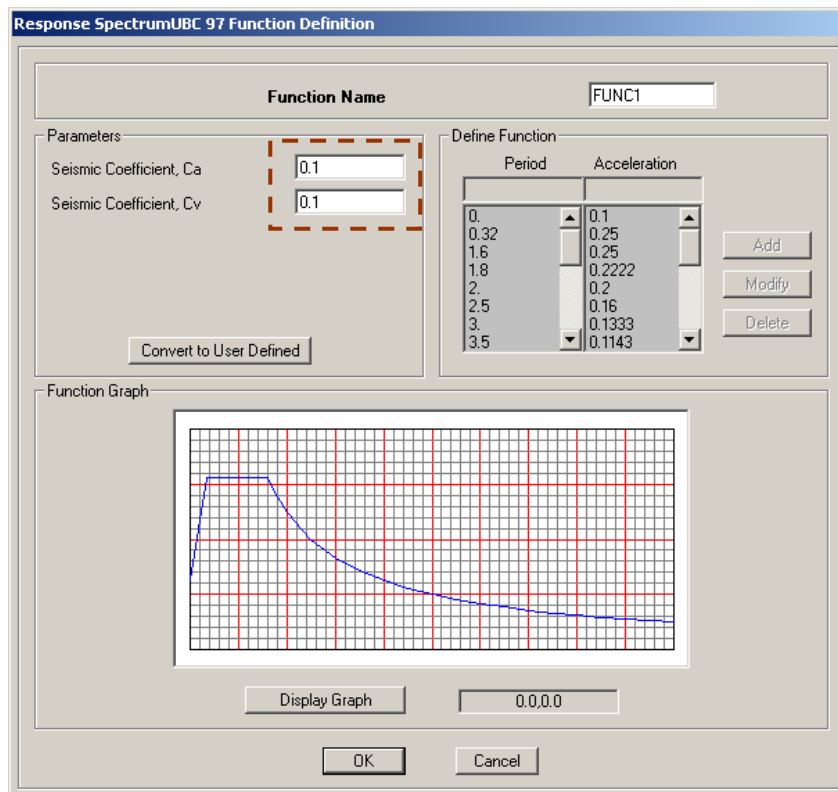


**Step 1-1:** This model is locked automatically after the analysis process is completed. To unlock this model, click on , a message window prompts with a message that 'all analysis results will be deleted if the model is unlocked'.

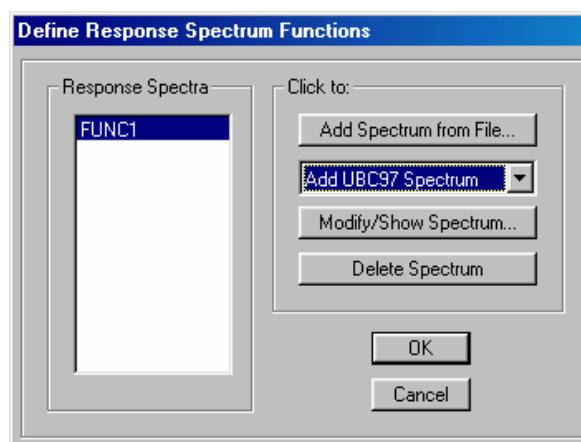
#### 2. Define Response Spectrum Function



**Step 2-1:** Click on  or go to **Define > Response Spectrum Functions**, select "Add UBC97 Spectrum" from drop-down menu.

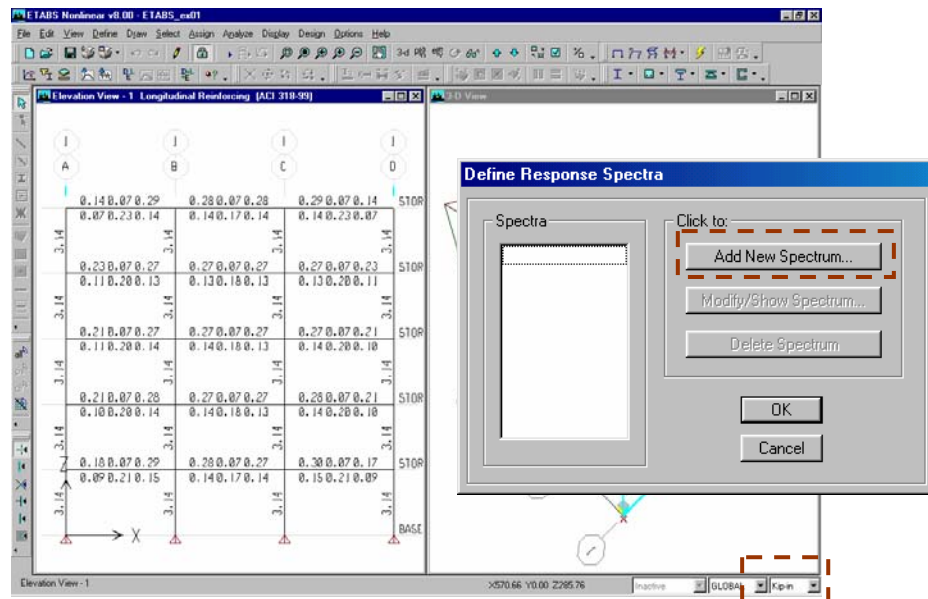


**Step 2-2:** Accept built-in "UBC97" response spectrum by clicking **OK**.



**Step 2-3:** "FUNC1" response spectrum function has been defined already, click **OK** to end this step.

### 3. Define Response Spectrum Cases



**Step 3-1:** Select working unit to be “in”, go to **Define > Response Spectrum Cases** and click on **Add New Spectrum**.

**Response Spectrum Case Data**

**Spectrum Case Name**

**Structural and Function Damping**

Damping

**Modal Combination**

☐ CQC ☒ SRSS ☐ ABS ☐ GMC

f1  f2

**Directional Combination**

☒ SRSS ☐ ABS  ☐ Modified SRSS (Chinese)

**Input Response Spectra**

Direction	Function	Scale Factor
U1	<input type="text"/>	<input type="text"/>
U2	<input type="text" value="FUNC1"/>	<input type="text" value="32"/>
U3	<input type="text"/>	<input type="text"/>

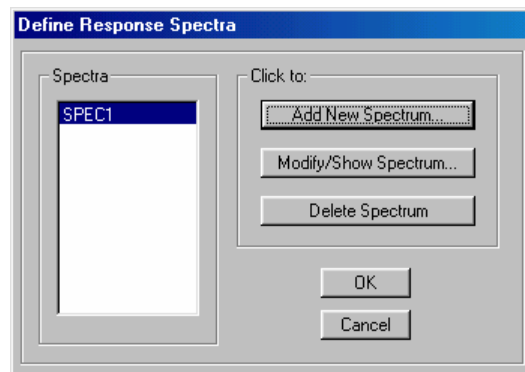
Excitation angle

**Eccentricity**

Eccentricity Ratio

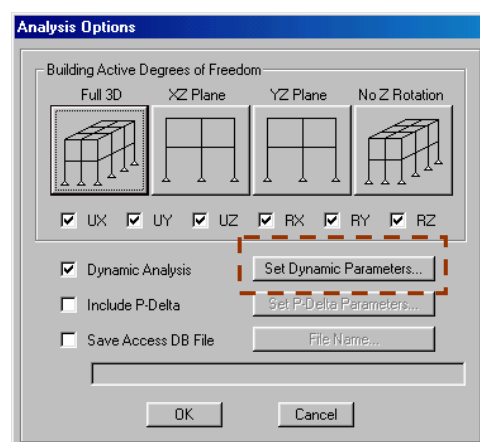
Override Eccentricities

**Step 3-2:** Specify response spectrum case data as shown in above figure and click **OK**.

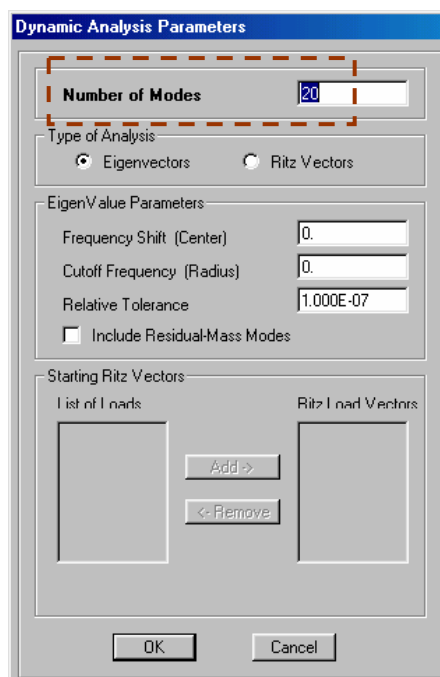


**Step 3-3:** "SPEC1" response spectrum function has been defined already in list, click **OK** to end this step.

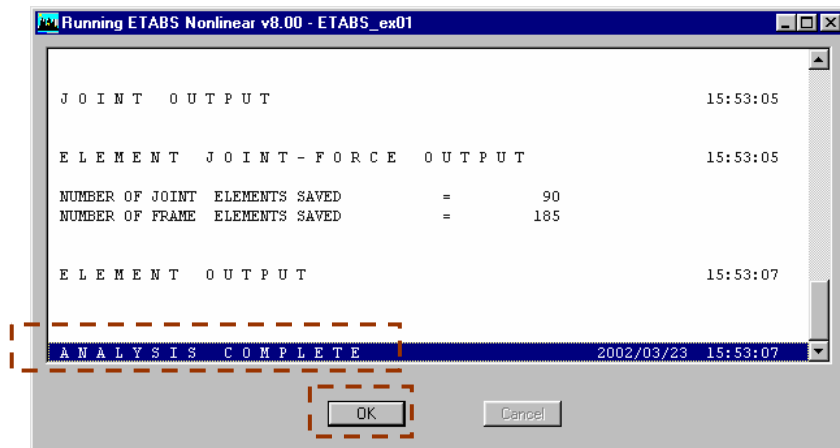
## 4. Run Analysis




**Step 4-1:** Go to **Analysis > Set Analysis Options** and click on **Set Dynamic Parameters**.



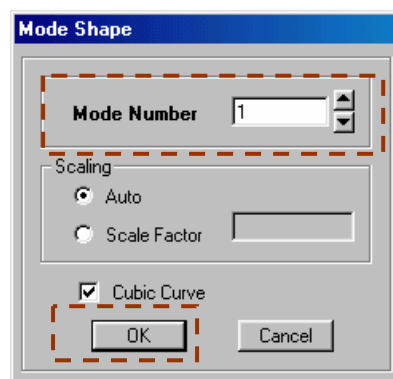
**Step 4-2:** Enter **Number of Modes** = "20" and click **OK** 2 times.




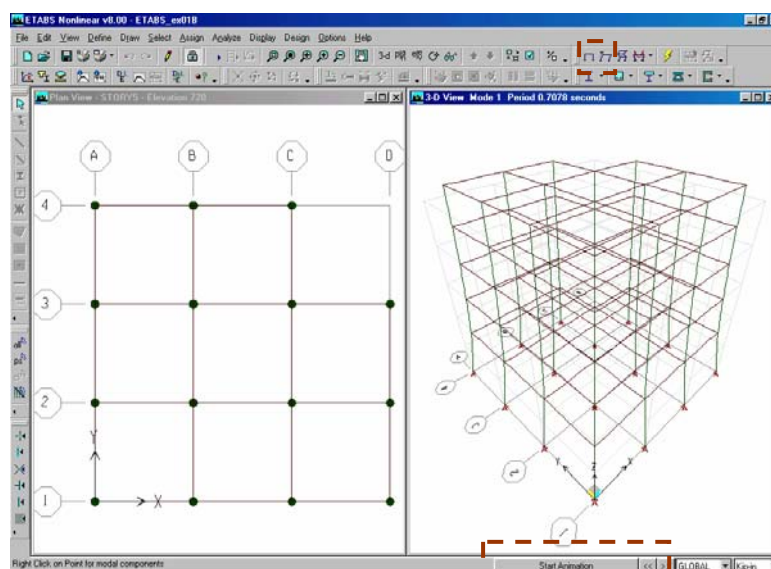
**Step 4-3:** Click on  or go to **Analysis > Run Analysis**, click on **Run** from Analysis Options "Window" and wait until ETABS displays "ANALYSIS COMPLETE" and click **OK**.


## 5. View dynamic analysis results

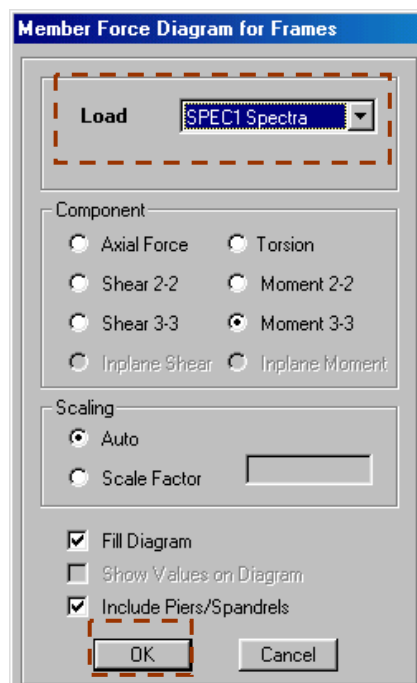
2 main types of dynamic analysis results can be displayed, first the mode shapes and second the member forces/stresses.




**Step 5-1:** Click on  or go to **Display > Show Mode Shape**, select **Mode Number** and click **OK**.

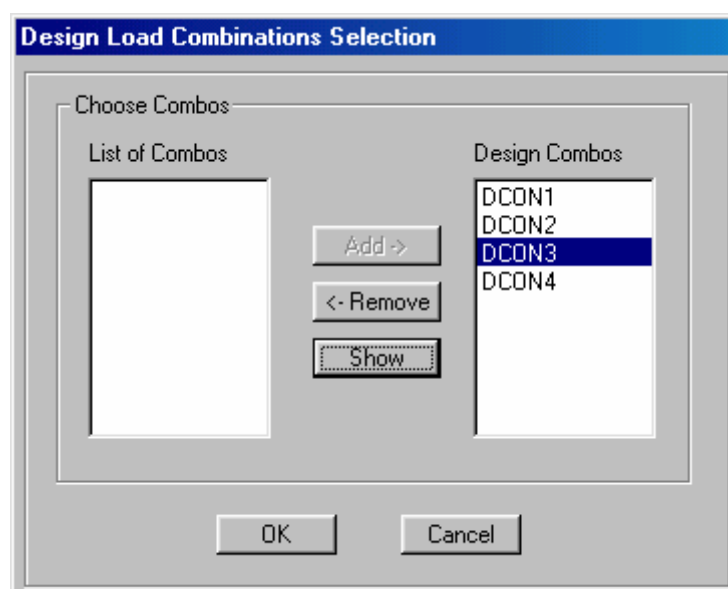


**Step 5-2:** Mode shape can be displayed in animation by clicking **Start Animation**. Click on  to change back to normal mode.



**Step 5-3:** Same as static analysis results, axial force, shear, moment or torsion can be displayed by clicking on  and selecting "SPEC1 Spectra". Click **OK**.

## 6. Design Concrete Frame



**Step 6-1:** Go to **Design > Concrete Frame Design > Select Design Combo**. ETABS has defined load combination for every load cases automatically as shown in **Design Combos** list. Load combinations details can be displayed by selecting one of them and clicking on **Show**.



**Load Combination Data**


Load Combination Name: DCON3

Load Combination Type: ADD

Define Combination

Case Name	Scale Factor
DEAD Static Load	1.05
LIVE Static Load	1.275
SPEC1 Spectra	1.4025
SUPERDL Static Load	1.05

OK

**Step 6-2:** Scale Factor for each load cases has been defined automatically by ETABS. Click on **OK** 2 times and click on  to start concrete frame design.

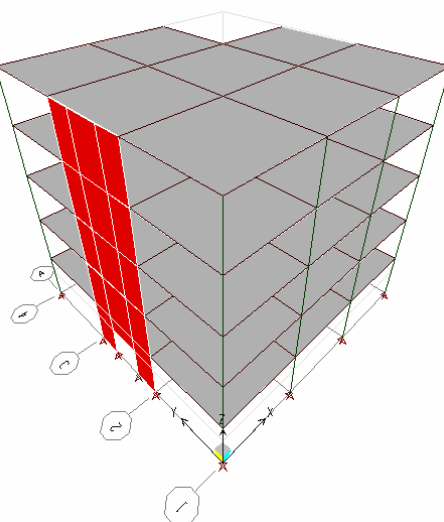
# Part C: Design of Shear Wall

## Step-by-Step Solution

### Problem

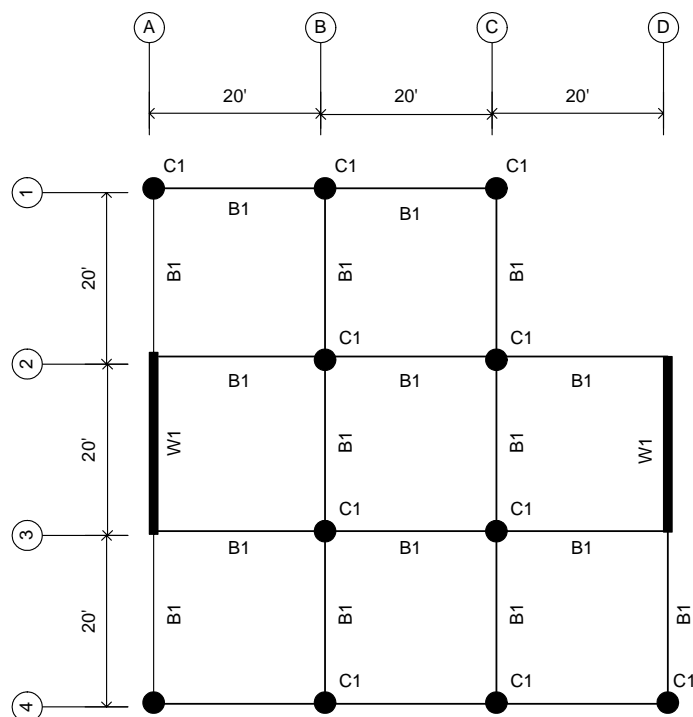
Add shear wall (12" thick.) to the model between grid line 2 and 3 along grid line A and D as shown in the following figures. Door openings on the wall are provided only at the first floor level.

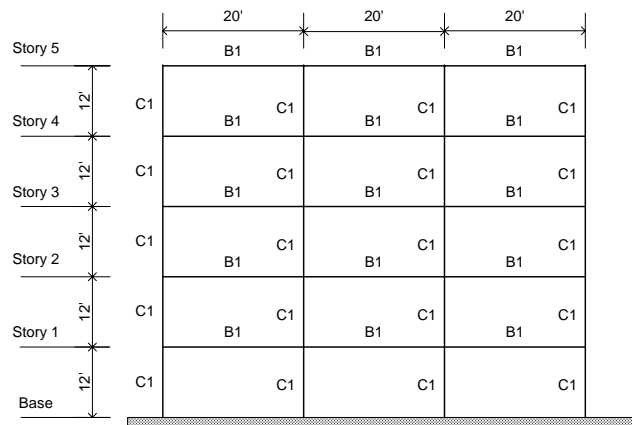
**3D View of the building**



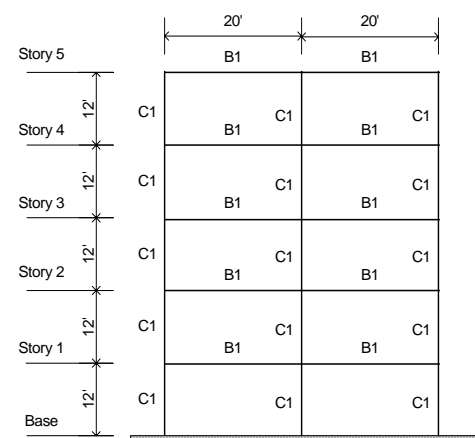
**Plan and Elevation View**

Plan View at Roof Floor

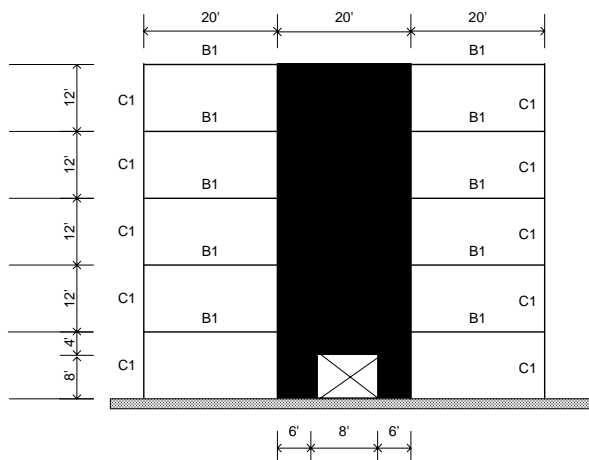




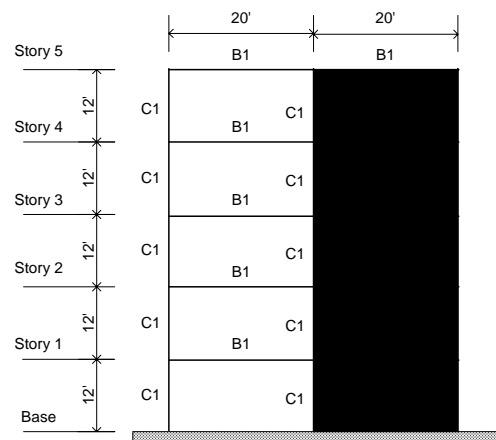
Elevation At Grid Lines 2, 3, 4, B and C



Elevation at Grid Line 1



Elevation at Grid Line A



Elevation at Grid Line D

## Typical Shear Wall Design Procedure

Following is a typical shear wall design process that might occur for a new building. Note that the sequence of steps you may take in any particular design may vary from this but the basic process will be essentially the same.

1. Use the **Options menu > Preferences > Shear Wall Design** command to review the shear wall design preferences and revise them if necessary. Note that there are default values provided for all shear wall design preferences so it is not actually necessary for you to define any preferences unless you want to change some of the default preference values.
2. Create the building model. See the section titled "Modeling Process" in Chapter 6 for more information.
3. Run the building analysis using the **Analyze menu > Run Analysis** command.
4. Assign the wall pier and wall spandrel labels. Use the **Assign menu > Frame/Line > Pier Label**, the **Assign menu > Shell/Area > Pier La-**

bel, the **Assign menu > Frame/Line > Spandrel Label**, and the **Assign menu > Shell/Area > Spandrel Label** commands to do this.

Note that the labels can be assigned before or after the analysis is run.

5. Assign shear wall overwrites, if needed, using the **Design menu > Shear Wall Design > View/Revise Pier Overwrites** and the **Design menu > Shear Wall Design > View/Revise Spandrel Overwrites** commands. Note that you must select piers or spandrels first before using these commands. Also note that there are default values provided for all pier and spandrel design overwrites so it is not actually necessary for you to define any overwrites unless you want to change some of the default overwrite values.

Note that the overwrites can be assigned before or after the analysis is run.

**Important note about selecting piers and spandrels:** You can select a pier or spandrel simply by selecting any line or area object that is part of the pier or spandrel.

6. If you want to use any design load combinations other than the default ones created by ETABS for your shear wall design then click the **Design menu > Shear Wall Design > Select Design Combo** command. Note that you must have already created your own design combos by clicking the **Define menu > Load Combinations** command.
7. Click the **Design menu > Shear Wall Design > Start Design/Check of Structure** command to run the shear wall design.
8. Review the shear wall design results. To do this you might do one of the following:
  1. Click the **Design menu > Shear Wall Design > Display Design Info** command to display design information on the model.
  2. Right click on a pier or spandrel while the design results are displayed on it to enter the interactive wall design mode. Note that while you are in this mode you can revise overwrites and immediately see the new design results.

If you are not currently displaying design results you can click the **Design menu > Shear Wall Design > Interactive Wall Design** command and then right click a pier or spandrel to enter the interactive design mode for that element.

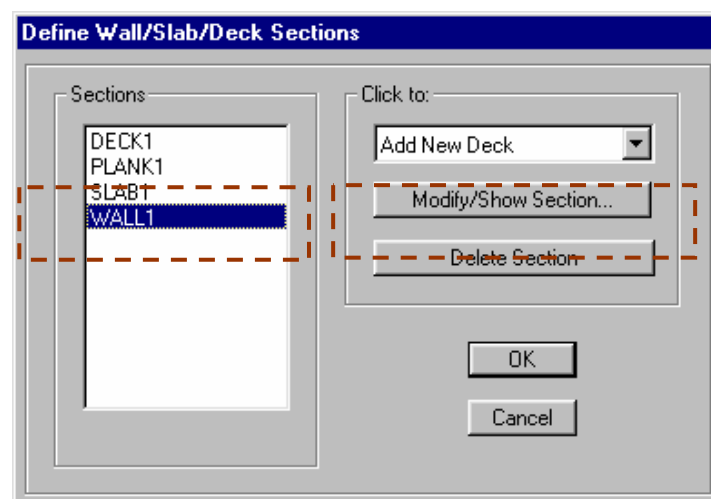
1. Use the **File menu > Print Tables > Shear Wall Design** command to print shear wall design data. If you select a few piers or spandrels before using this command then data is printed only for the selected elements.
2. If desired, revise the wall pier and/or spandrel overwrites, rerun the shear wall design, and review the results again. Repeat this step as many times as needed.
3. If desired, create wall pier check sections with user-defined (actual) reinforcing specified for the wall piers using the Section Designer utility. Use the **Design menu > Shear Wall Design > Define Pier Sections for Checking** command to define the sections in Section Designer. Be


sure to indicate that the reinforcing is to be checked. Use the **Design menu > Shear Wall Design > Assign Pier Sections for Checking** command to assign these sections to the piers. Rerun the design and verify that the actual flexural reinforcing provided is adequate.

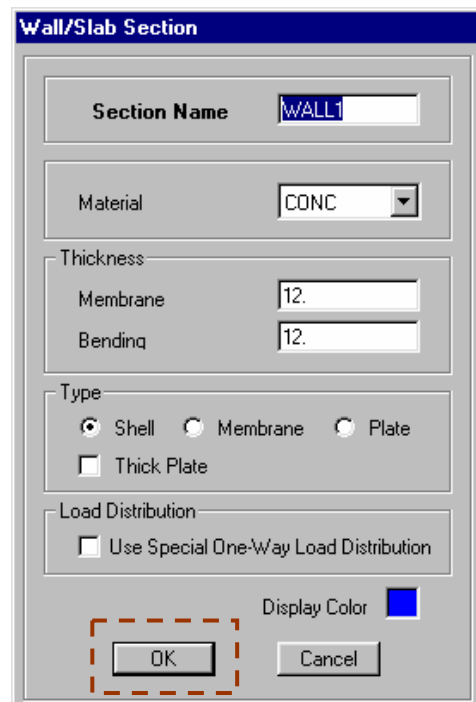
4. Assign these check sections to the piers, change the pier mode from Design to Check, and rerun the design. Verify that the actual flexural reinforcing provided is adequate.
5. If necessary, revise the geometry or reinforcing and rerun the design.
6. Print or display selected shear wall design results if desired.

Note that shear wall design is performed as an iterative process. You can change your wall design dimensions and reinforcing during the design process without rerunning the analysis. However, you always want to be sure that your final design is based on analysis properties (wall dimensions) that are consistent with your design (actual) wall dimensions.

## Define Wall Section



**Step 1-1:** Click on  or go to **Define > Wall/Slab/Deck Sections**, select "WALL1" and click on **Modify/Show Section**.



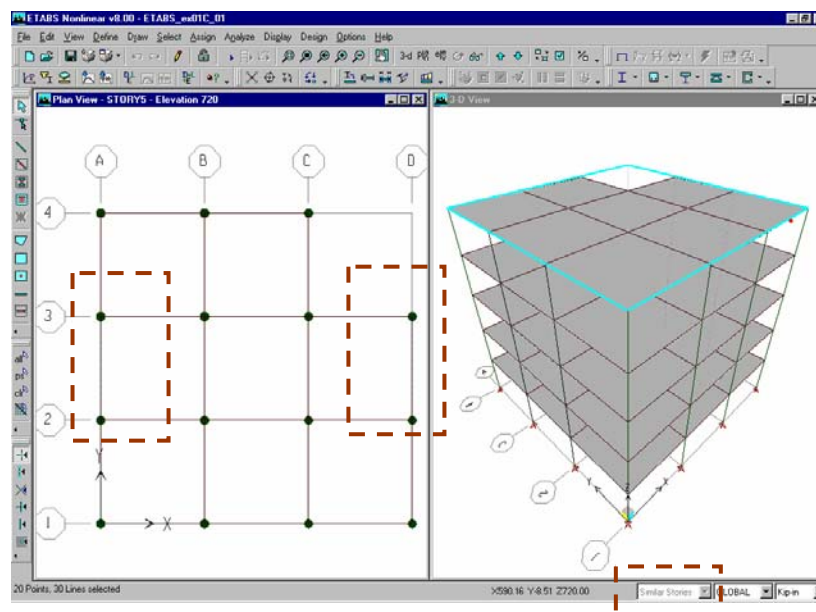
The 'Wall/Slab Section' dialog box is shown with the following settings:

- Section Name:** WALL1
- Material:** CONC
- Thickness:**
  - Membrane: 12.
  - Bending: 12.
- Type:**
  - ☒ Shell
  - ☐ Membrane
  - ☐ Plate
  - ☐ Thick Plate
- Load Distribution:**
  - ☐ Use Special One-Way Load Distribution
- Display Color:** Blue
- Buttons:** OK and Cancel. The OK button is highlighted with a dashed orange border.

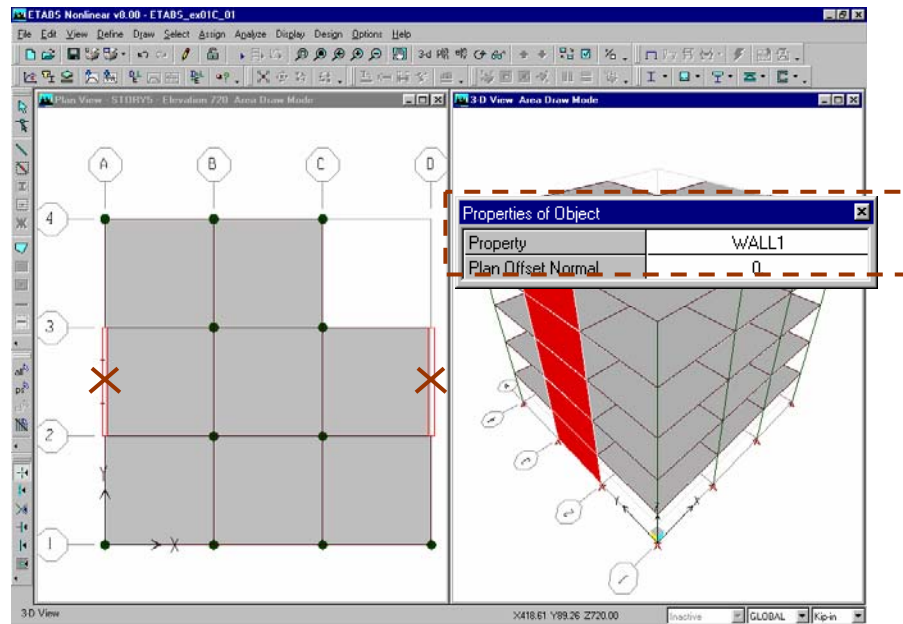
**Step 1-2:** Accept the defaults by clicking **OK** to 2 times.


## 7. Draw Wall Sections

Columns and beams around wall areas will be deleted before drawing the wall sections.



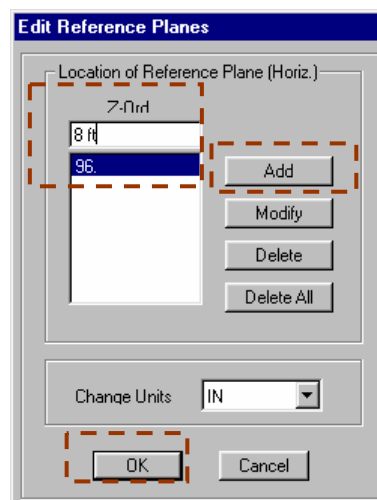
**Step 2-1:** Change "Plan View" window to "STORY5" and select "Similar Stories". To delete all frame objects between grid line 2 and 3 along grid line A and D, draw section rectangular to cover these areas and press **Delete** on the keyboard



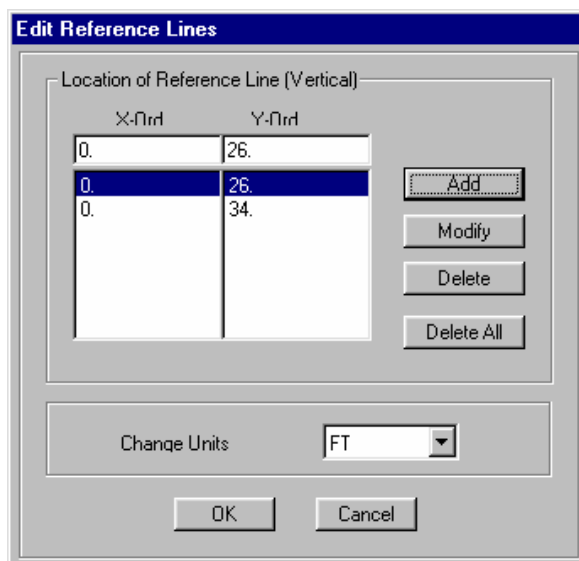
**Step 2-2:** To assign Wall in "Plan View", select , select **Property** = "WALL1" and click on 2 bays. Walls will be added to all stories as shown in "3D View" window.

## 8. Define Reference Planes and Lines

Reference planes and lines are required to define additional working planes other than the real floors (stories). These are useful to define openings, staircases, ramps etc. This is similar to defining additional grid lines between the real floors. In this example these features will be used to create door openings on the wall.



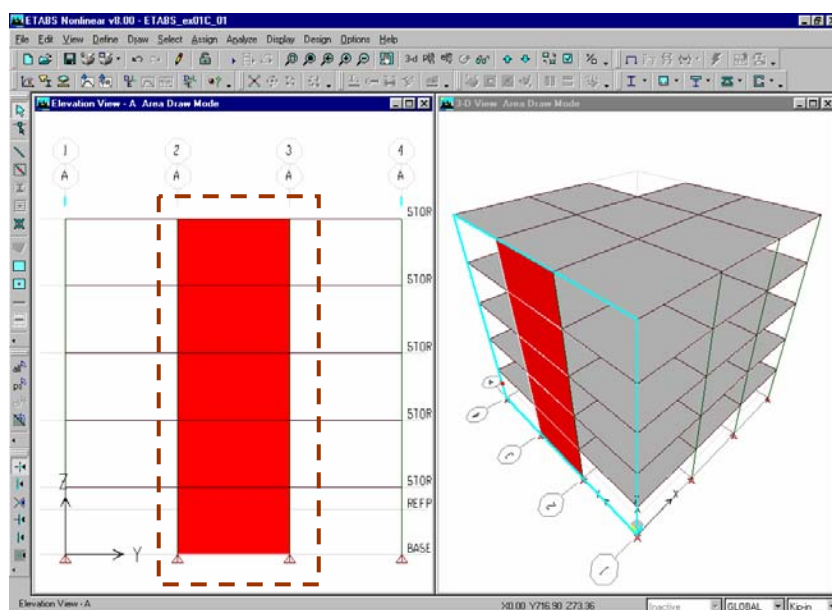
**Step 3-1:** Reference plane will be defined to at the top and bottom of door opening. Go to **Edit > Edit Reference Planes**, Enter "8 ft" (vertical location of the reference plane), click **Add** and click **OK**.



**Step 3-2:** Reference lines are required at the left and right side of the door opening. Go to **Edit > Edit Reference Lines**, Enter X and Y coordinate as shown in above figure, click **Add** twice and click **OK**.

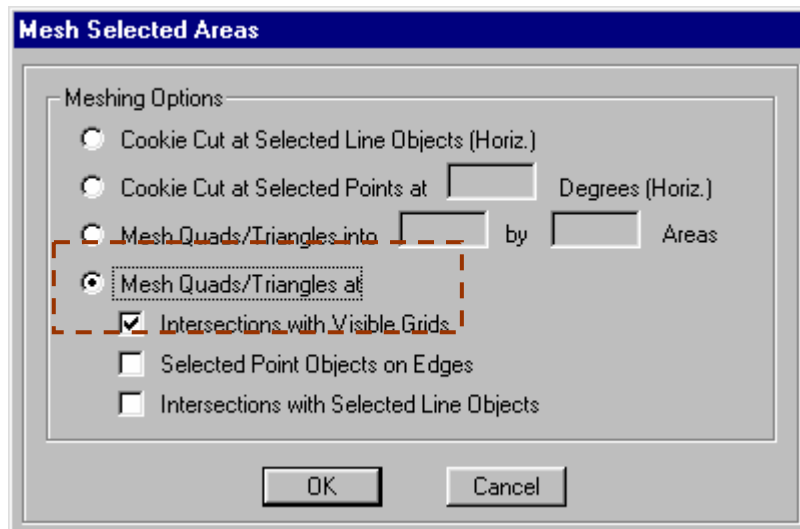
## 9. Add Wall Opening

Walls on 1<sup>st</sup> floor along grid line A will be meshed by using reference plan and lines that have been defined in the previous step. The area object (wall) at the door opening will be deleted.

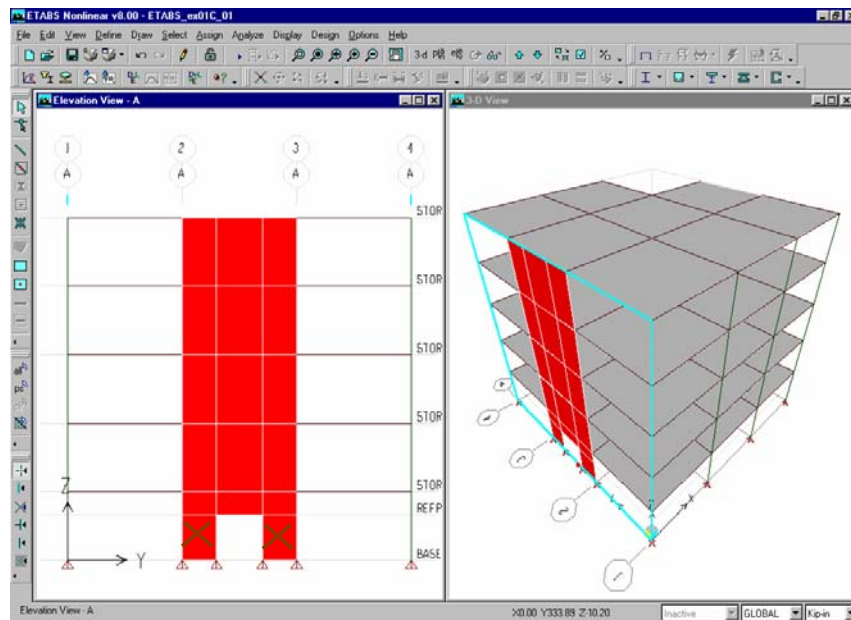


**Step 4-1:** Change "Plan View" to "Elevation View" by clicking on **ele** and selecting "A" from "Set Elevation View" window. Draw selection rectangular to cover all walls in this "Elevation View".





**Step 4-2:** Go to **Edit > Mesh Areas**, select "Mesh Quads/Triangles at" and select "Intersections with Visible Grids".



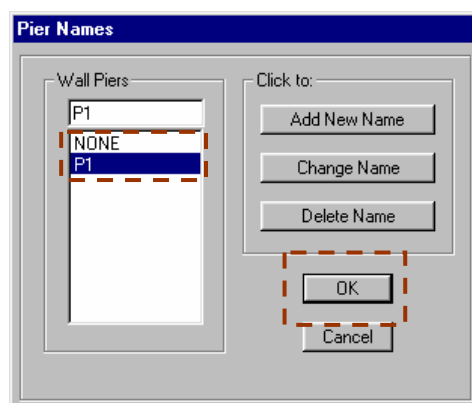
**Step 4-3:** All walls on this side are meshed based on reference plane and line, click on wall at door opening area and press **Delete** on the keyboard.

## 10. Assign Piers Labels

Piers labels are essential for the analysis and design of shear walls.



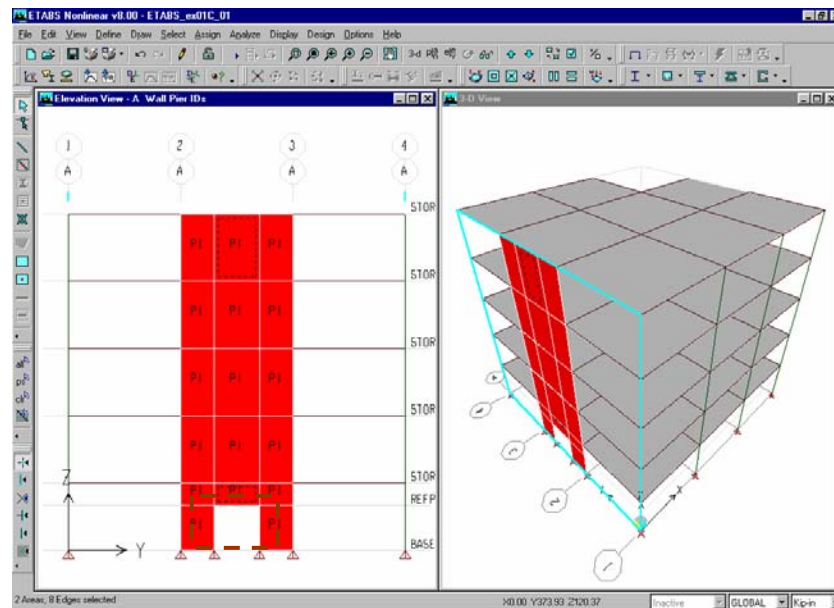
**Step 5-1:** Select all wall sections, go to **Select > by Wall/Slab/Deck Sections**, select "Wall1" and click **OK**.



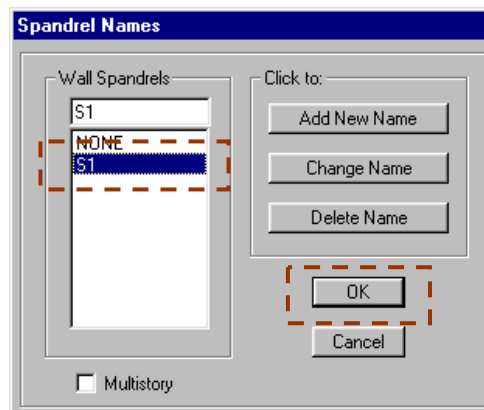
**Step 5-2:** Go to **Assign > Shell /Area > Pier label**, Select "P1" and click **OK**.

## 11. Assign Spandrels Labels

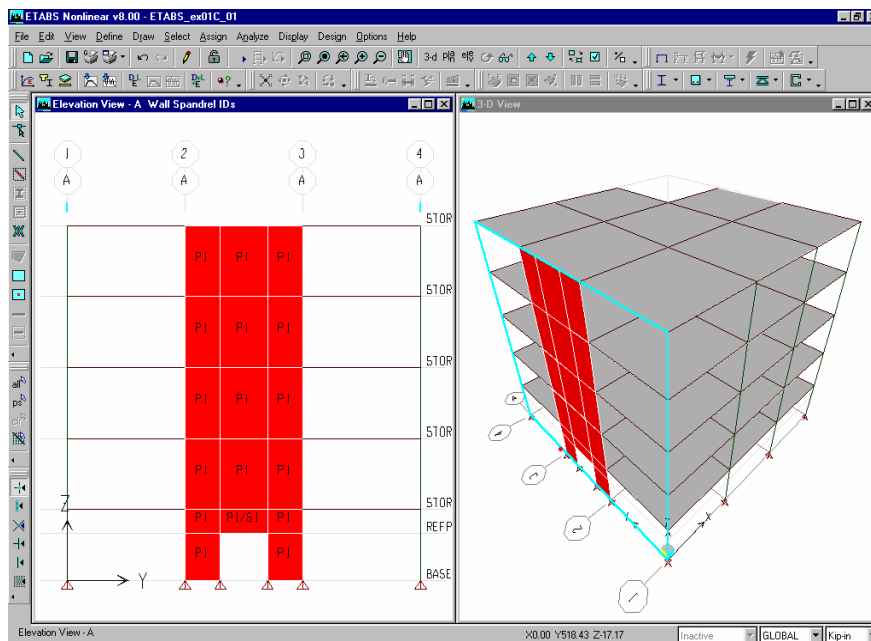
Similar to the piers spandrels labels are required by ETABS for proper analysis and design of shear walls. Normally they are provided over the openings.



**Step 6-1:** Click on wall at the top of door opening. Go to **Assign > Shell /Area > Spandrel** label.



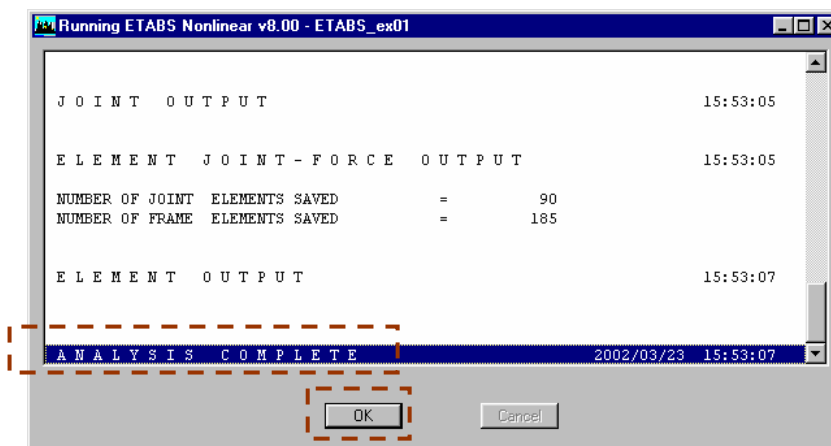
**Step 6-2:** Select "S1" and click **OK**.




**Step 6-3:** Wall over the door opening is labeled as "P1" and spandrel ("S1").

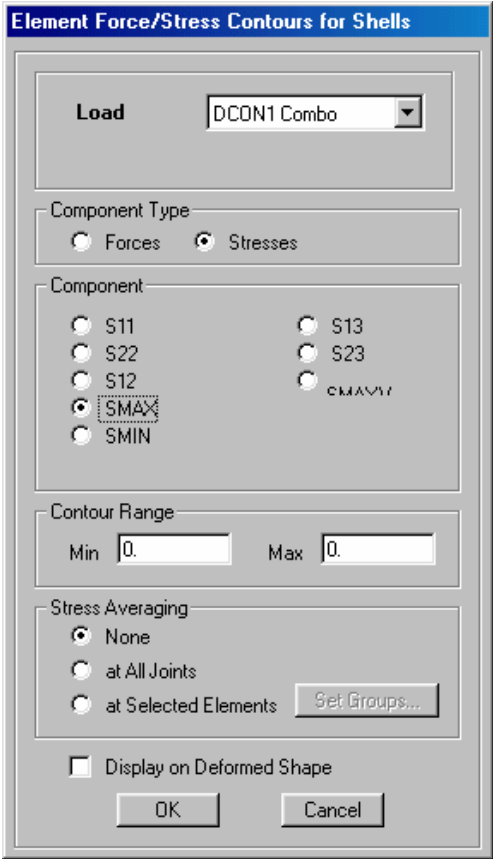
## 12. Run Analysis


Analysis option has been setup for dynamic analysis from Part B already. Rerun the problem.

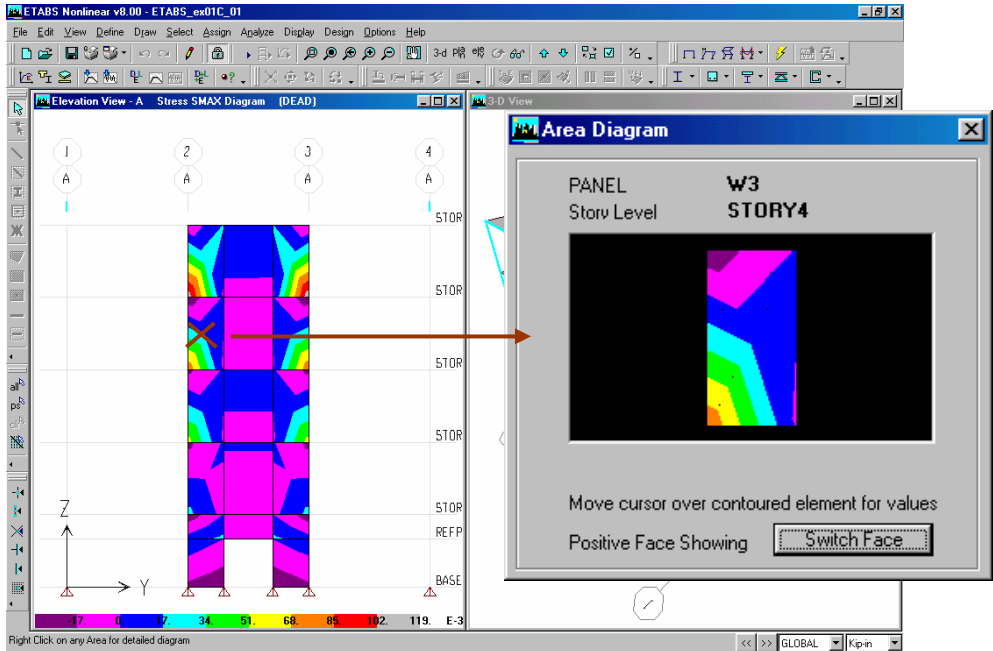


**Step 7-1:** Click on  or go to **Analysis > Run Analysis**, click on **Run** from Analysis Options "Window" and wait until ETABS displays "ANALYSIS COMPLETE" and click **OK**.

\_\_\_\_\_

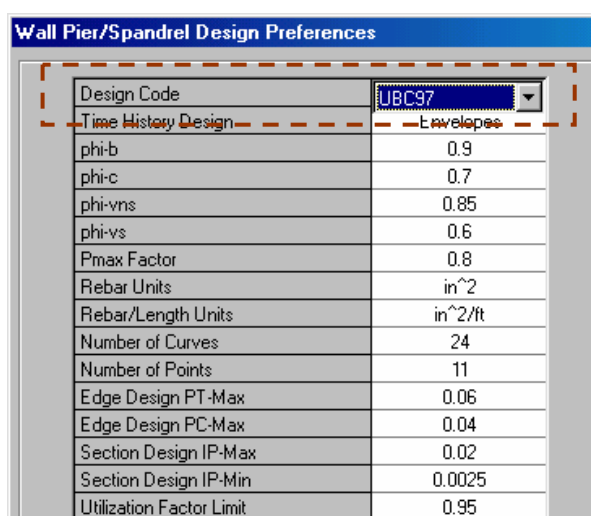


**Step 8-1:** Click on  or go to **Display > Show Member Forces/Stresses Diagram > Shell Stresses Forces**, select result parameter and click **OK**.



**Step 8-2:** Stresses or forces are displayed as color contours, click the right mouse on any one of these walls. “Area Diagram” at that wall will be displayed. Move the cursor over this diagram to check the value at desired location. Click **X** at the top-right of this window to close this “Area Diagram”.

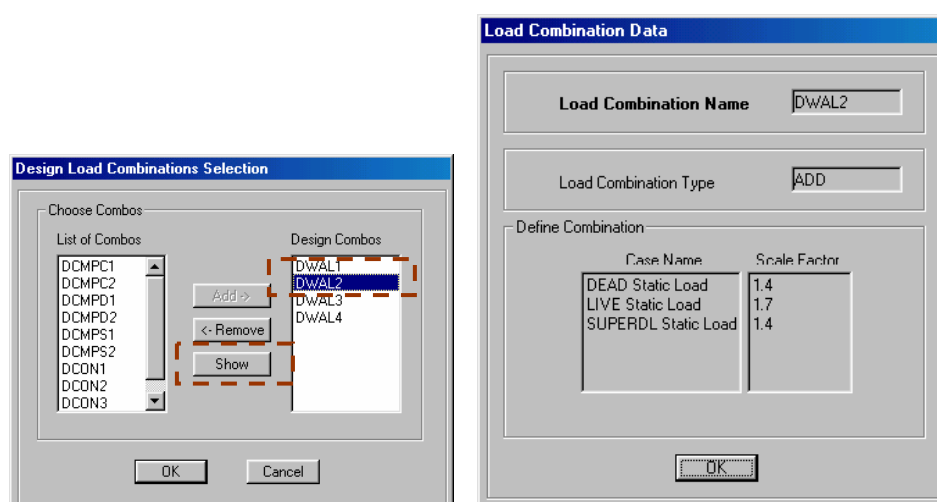
## 14. Design Shear Walls



Wall Pier/Spandrel Design Preferences

Design Code	UBC97
Time History Design	Envelopes
phi-b	0.9
phi-c	0.7
phi-vns	0.85
phi-vs	0.6
Pmax Factor	0.8
Rebar Units	in <sup>2</sup>
Rebar/Length Units	in <sup>2</sup> /ft
Number of Curves	24
Number of Points	11
Edge Design PT-Max	0.06
Edge Design PC-Max	0.04
Section Design IP-Max	0.02
Section Design IP-Min	0.0025
Utilization Factor Limit	0.95

**Step 9-1:** To select design code, go to **Options > Preferences > Shear Wall Design** menu, select "UBC97" and click **OK**.



**Design Load Combinations Selection**

Choose Combos

List of Combos

- DCMPC1
- DCMPC2
- DCMPD1
- DCMPD2
- DCMPS1
- DCMPS2
- DCON1
- DCON2
- DCON3

Design Combos

- DWAL1
- DWAL2
- DWAL3
- DWAL4

Buttons: Add, Remove, Show, OK, Cancel

**Load Combination Data**

Load Combination Name: DWAL2

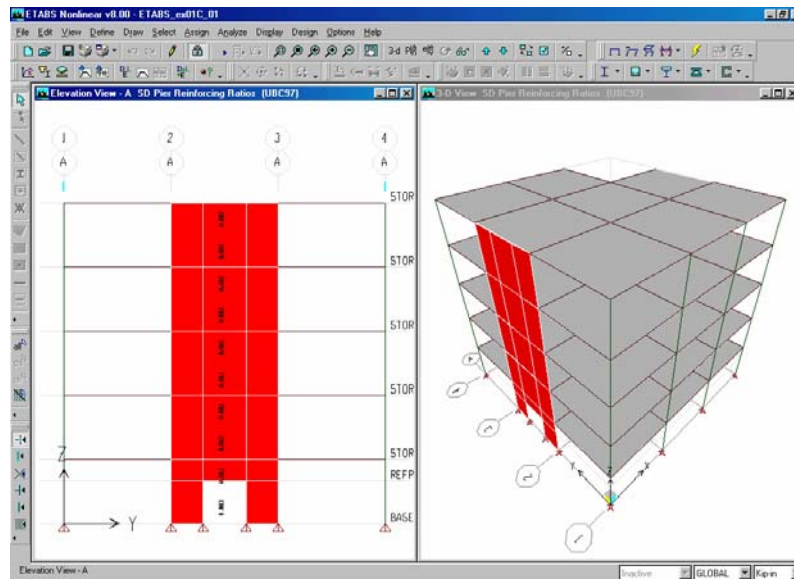
Load Combination Type: ADD


Define Combination

Case Name	Scale Factor
DEAD Static Load	1.4
LIVE Static Load	1.7
SUPERDL Static Load	1.4

Buttons: OK

**Step 9-2:** To see load combination for design, go to **Design > Concrete Frame Design > Design Combo**. 2 load combinations are defined and selected for design automatically, select "DCON2" and click **Show**. Load factors have been defined for to comply with the selected code. Click **OK** 2 times.



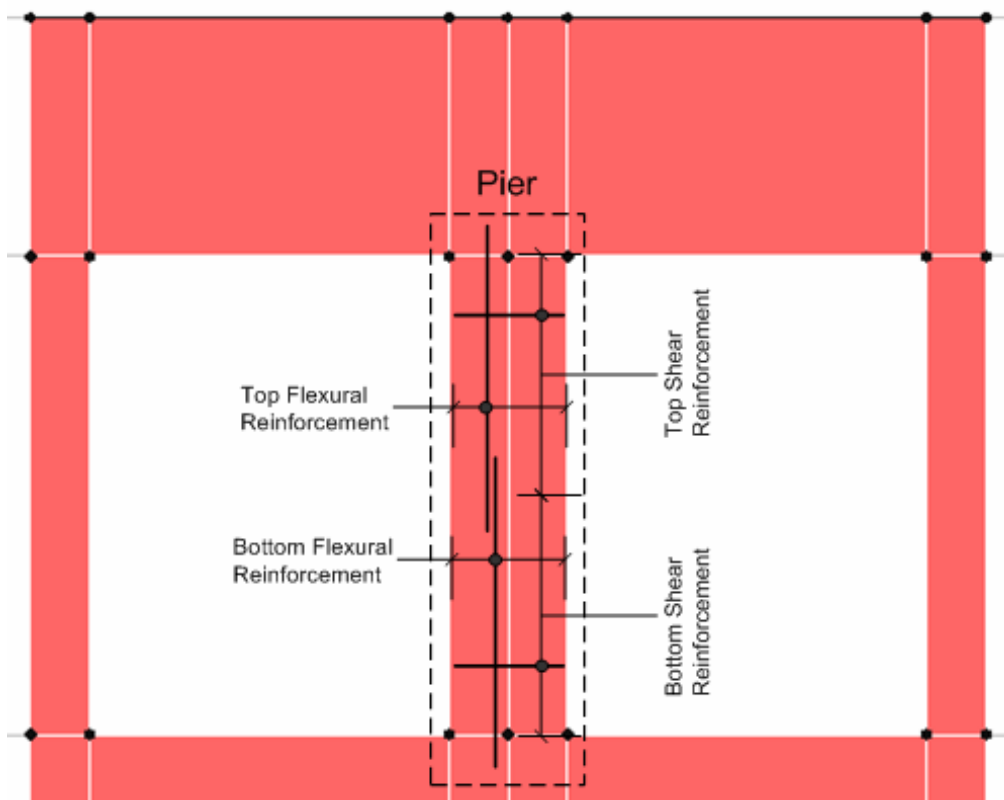
**Step 9-3:** Click on  or **Design > Shear Wall Design > Start Design/Check Structure**. After shear wall design is over, ETABS will display “Pier Reinforcing Ratios” as active window.

Uniform Reinforcing Pier Section - Flexural Design (UBC97)									
Story ID: STORY2 Pier ID: P1 X Loc: 360 Y Loc: 360 Units: Kip-in									
Flexural Design for P-M2-M3 (RLLF = 1.000)									
Station	Required	Current	Flexural					Pier	
Location	Reinf Ratio	Reinf Ratio	Combo	Pu	M2u	M3u		Ag	
Top	0.0025	0.0064	DWAL4	-348.333	4721.645	-2419.155		2880.000	
Bottom	0.0025	0.0064	DWAL4	-419.608	4534.067	-2398.168		2880.000	
Shear Design - First Inadequate Leg or Leg Requiring Most Rebar per Unit Length (EQF = 1.000)									
Station	Rebar	Shear					Capacity	Capacity	
Location	in <sup>2</sup> /ft	Combo	Pu	Mu	Vu		phi Vc	phi Vn	
Top Leg 1	0.360	DWAL3	-240.780	75.392	-0.676		317.415	524.775	
Bot Leg 1	0.360	DWAL3	-288.297	143.406	-0.676		323.117	530.477	
Boundary Element Check - First Inadequate Leg or Leg Requiring Longest Boundary Zone									
Station	B-Zone	B-Zone							
Location	Length	Combo	Pu	Mu	Vu	Pu/Po			
Top Leg 2	36.000	DWAL4	-167.748	-2312.266	0.764	-0.0164			
Bot Leg 2	36.000	DWAL4	-203.386	-2202.242	0.764	-0.0199			
Number of legs not checked because Pu/Po < -0.35 (top, bottom) = 0, 0									
Number of legs not requiring boundary zones (top, bottom) = 1, 1									
Number of legs not requiring boundary zones (top, bottom) = 1, 1									
Combos... Overwrites... OK Cancel									

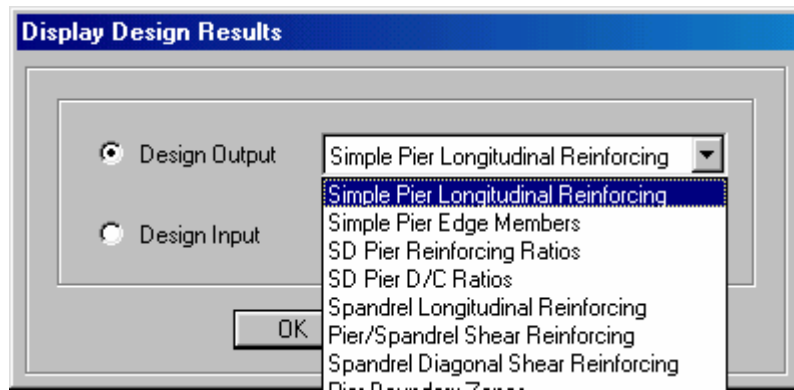
**Step 9-4:** Right mouse click on any wall to open pier section design details, click **OK** to close this window.

## Reinforcement Location for Pier

Uniform Reinforcing Pier Section -Design (ACI 318-99)								
Story ID: STORY10		Pier ID: P FRONT2	X Loc: 900	Y Loc: 800	Units: Kgf-cm			
Flexural Design for P-M2-M3 (RLLF = 1.000)								
Station	Required	Current	Flexural					Pier
Location	Reinf Ratio	Reinf Ratio	Combo	Pu	M2u	M3u		Ag
Top	0.0025	0.0047	UDWAL14	9353.895	0.000	7242.872		1000.001
Bottom	0.0025	0.0047	UDWAL14	10091.760	0.000	-6812.358		1000.001
Shear Design								
Station	Rebar	Shear				Capacity	Capacity	
Location	cm^2/m	Combo	Pu	Mu	Vu	phi Vc	phi Vn	
Top Leg 1	5.000	UDWAL13	4885.713	169450.886	-1786.765	3519.595	6399.595	
Bot Leg 1	5.000	UDWAL13	3782.292	160138.815	-1726.146	3509.726	6389.726	
Boundary Element Check								
Station	B-Zone	B-Zone						
Location	Length	Combo	Pu	Mu	Vu	Pu/Po		
Top Leg 1	Not Needed	UDWAL12	11545.206	13952.275	-147.560	0.0467		
Bot Leg 1	Not Needed	UDWAL12	10831.310	-11237.078	-128.854	0.0438		



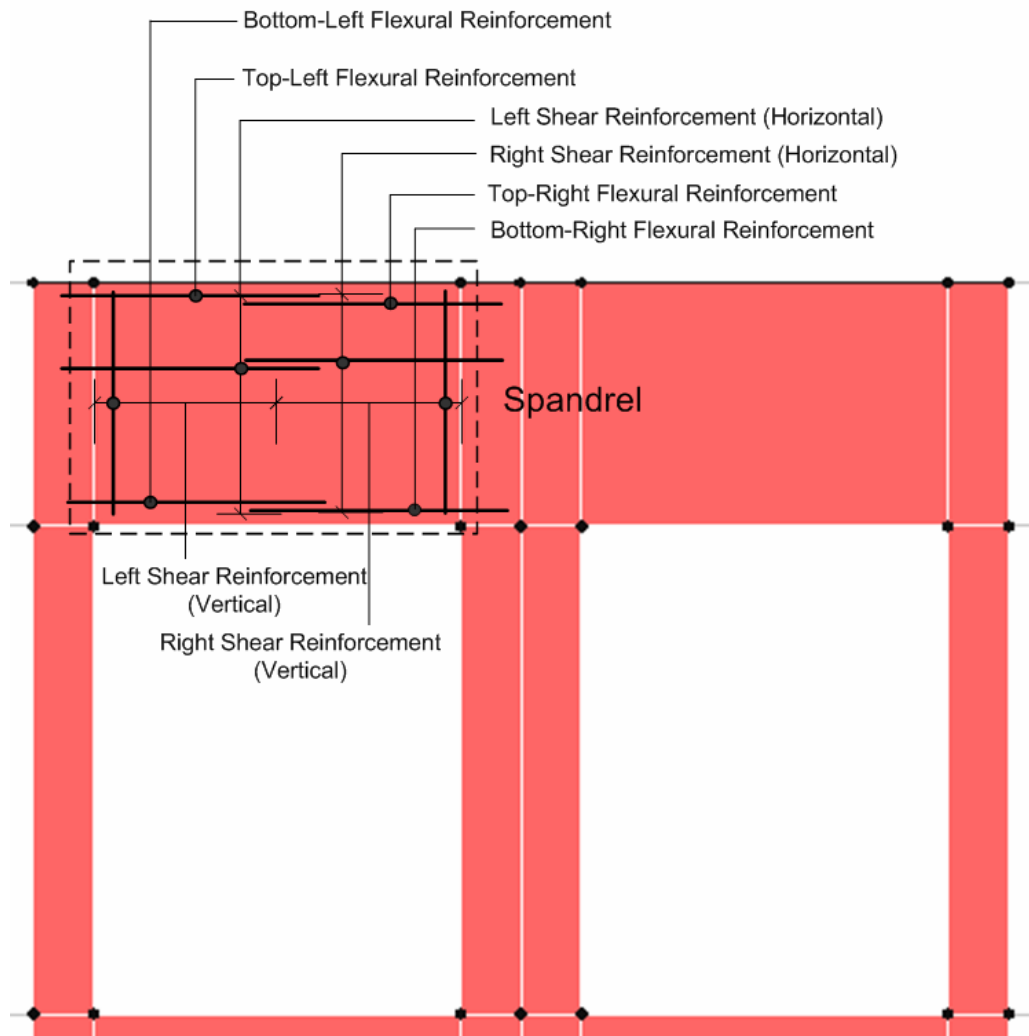




**Step 9-5:** Spandrel design details or any shear wall design input and/or output information can be displayed from **Design > Shear Wall Design > Display Design Info**.

### Reinforcement Location for Spandrel

Spandrel Design											
ACI 318-99		Story ID: STORY10		Spandrel ID: S1		X Loc: 725		Y Loc: 800		Units: Kgf-cm	
Flexural Design (RLLF = 1.000)											
Station Location		Top Steel cm <sup>2</sup>	Top Steel Ratio	Top Steel Combo	Mu						
Left		1.389	0.0634%	UDWAL11	-335842.411						
Right		2.679	0.1339%	UDWAL2	-644788.705						
Station Location		Bot Steel cm <sup>2</sup>	Bot Steel Ratio	Bot Steel Combo	Mu						
Left		0.245	0.0122%	UDWAL13	59440.619						
Right		0.000	0.0000	N/A	N/A						
Shear Design											
Station Location		Avert cm <sup>2</sup> /m	Ahoriz cm <sup>2</sup> /m	Shear Combo	Vu	Capacity Phi Vc	Capacity Phi Vs	Capacity Phi Vn			
Left		3.000	5.000	UDWAL11	3346.801	9583.672	5904.000	15487.672			
Right		3.000	5.000	UDWAL11	5250.416	9583.672	5904.000	15487.672			
Station Location		Adiag cm <sup>2</sup>	Shear Combo	Vu	Diag Reinf Required						
Left		1.743	UDWAL11	3346.801	No						
Right		2.735	UDWAL11	5250.416	No						



**Note:** Typical Detailing of Shear Wall

