Seismic Analysis & Design of 10 Story RC Building

(Equivalent Lateral Force)

Using ETABS

(Metric Units)

ACECOMS, AIT

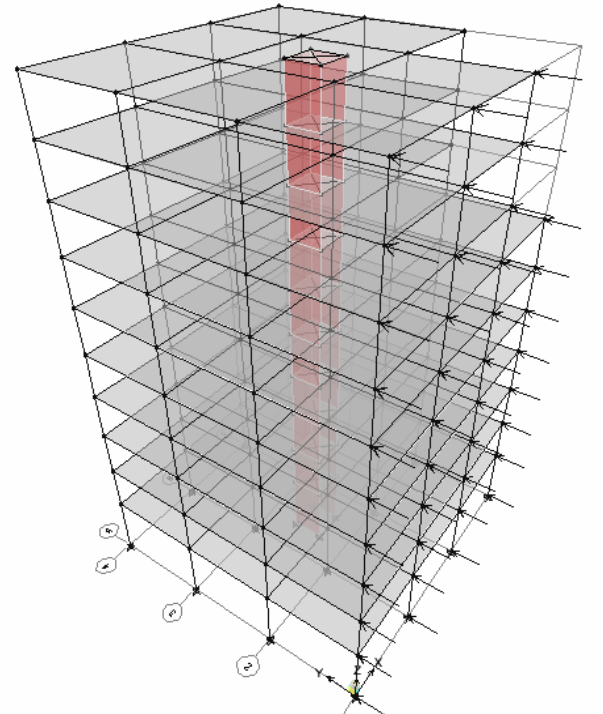


Table of Content

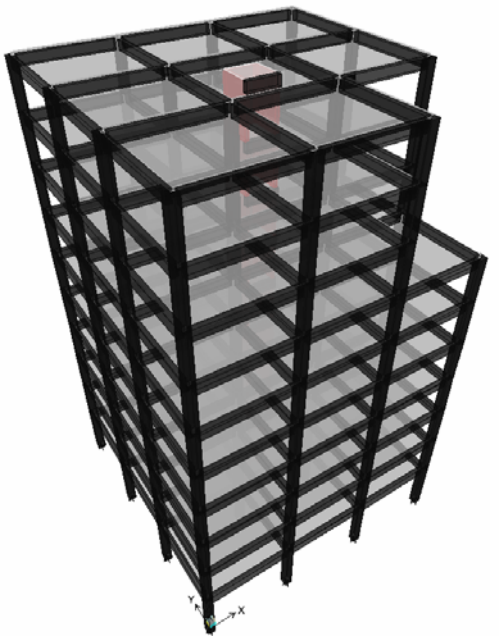
Objective

To demonstrate and practice step-by-step on the modeling, analysis and design of 10 story RC building for seismic equivalent lateral force.

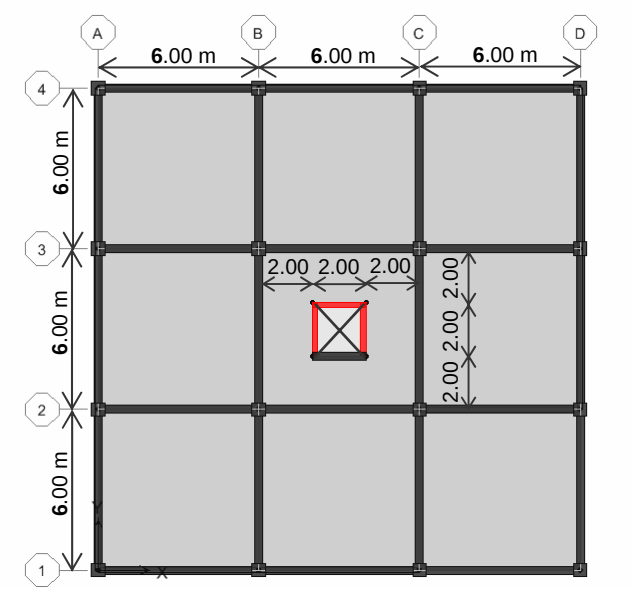
Problem

Carry out analysis, and design of 10 story RC building as shown in following details using IBC2000 equivalent lateral force.

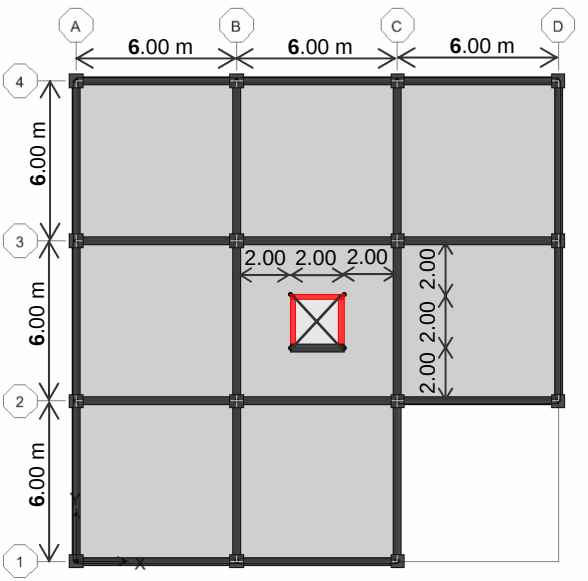
3D View



Plan View (Unit in m)

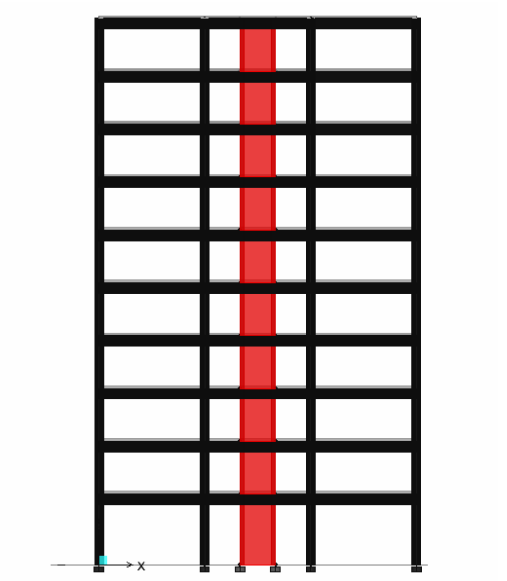


BASE – STORY 7

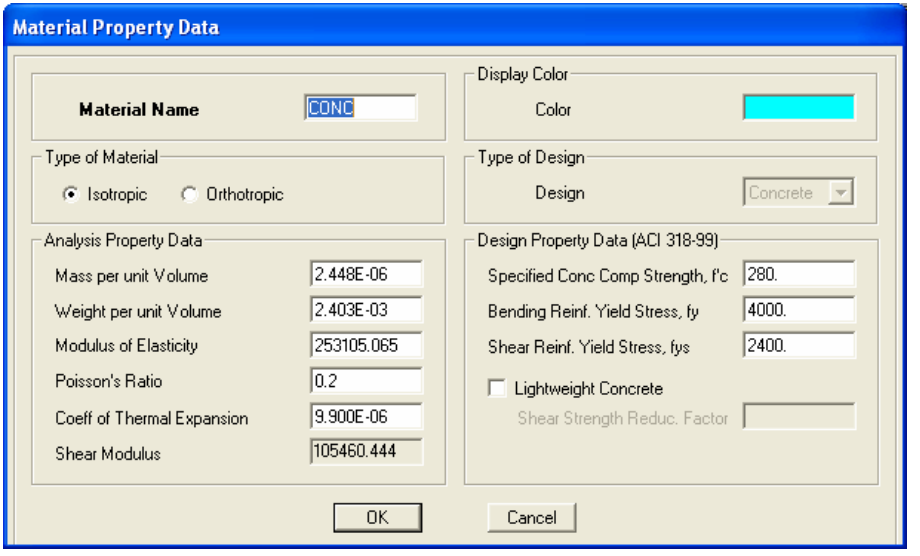


STORY 8 – STORY 10

Elevation View



Material Properties for Concrete (Unit in kg and cm)



Section Properties

Member	Dimension
Beam (width x Height)	30 x 60 cm
Column	50 x 50 cm
Slab	Thickness = 15 cm
Shear wall	Thickness = 20 cm

Story Height Data

Story	Height
Typical Story	3.00 m
Story at base of building	4.00 m

Static Load Cases

Load Name	Load Type	Details	Value
DEAD	Dead Load	Self Weight of Structural Members Calculate automatically using Self Weight Multiplier in ETABS	-
		Uniform Load on Slabs: (Finishing + Partition Load)	0.20 t/m^2
		Uniform Load on Beams: (Wall Load)	0.50 t/m
LIVE	Reducible Live Load	Uniform Load on Slabs: (Use Tributary Area: UBC97)	0.25 t/m^2

Wind Load Cases (UBC97)

Parameter	Load Case	
	WINDX	**WINDY**
Wind Direction	X	Y
Wind Speed	90 mph	
Exposure Type	B (Suburban area)	
Importance Factor	1 (Building normal importance)	

Equivalent Static Force Parameters (IBC2000)

Parameter	Values	Remark
Time Period (T)	1.47	Equation 16-39 (C_t = 0.020)
Response Modification Factor (R)	5.5	Table 1617.6 (Dual System: Ordinary RC Shear Wall)
Seismic Group	I	Section 1616.2
Site Class	E	Table 1615.1.1 (Soft Clay)
Response Acceleration at Short Period (S_s)	0.45	
Response Acceleration at 1 Second (S_1)	0.18	

Equivalent Static Force Case

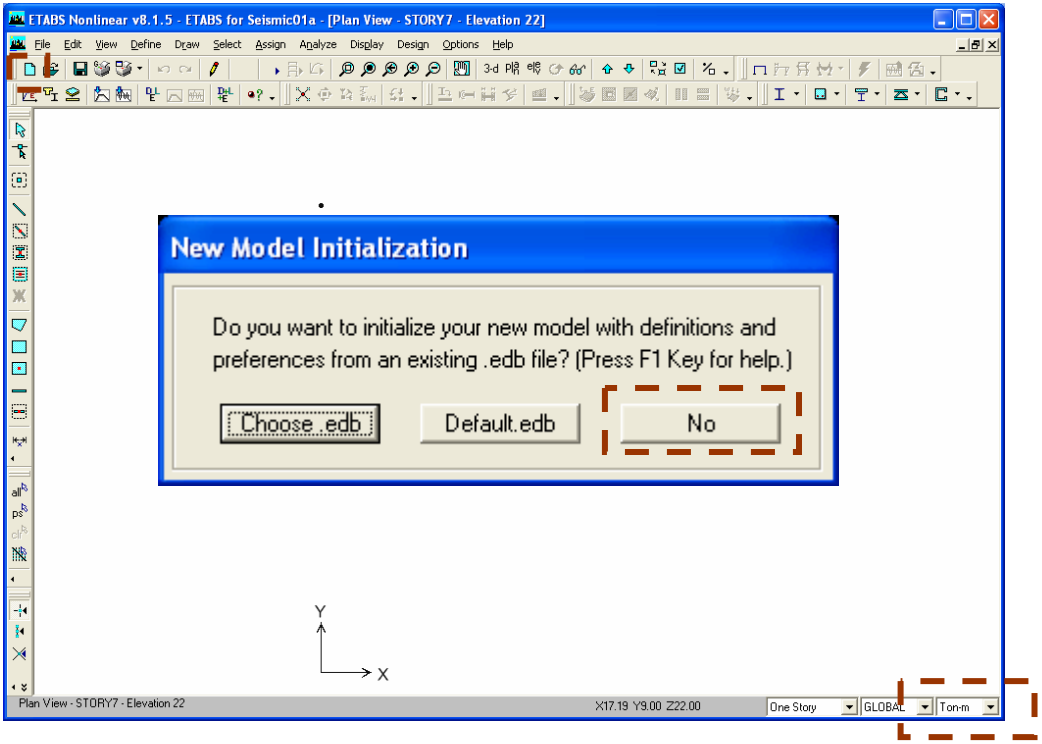
Load Case Name	Direction and Eccentricity	% Eccentricity
EQXA	X Dir + Eccen. Y	0.05
EQXB	X Dir - Eccen. Y	0.05
EQYA	Y Dir + Eccen. X	0.05
EQYB	Y Dir - Eccen. X	0.05

Step by Step

1. Start Model with Template

Step 1-1: Select Working Unit and Start New Model using Template

Start up screen of ETABS, select working unit to be “ton-m” at drop-down menu on the bottom-right of screen and click on *New Model* button to start new model using template



Note: 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.

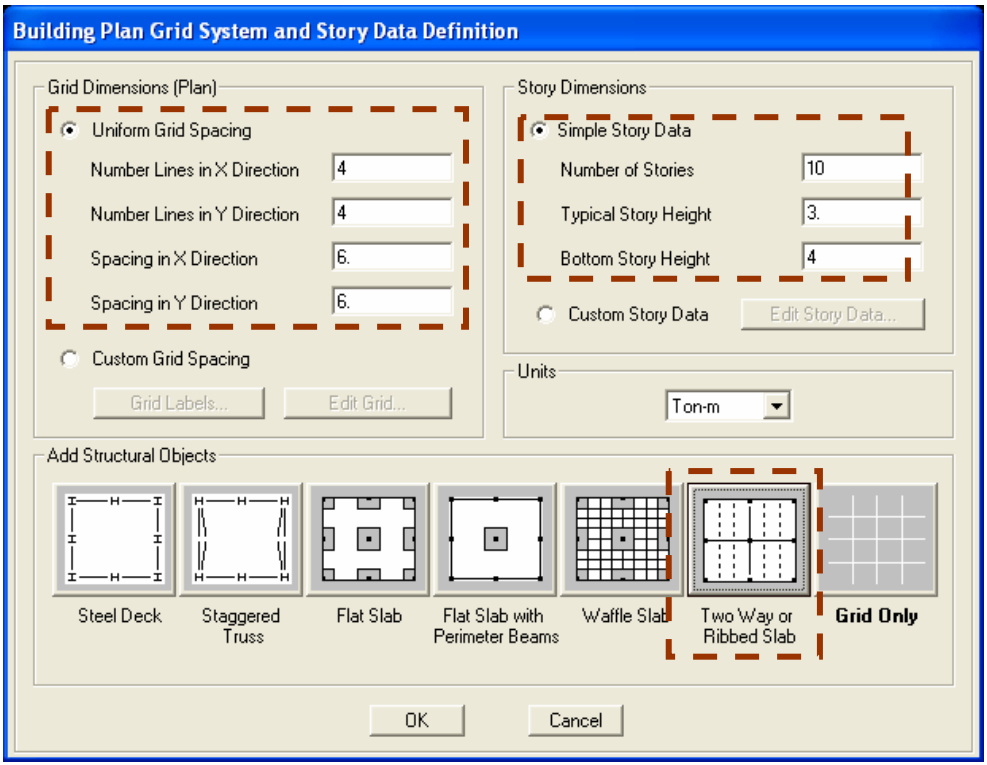
You should create your Default.edb file such that you most commonly click this button.

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. This will allow setting of the repeatedly used preferences.

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

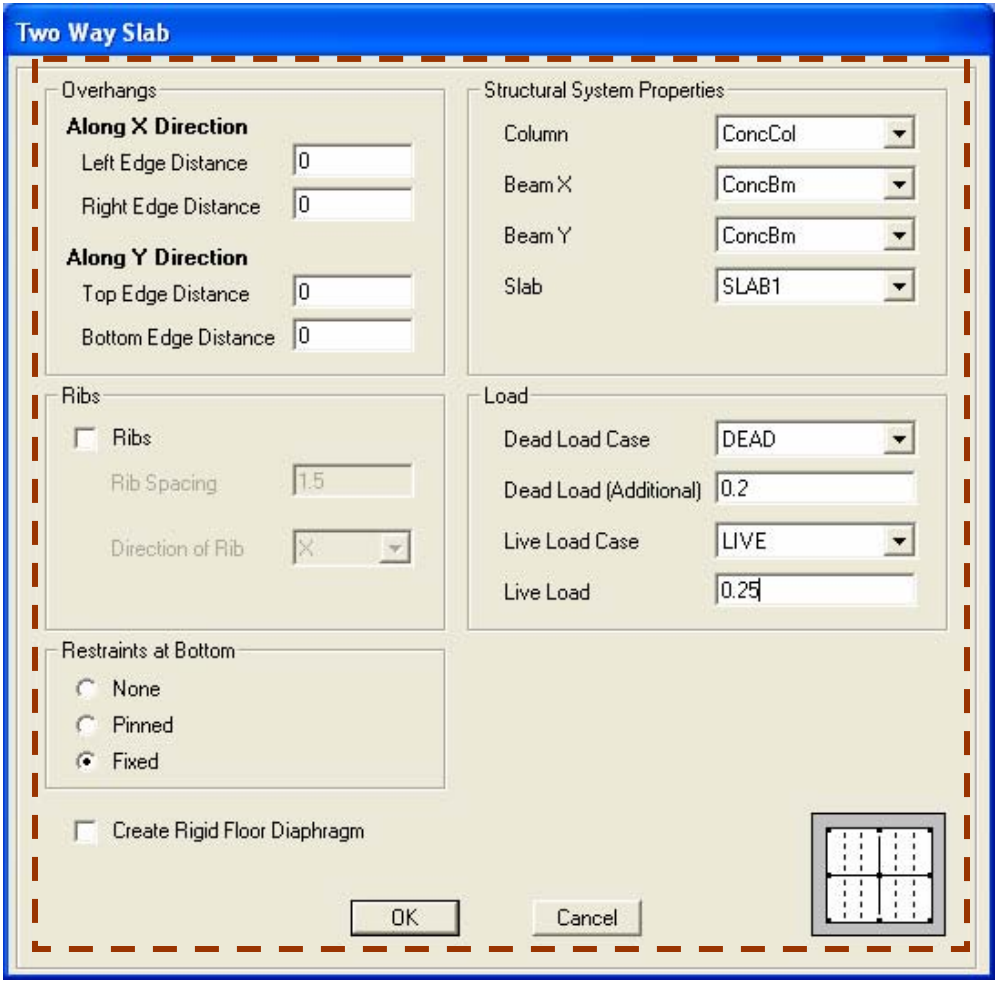
Step 1-2: Specify Grid and Story Dimension

Specify grid dimension and story dimension as shown in figure below. Select “Two Way or Ribbed Slab” from “Structural Objects” list.



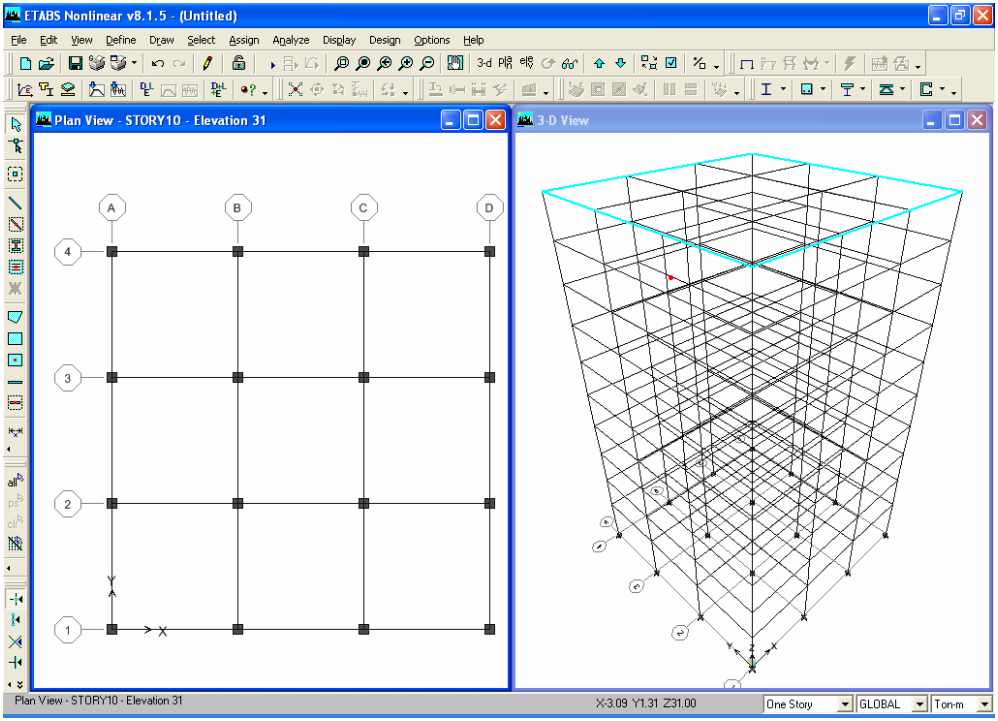
Step 1-3: Enter Two Way Slab System Parameters

Specify parameters as shown in figure below.



Step 1-4: Create Two Way Slab System Model

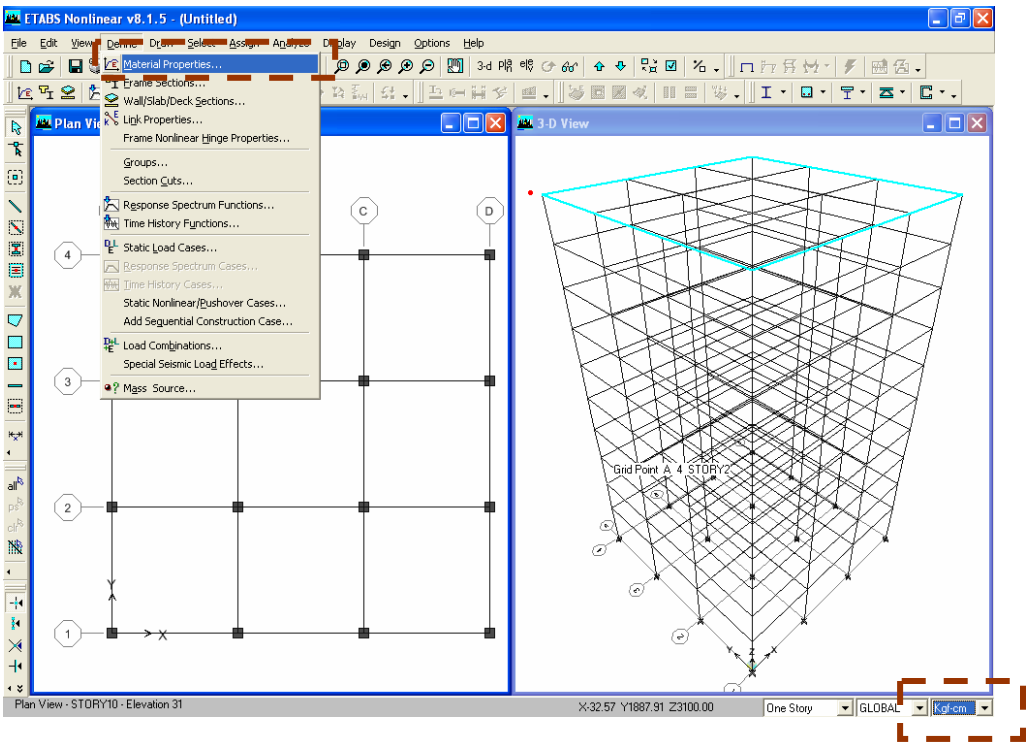
Two way slab model has been created as parameters specified from previous steps.



2. Define Material Properties

Step 2-1: Change Working Unit

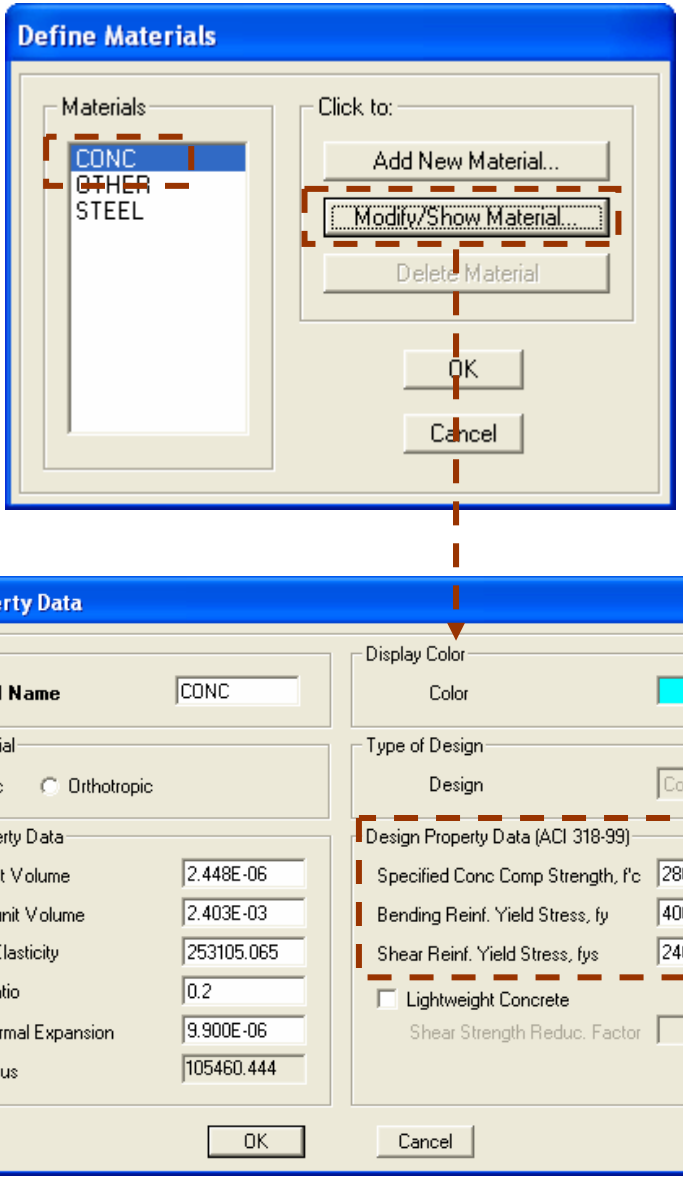
Change working unit to “kg-cm” and go to **Define >> Material**



Note: You may select “N-mm” or “Kip-in” or whatever unit to input material properties.

Step 2-2: Check Material Properties

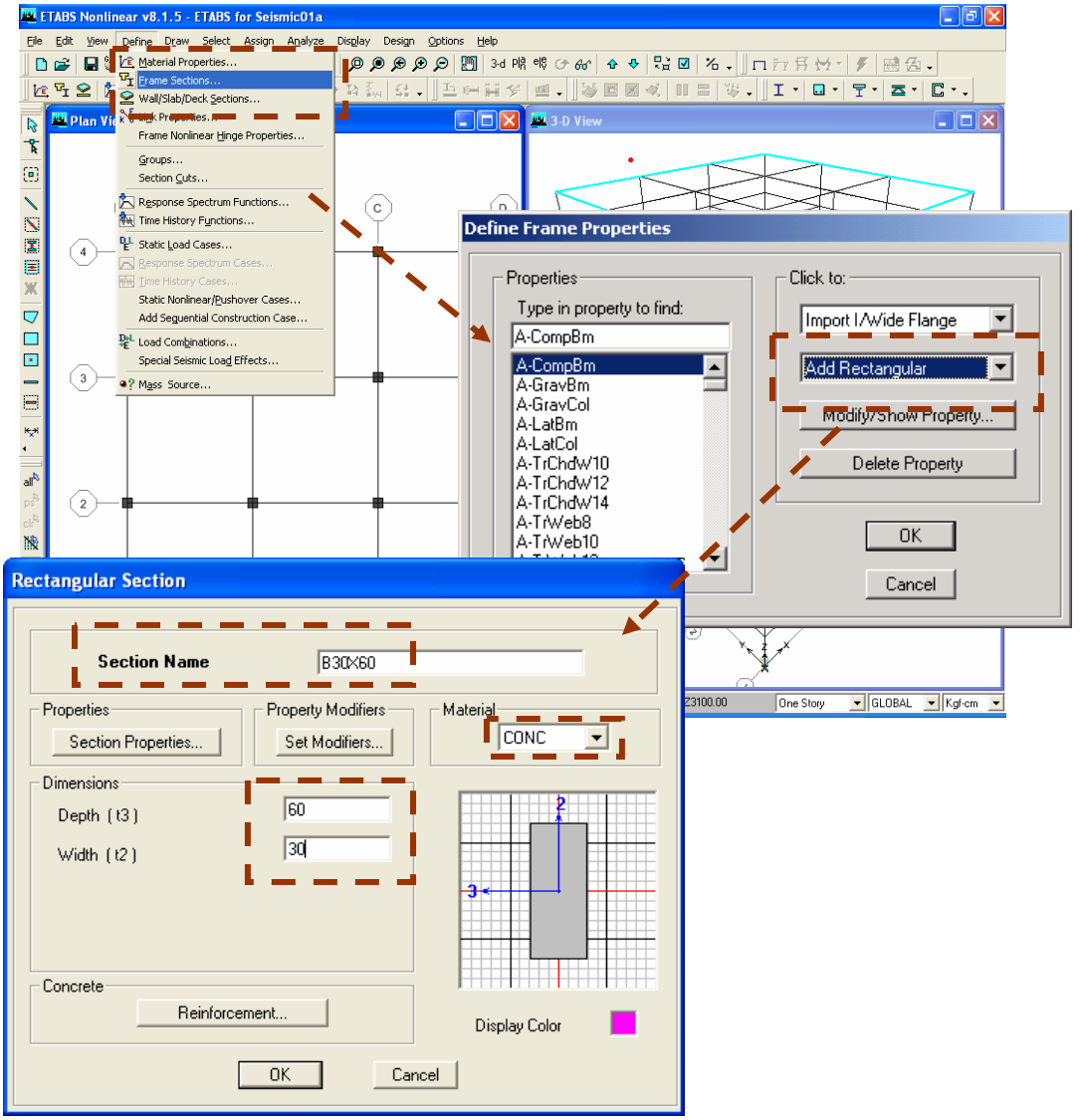
Select “CONC”, click on “Modify/Show Material..” button and specify material properties as shown in the figure below.



3. Define and Assign Section Properties

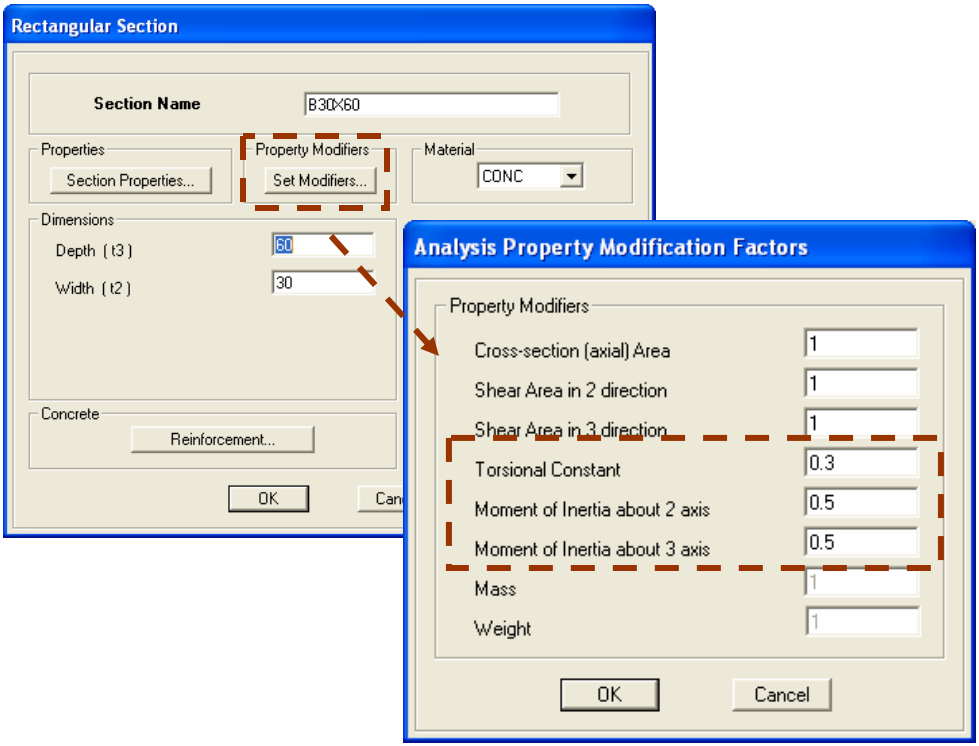
Step 3-1: Define New Frame Section and Specify Section Properties for Beam

Go to **Define >> Frame Sections** and select on "Add Rectangular" from second drop-down menu. Enter beam section properties as shown in figure below.



Step 3-2: Enter Property Modifiers

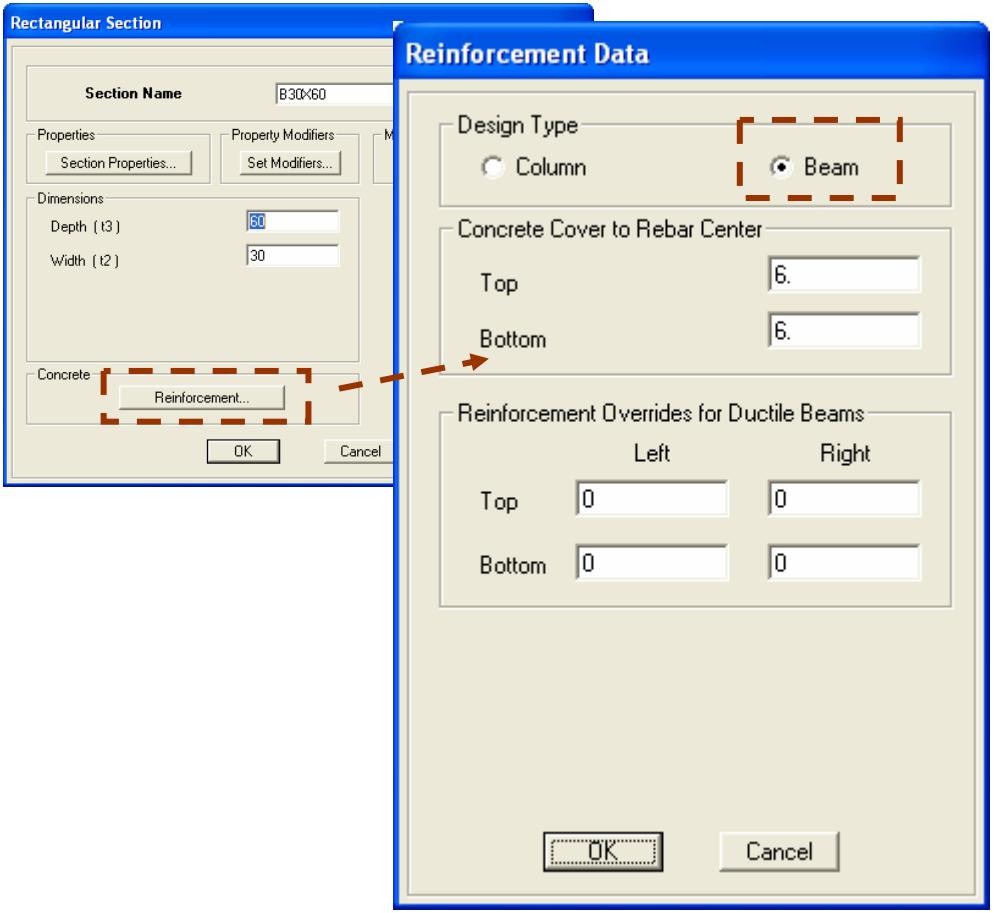
Click on “Set Modifiers” and enter property modifiers as shown in figure below



Note: Property modification factors are used to reduce moment and torsion stiffness due to crack section.

Step 3-3: Specify Reinforcement Data for Beam

Click on “Reinforcement” and specify reinforcement data as shown in the following figure.



Note for Reinforcing Information for Beam

For concrete beams there are two types of reinforcing information that you specify.

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.

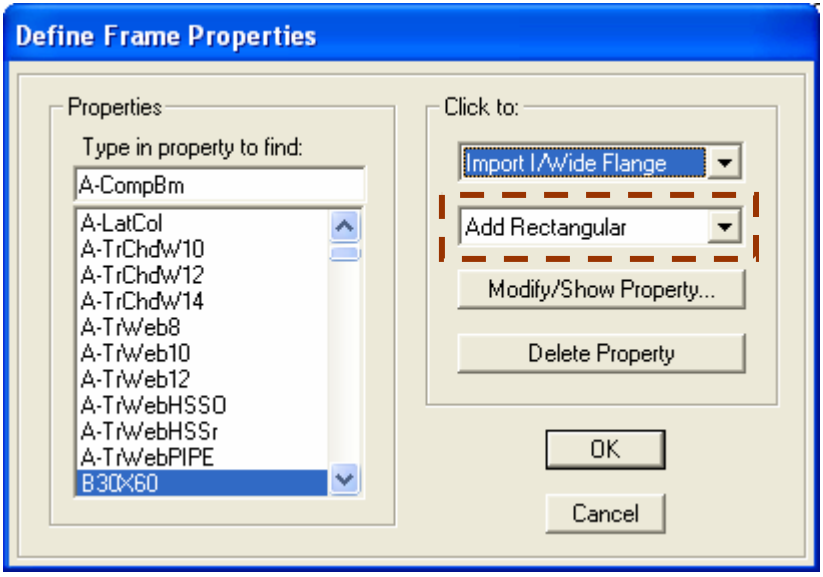
In the Concrete Frame Design postprocessor when the minimum reinforcing in the middle of a beam is to be based on some percentage of the reinforcing at the ends of the beam ETABS compares the calculated required reinforcement at the ends of the beam with that specified in the reinforcement overrides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

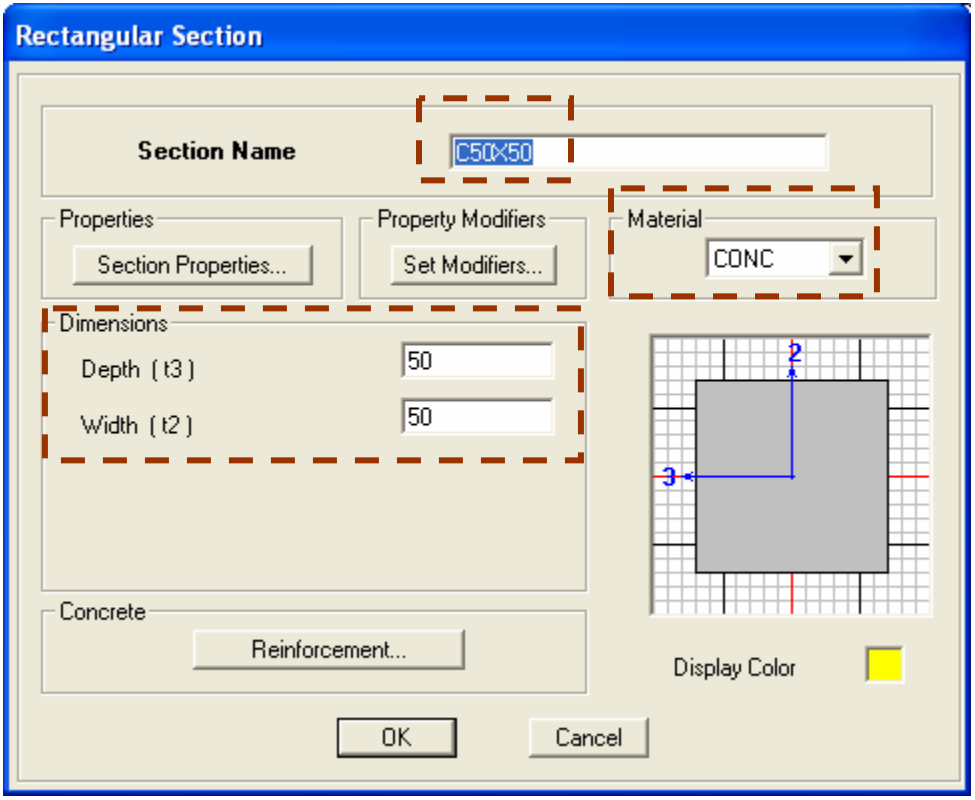
Step 3-4: Add Frame Section for Column

Select on “Add Rectangular” from second drop-down menu.



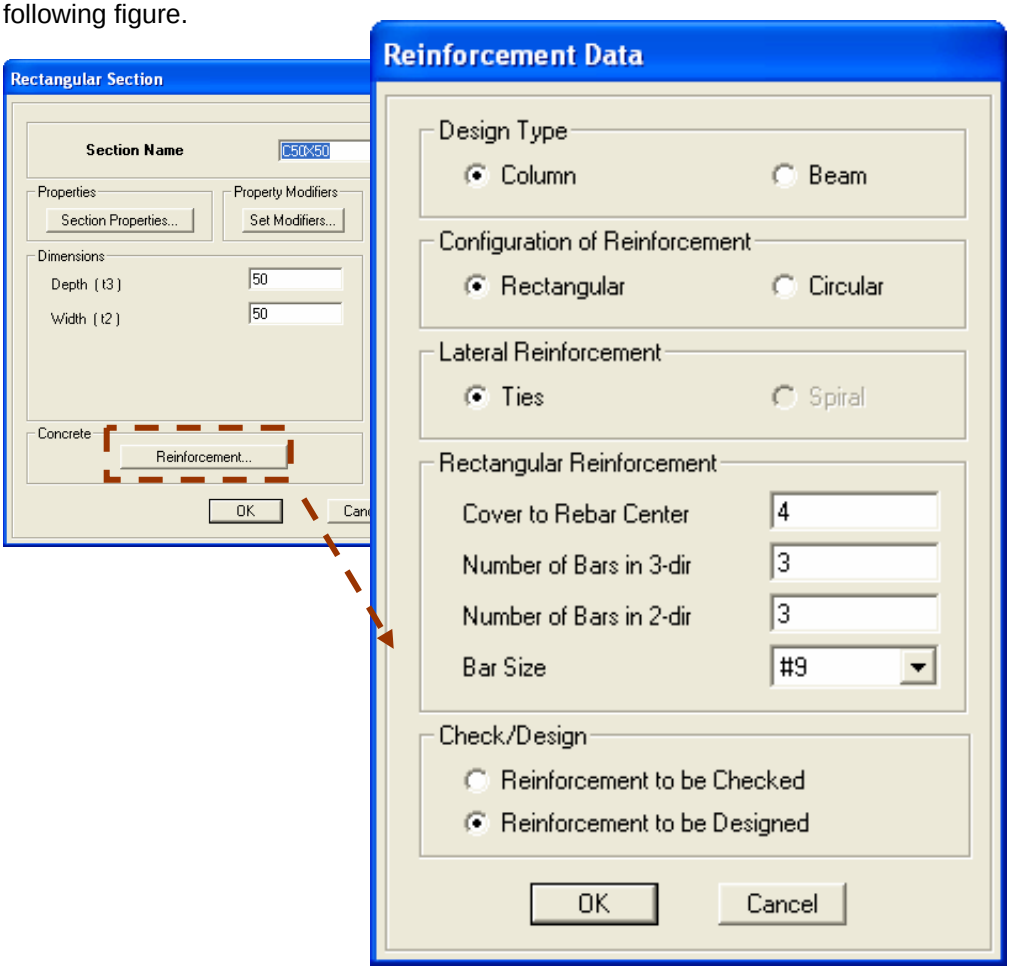
Step 3-5: Specify Column Section Properties

Specify column section properties as shown in the following figure.



Step 3-6: Specify Reinforcement Data for Column

Click on "Reinforcement" button and specify reinforcement data as shown in the following figure.



Note for 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.

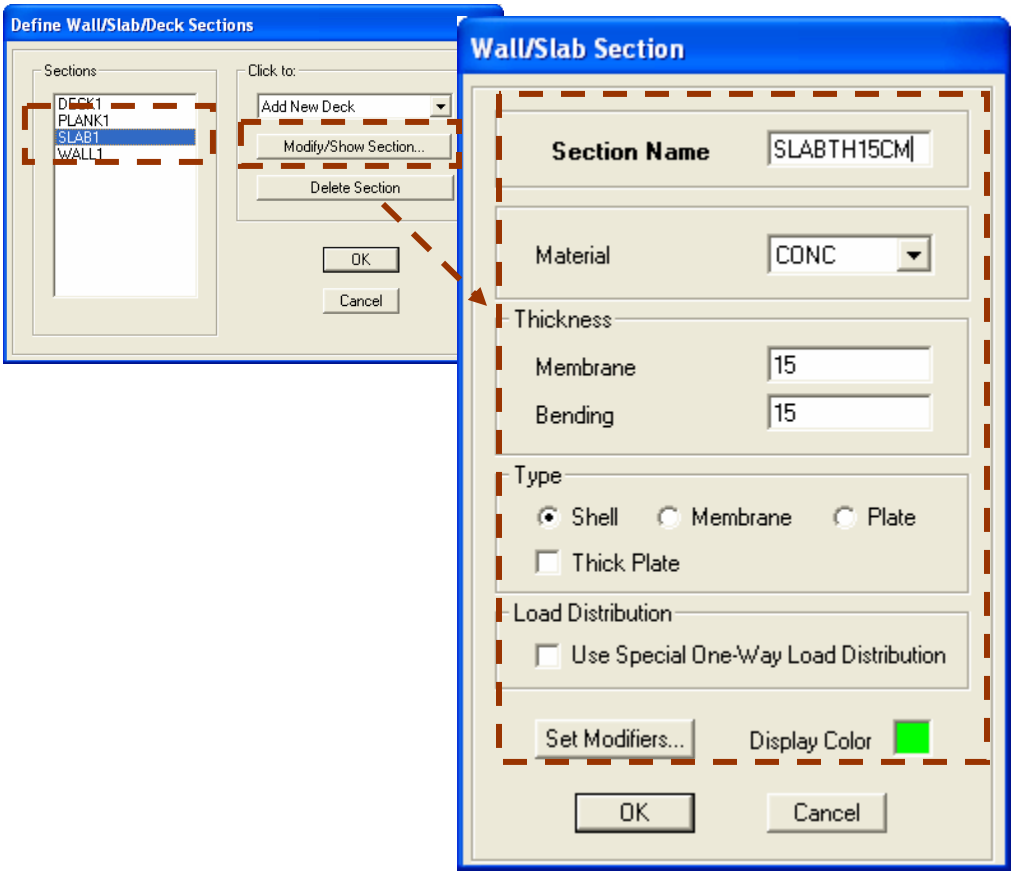
- **Cover to Rebar Center:** This is the distance from the edge of the column to the center of a longitudinal bar. In the special case of circular reinforcement in a rectangular column the cover is taken to be the minimum distance from the edge of the column to a circle drawn through the center of all 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 3-7: Define Slab Section Properties

Go to **Define >> Wall/Slab/Deck Sections**, select "SLAB1", click on "Modify/Show Section" and specify slab section properties as shown in figure below.



Note for Area Thickness

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:

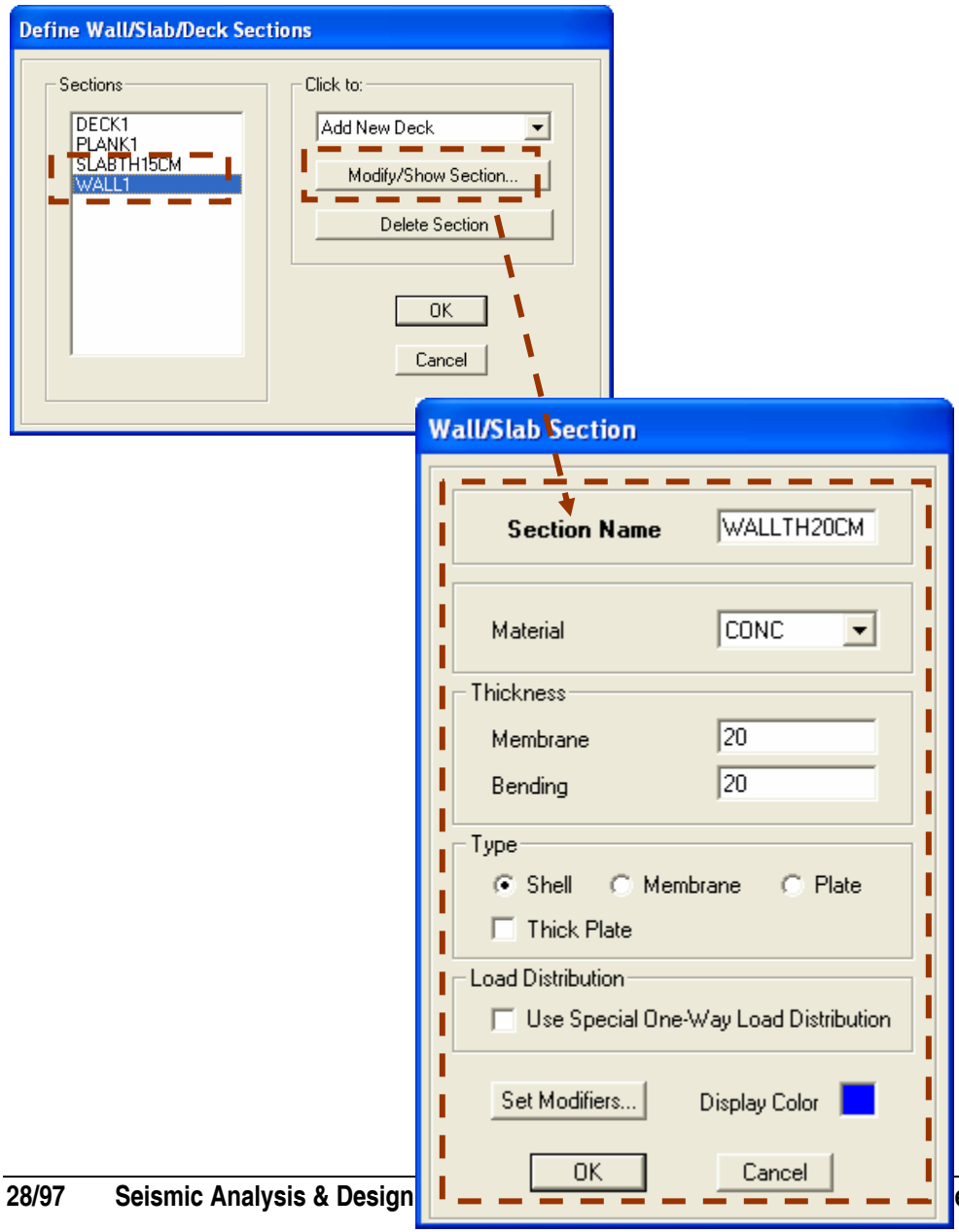
The membrane stiffness for full shell and pure membrane sections.

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.

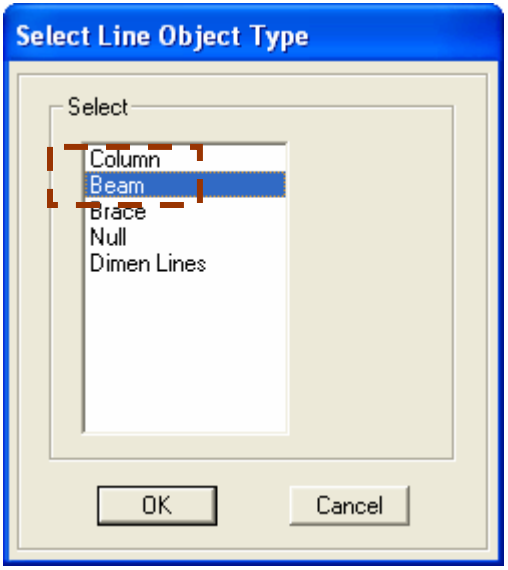
Step 3-8: Define Wall Section Properties

Select “WALL1”, click on “Modify/Show Section” and specify wall section properties as shown in figure below.

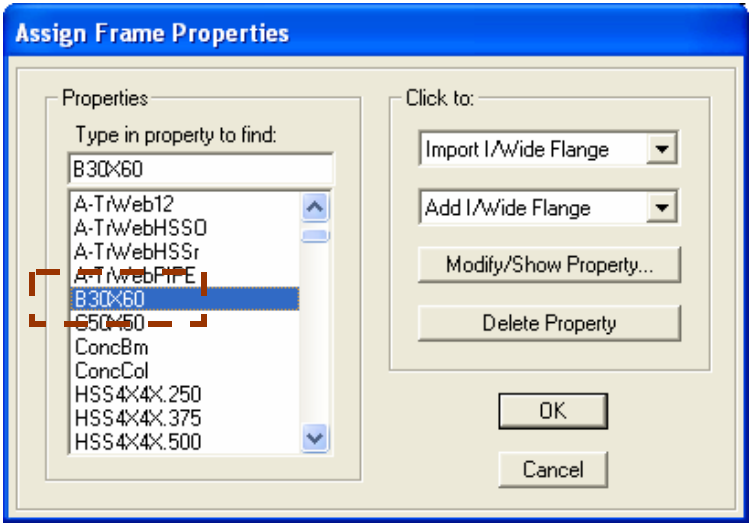


Step 3-9: Select All Beams and Assign "B30x60" Section Properties

Go to **Select >> by Line Object Type**, select "Beam" to select all beams in model.

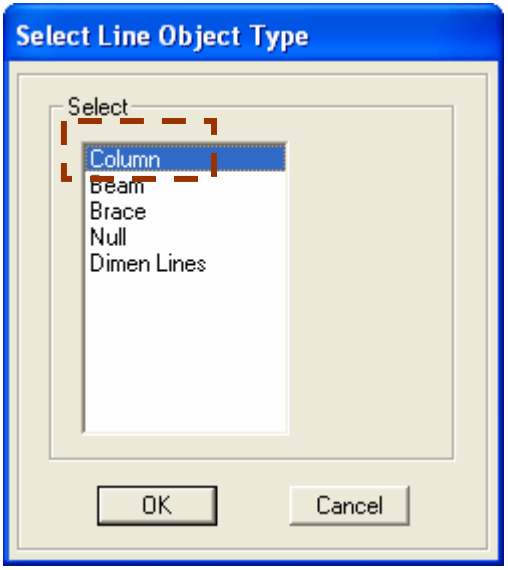


Go to **Assign >> Frame/Line >> Frame Section** and select "B30x60" from section property list

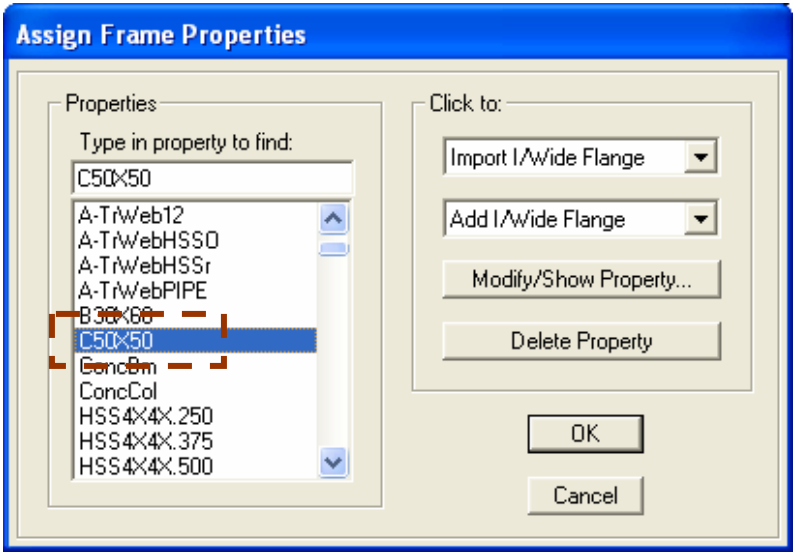


Step 3-10: Select All Columns and Assign "C50x50" Section Properties

Go to **Select >> by Line Object Type**, select "Column" to select all columns.

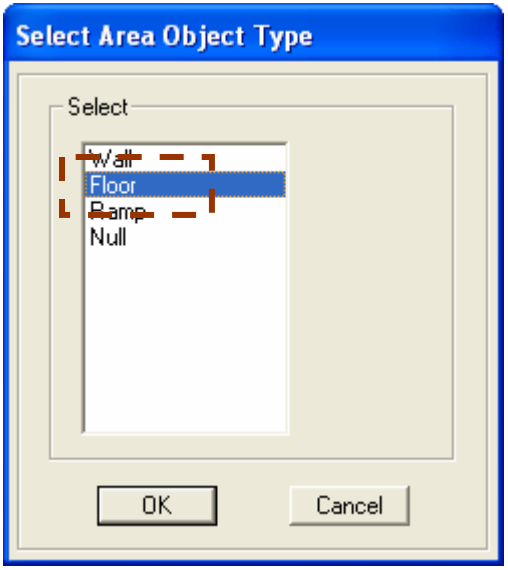


Go to **Assign >> Frame/Line >> Frame Section** and select "C50x50" from section property list

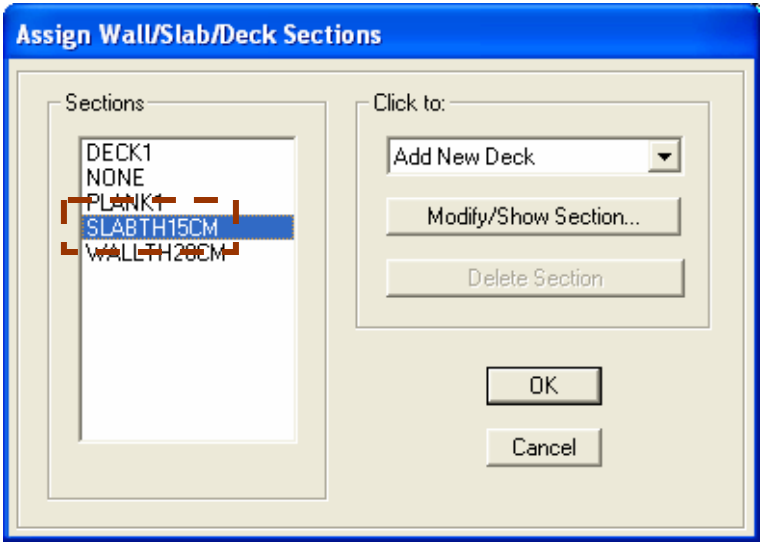


Step 3-11: Select All Slabs and Assign "SLABTH15CM" Section Properties

Go to **Select >> by Area Object Type**, select "Floor" to select all columns.



Go to **Assign >> Shell/Area >> Wall/Slab/Deck Section** and select "SLABTH15CM" from section property list

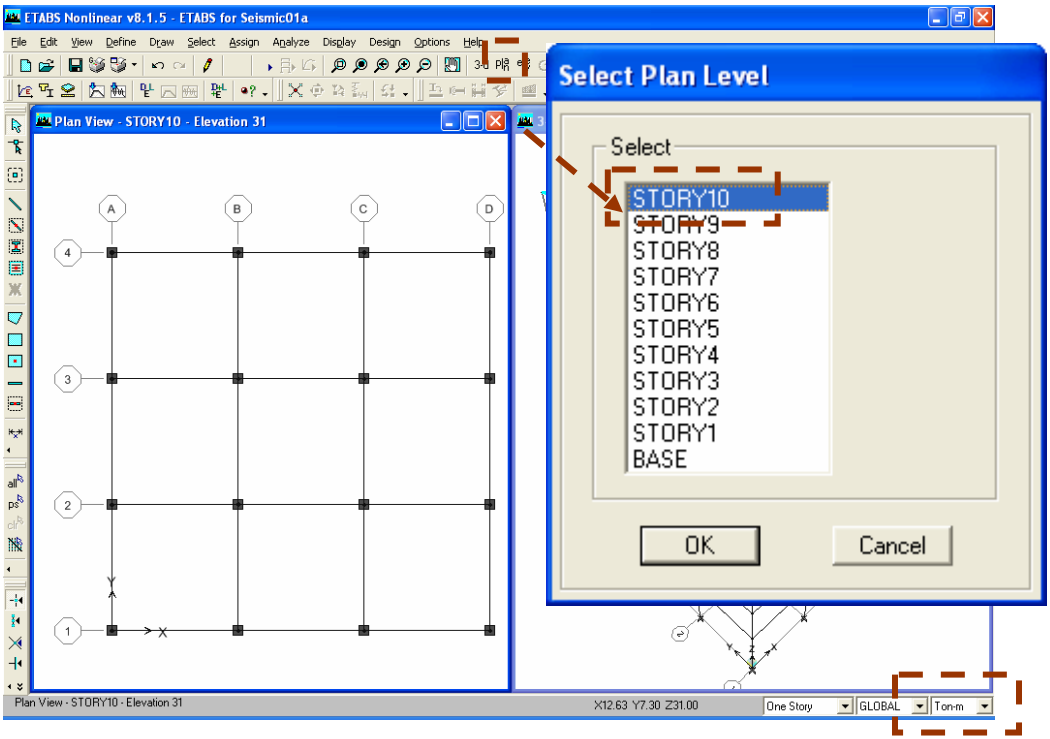


4. Draw Shear Wall and Define Pier Labels

Pier labels will define at shear wall panels in this step for shear wall design.

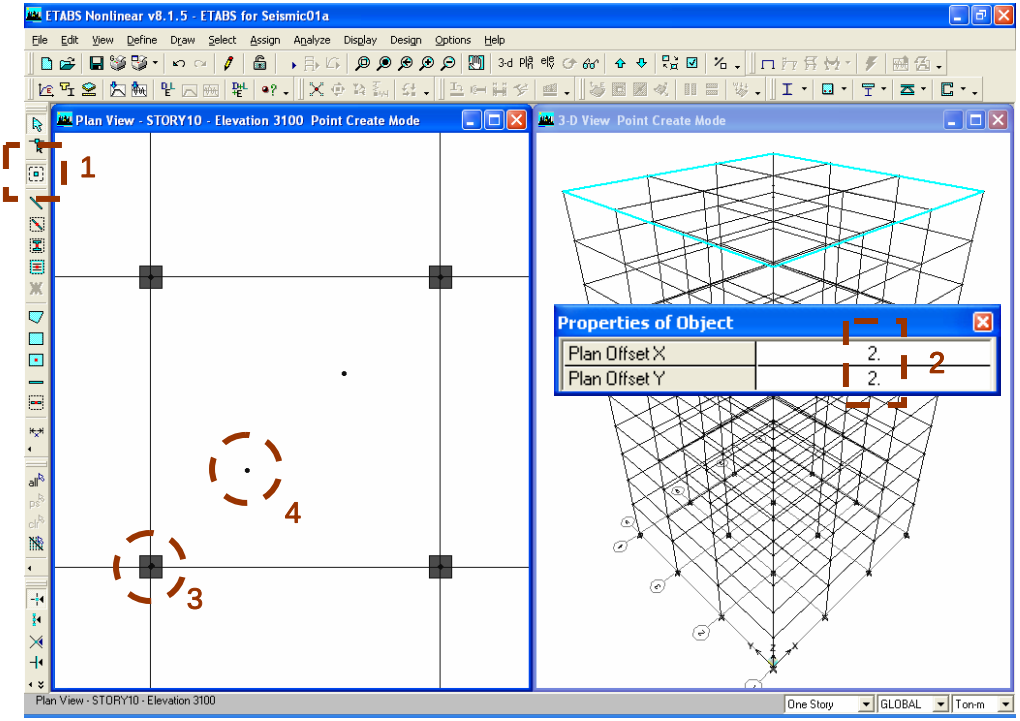
Step 4-1: Change View to Plan View and Change Working Unit to "Ton-m"

Activate left window by clicking on left window area, click on *Set Plan View* button and select "STORY10". Change working unit to "Ton-m"



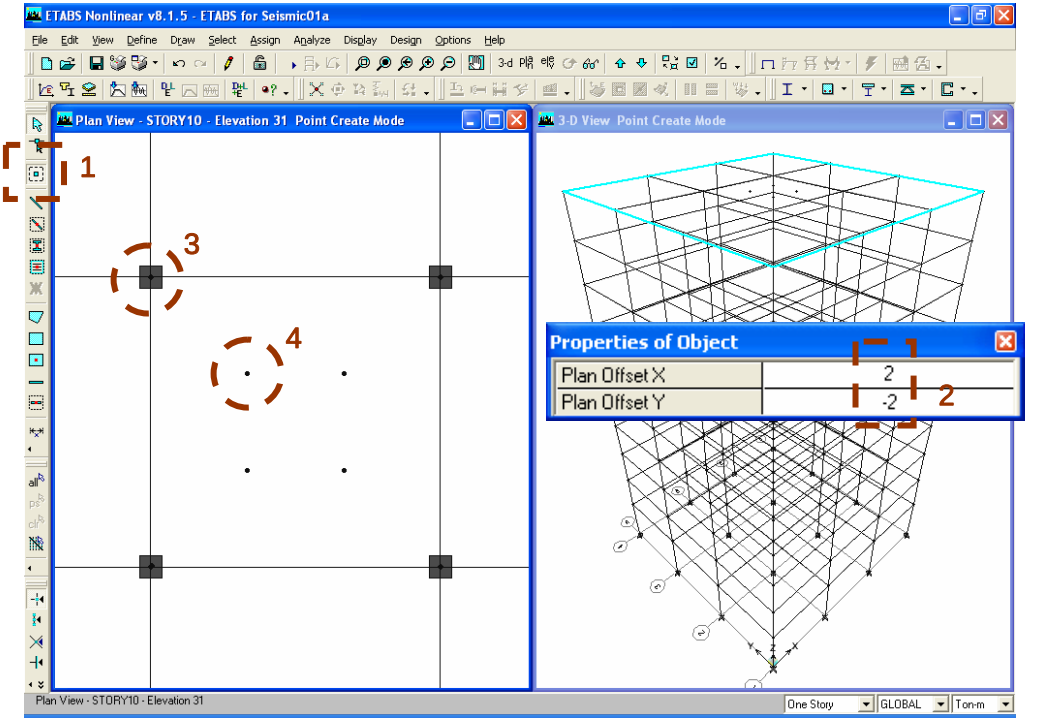
Step 4-2: Add Nodes at Shear Wall Corner Location

Click on *Rubber Band Zoom* button to zoom plan view at shear wall location, click on *Draw Point Objects* button, enter "Plan Offset X" and "Plan Offset Y" in "Properties of Object" dialogue and click 2 nodes as shown in figure below.



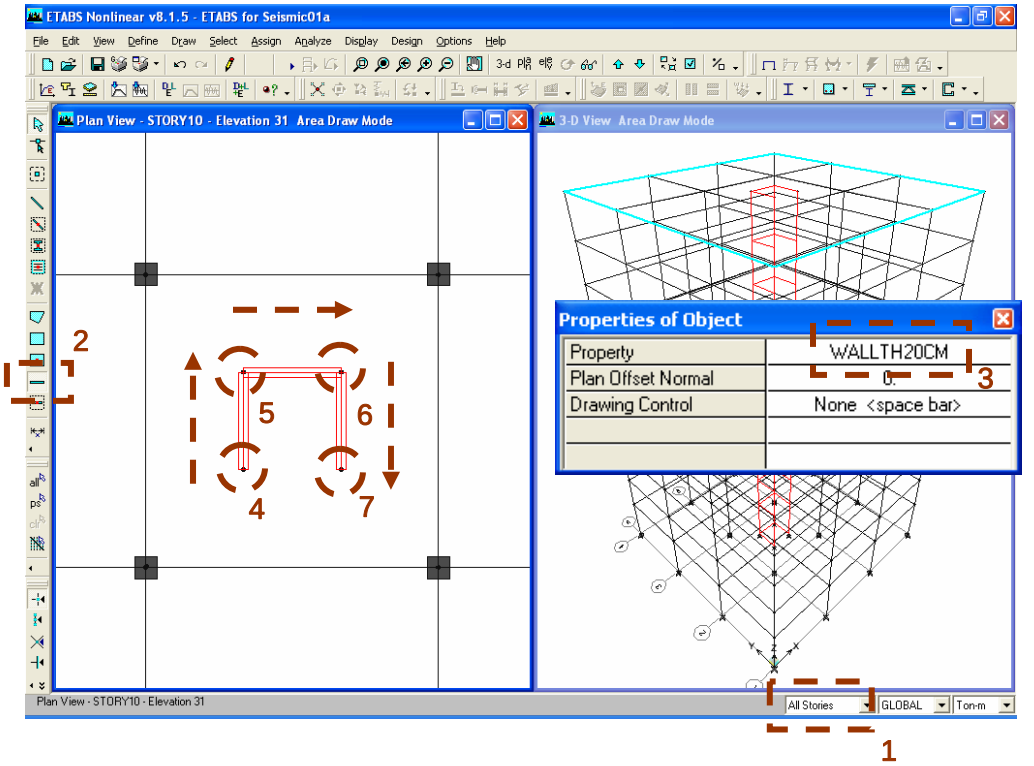
Step 4-3: Add 2 Nodes at Shear Wall Corner Location

Repeat Step 4-2 to add nodes at shear wall corner location as shown in figure below.



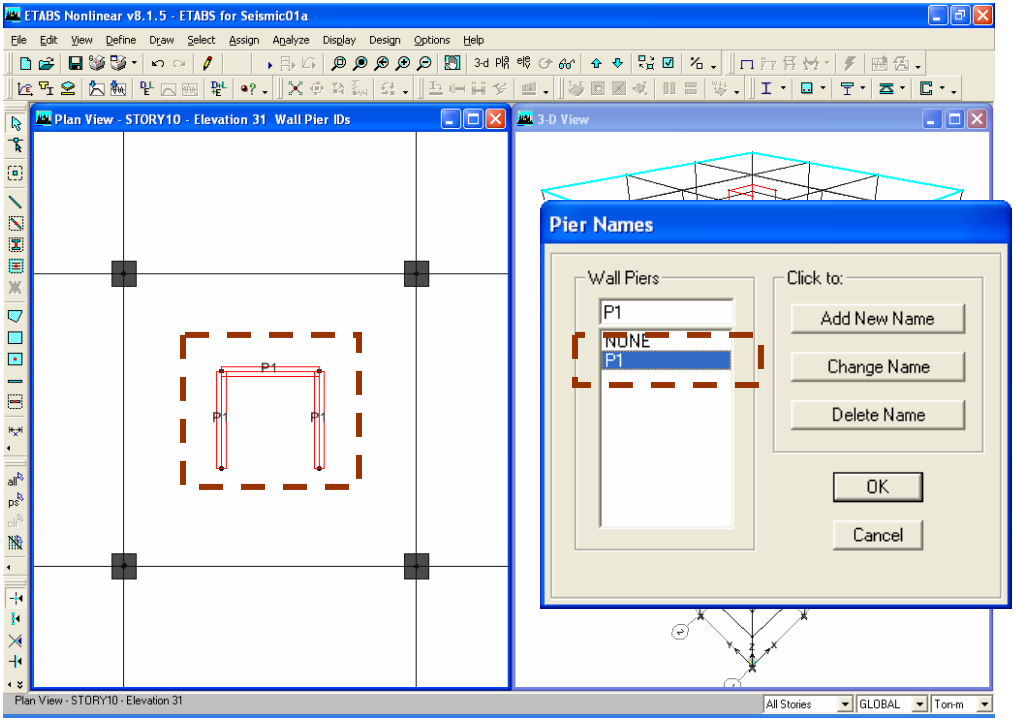
Step 4-4: Select "All Stories" and Draw Shear Walls

Select "All Stories" at first drop-down menu, click on *Draw Wall* button and draw shear walls as shown in figure below.



Step 4-5: Assign Pier Label for Shear Wall Design

Click on *Select Object* button  to change to selecting mode, select all shear wall panels by drawing rectangular cover all shear wall panels, go to **Assign >> Shell/Area >> Pier Label** and select "P1".

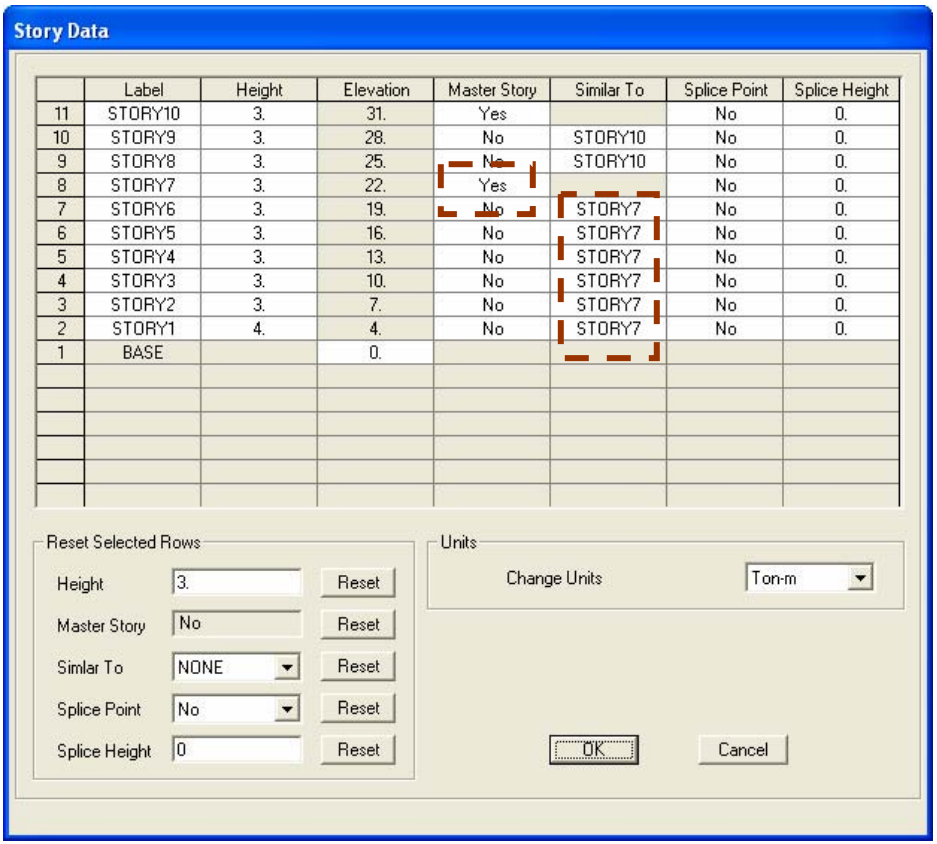


Note: For this example, all shear wall panels have been assigned in same pier labels then ETABS will design all shear wall panels as 3D shear wall (3 panels combined together). Each shear wall panel can be designed separately as 2D shear wall by assigning difference pier labels.

5. Define "Similar Stories" Option

Step 5-1: Define Master Story

Go to **Edit >> Edit Story Data >> Edit Story**, change "Master Story" at "STORY7" from "No" to "Yes" and change "Similar To" at "STORY1" to "STORY6" from "STORY10" to "STROY7" as shown in figure below.



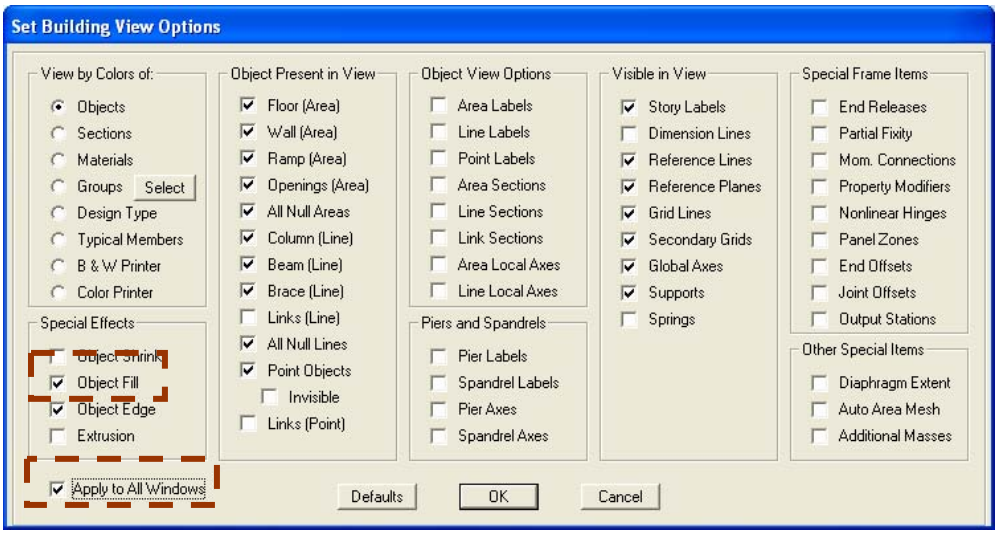
Note: "Similar Stories" option in ETABS help user to do duplicate work at typical stories, when "Similar Stories" is activated, all assignments on plan view at any stories in similar stories group will affect to every similar story.

6. Modify Floor Plan at "STORY8" to "STORY10"

To delete all elements in corner of building at "STORY8" to "STORY10", slab panel at each floor will be divided manually from one big panel to 9 panels at frame location.

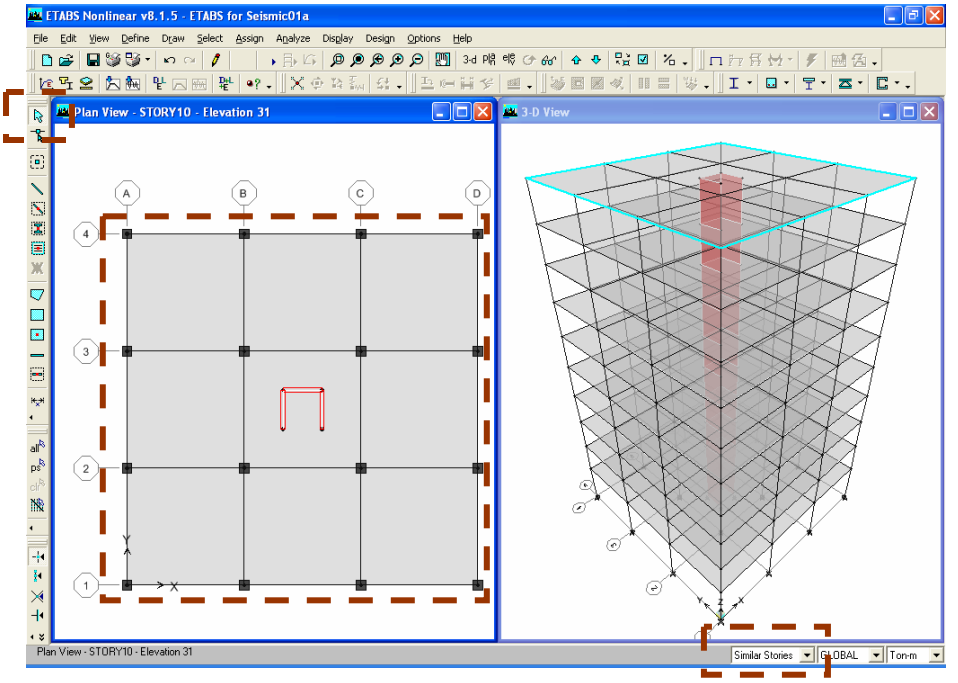
Step 6-1: Show Shell Panel in Solid Shade

Click on *Restore Full View* button to view full area of plan, click on *Set Building View Options* button and select "Object Fill" and ""Apply to All Windows"

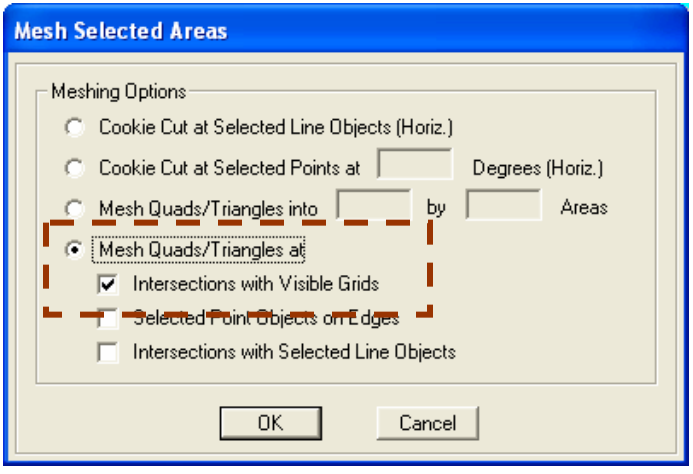


Step 6-2: Select All Slab Panels at "STORY10"

Make sure that current "Plan View" window is at "STORY10", select "Similar Stories", click on *Select Object* button and draw rectangular to cover all slab panels in plan

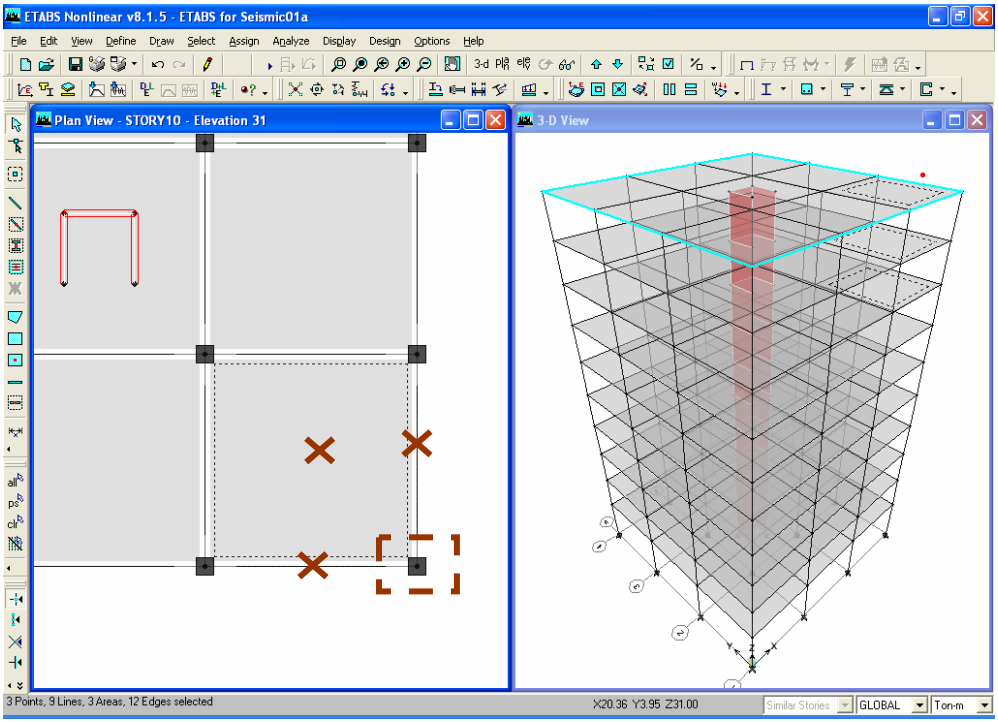


Go to **Edit >> Mesh Area** and select parameters as shown in figure below



Step 6-3: Delete All Elements at Bottom-right Corner of Building

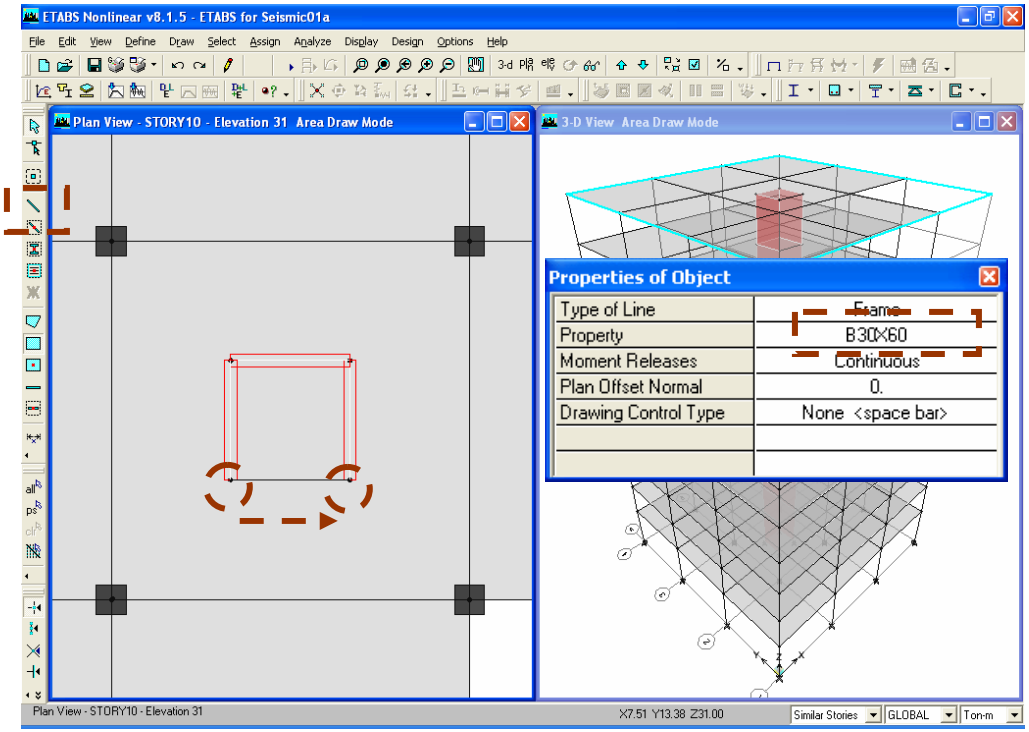
Select beams and slab panel by clicking on them, select columns by drawing rectangular to cover column in plan as shown in figure below and click on “Delete” button in keyboard



Note: Click on *Object Shrink Toggle* button  to see connectivity of each element

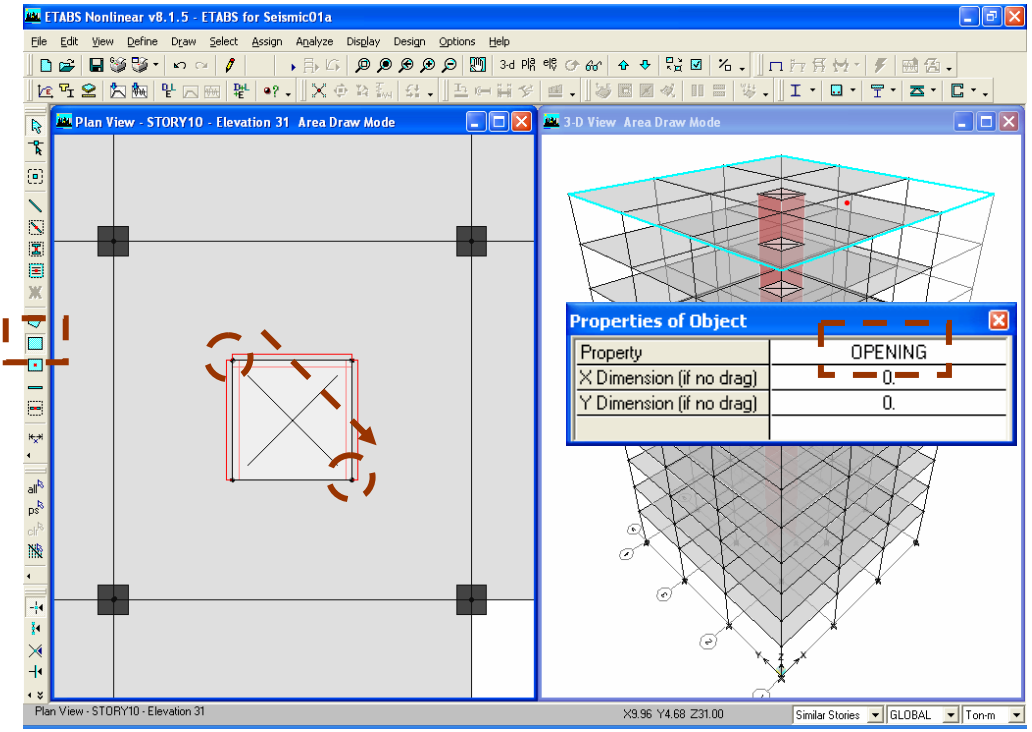
Step 6-4: Draw Beam at Front of Elevator

Use *Restore Full View* button and *Rubber Band Zoom* button to change plan view to elevator location, click on *Create Lines or at Clicks* button , select "B30x60" and draw beam at front of elevator



Step 6-5: Draw Opening at Elevator Area

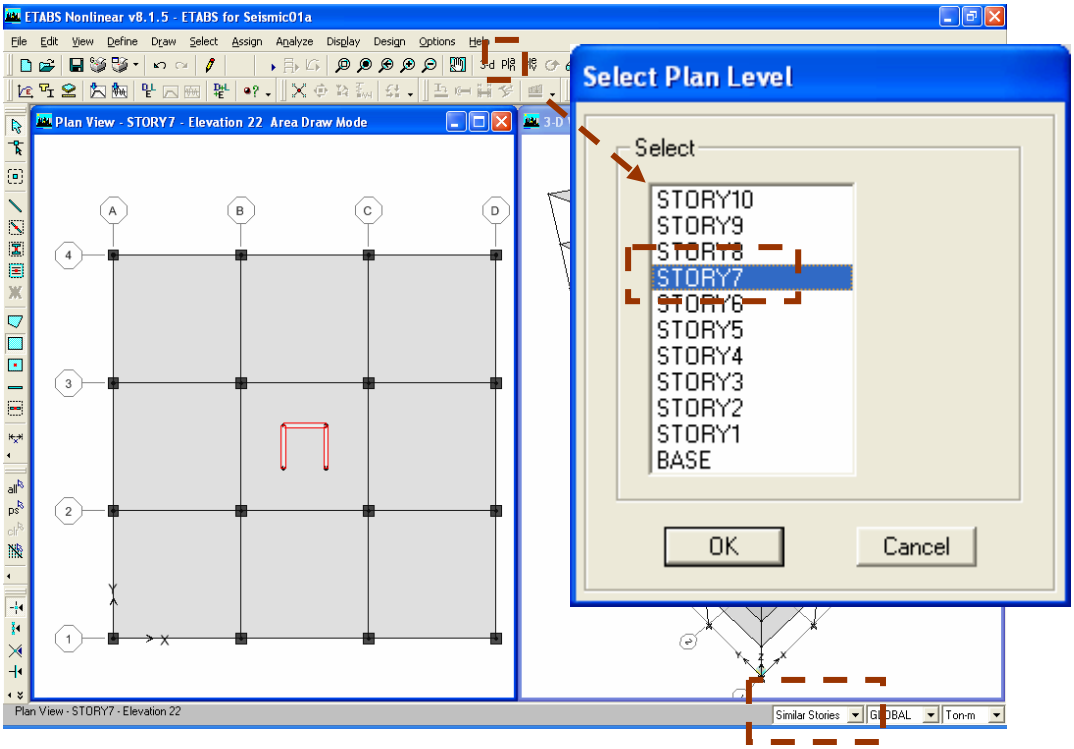
Click on *Draw Rectangular Areas* button , select “OPENING” and draw opening at elevator area.



7. Modify Floor Plan at “BASE” to “STORY7”

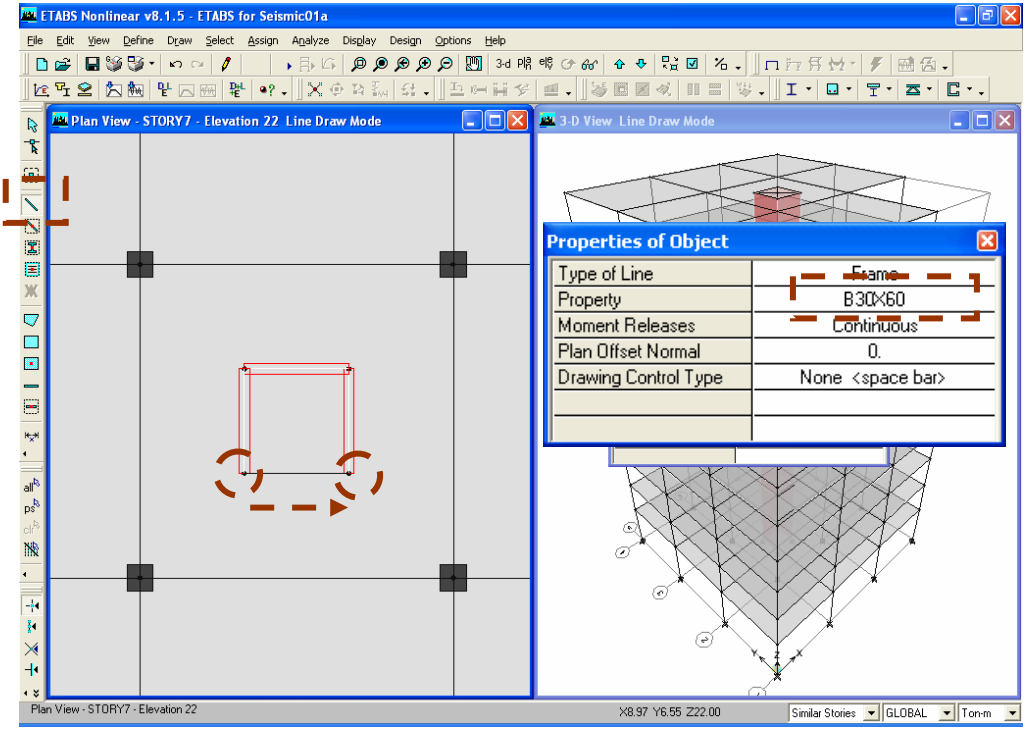
Step 7-1: Change View to “STORY7”

Click on *Set Plan View* button and select “STORY7” and make sure that “Similar Stories” is selected.



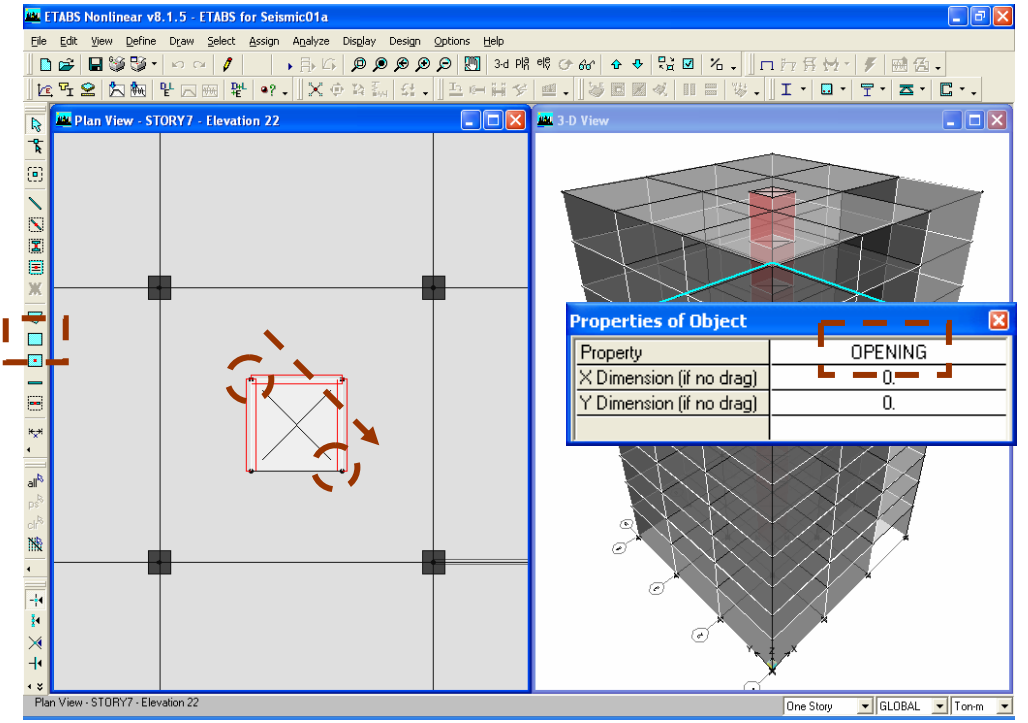
Step 7-2: Draw Beam at Front of Elevator

Use *Restore Full View* button and *Rubber Band Zoom* button to change plan view to elevator location, click on *Create Lines or at Clicks* button , select "B30x60" and draw beam at front of elevator



Step 7-3: Draw Opening at Elevator Area

Click on *Draw Rectangular Areas* button , select “OPENING” and draw opening at elevator area.

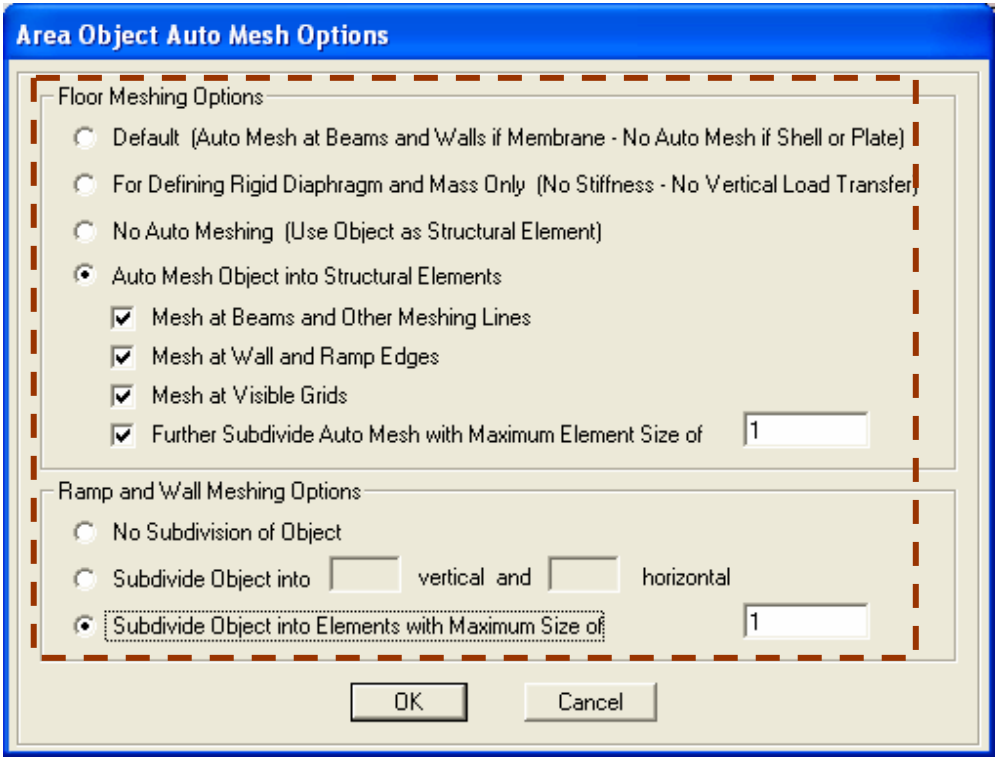


8. Assign Auto Mesh Options at Shell Panels

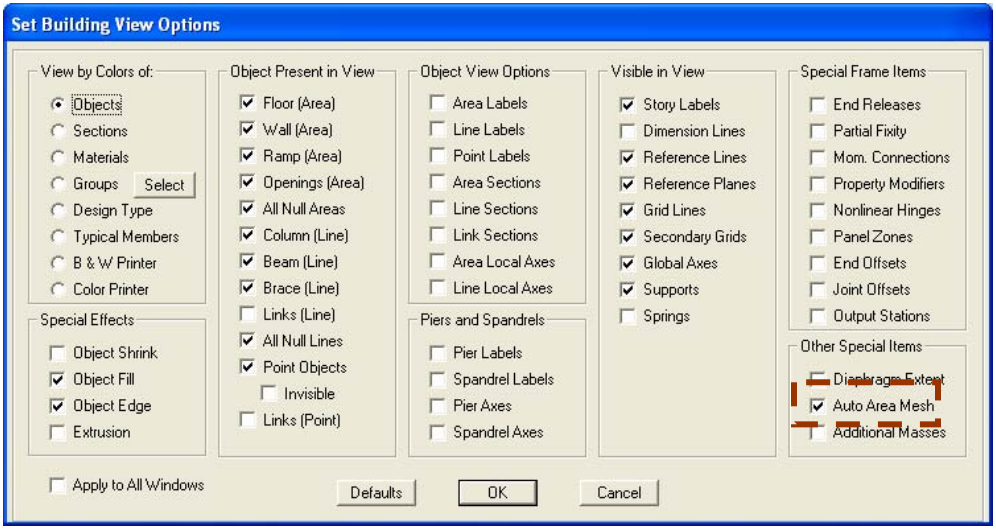
Each shell panel (slab and shear wall) will be divided into small panels by using Auto Meshing Option in ETABS. Maximum size of small panel is not bigger than 1 m.

Step 8-1: Use Auto Mesh on Slab and Wall Panels

Select all elements in building by clicking on *Select All* button  , go to **Assign >> Shell/Area >> Area Object Mesh Options** and specify parameters as shown in figure below



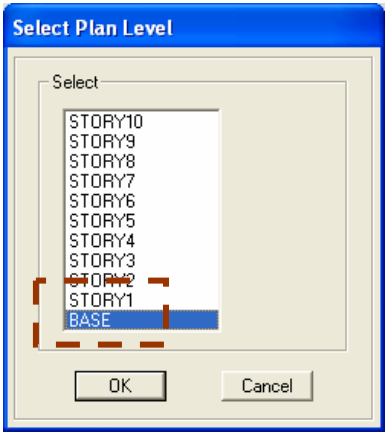
Note: Auto mesh side and location can be display by clicking on *Set Building View Options* button and selecting "Auto Area Mesh" to view auto area meshing line.



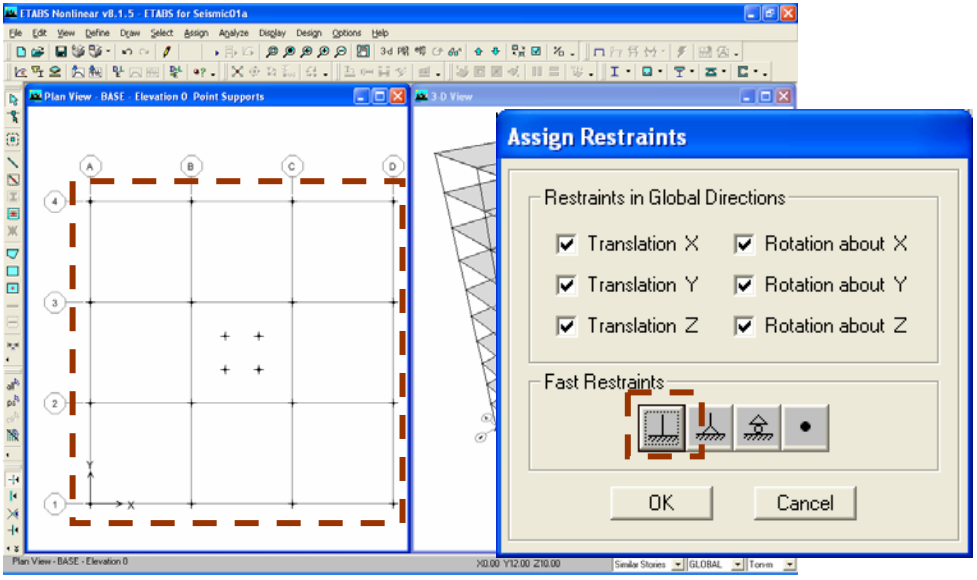
9. Assign Supports

Step 9-1: Change Plan View to "BASE" and Select All Nodes on "BASE" Floor

Click on *Set Plan View* button and select "BASE"



Click on *Select Object* button and draw selection rectangular to cover all nodes, go to **Assign >> Joint/Point >> Restraints** and select "Fix Support" .



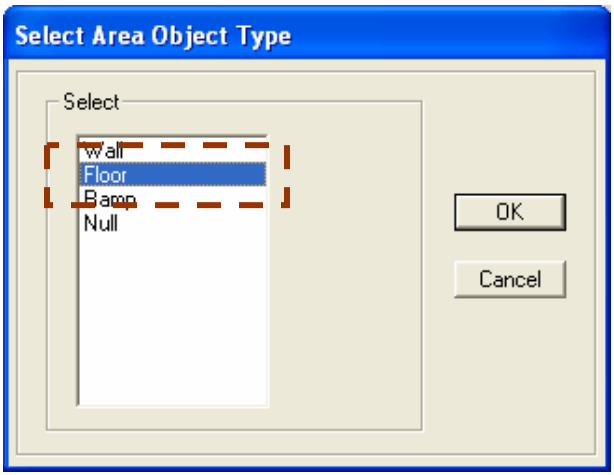
Note: This example focuses on simplify analysis. Spring support will be demonstrated in some other example.

10. Assign "DEAD" and "LIVE" Load

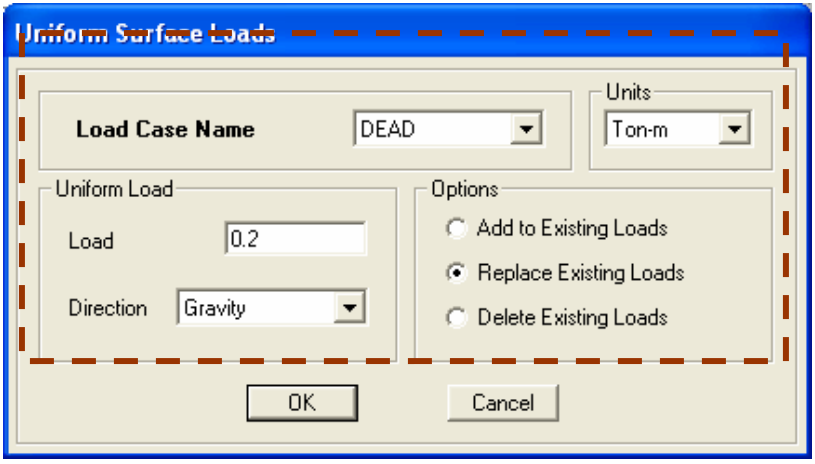
Uniform load on slab panels for "DEAD" and "LIVE" load have been assigned from Step 1-2 then step 10-1 and 10-4 can be skipped.

Step 10-1: Assign "DEAD" Uniform Load on Slab Panels

Go to **Select >> By Area Object Type** and select "Floor"



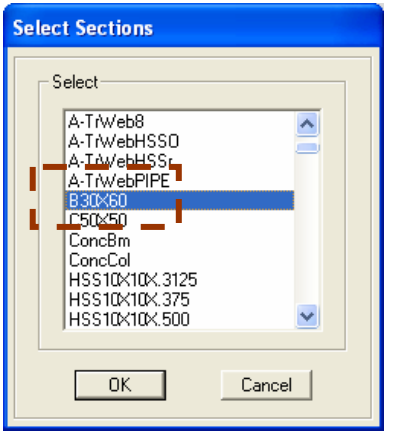
Go to **Assign >> shell/Area Loads >> Uniform** and specify parameter as shown in figure below.



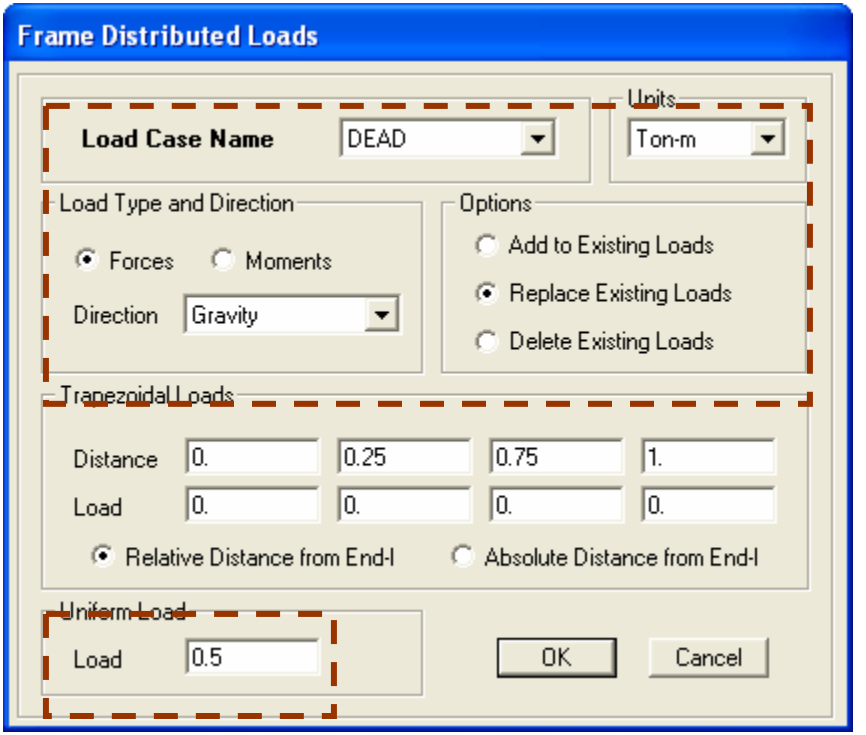
Note: The load from slab will be transferred to beam and column automatically (for one way, two way, flat slab)

Step 10-2: Assign "DEAD" Uniform Load on Beams

Go to **Select >> By Frame Sections** and select "B30x60".

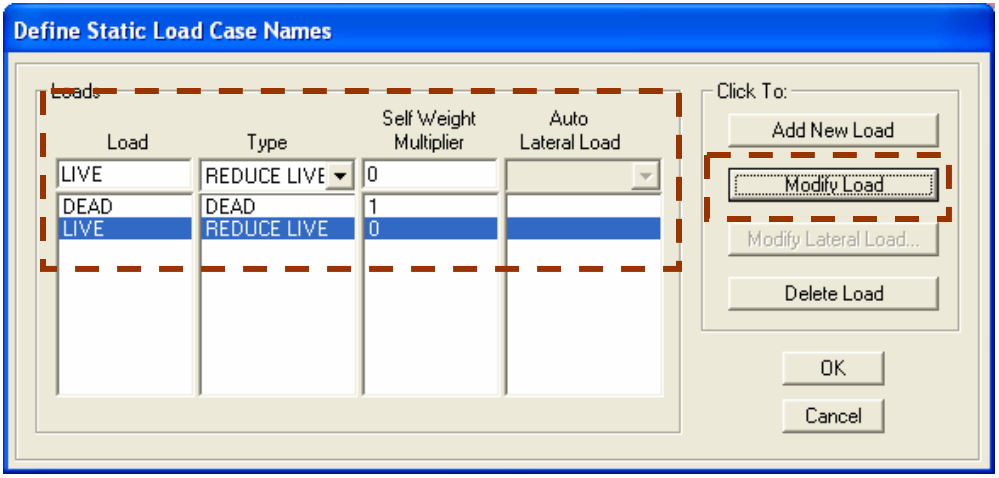


Go to **Assign >> Frame/Line Loads >> Distributed** and specify parameter as shown in figure below.

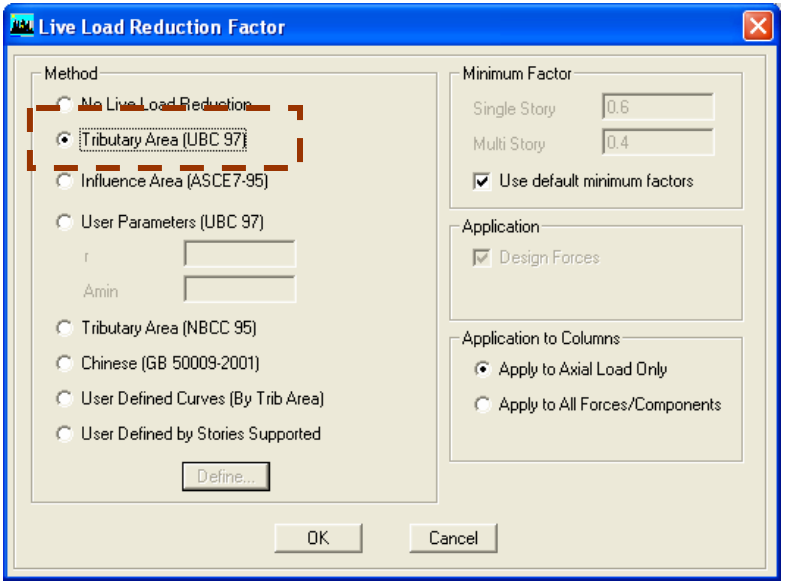


Step 10-3: Change Load Type for "LIVE" Load Case

Go to **Define >> Static Load Case**, select "LIVE" load case, change "Type" from "LIVE" to "REDUCE LIVE" and click on "Modify Load"



Go to **Options >> Preferences >> Live Load Reduction** and select "Tributary Area (UBC97)" as shown in figure below.



Note:

If this check box is checked, the tributary area live load reduction method based on Section 1607.5 of the 1997 UBC is used. The basic formula is as follows:

$$\mathbf{RLLF = 1 - 0.0008(A - 150)}$$

where,

RLLF = The reduced live load factor for an element, unitless. The RLLF is multiplied times the unreduced live load to get the reduced live load.

A = Tributary area for the element, ft^2. If A does not exceed 150 ft^2, no live load reduction is used. See Tributary Area for more information.

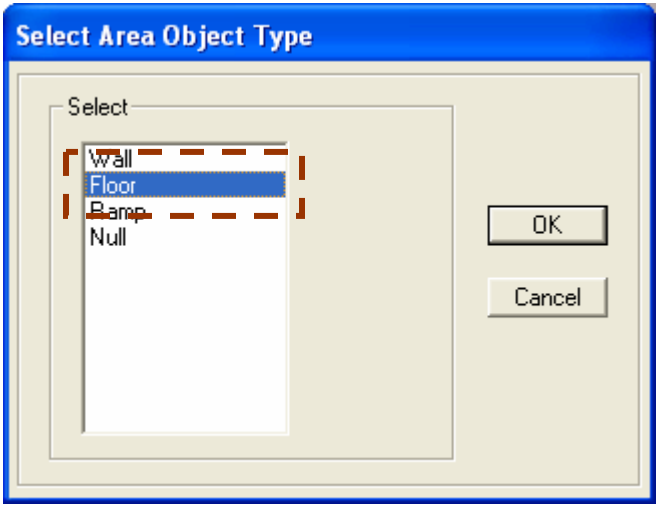
The RLLF factor can not be less than the minimum factor described in the Minimum Factor Area description.

Note that no check is done to limit the RLLF based on Equation 7-2 in Section 1607.5 of the 1997 UBC.

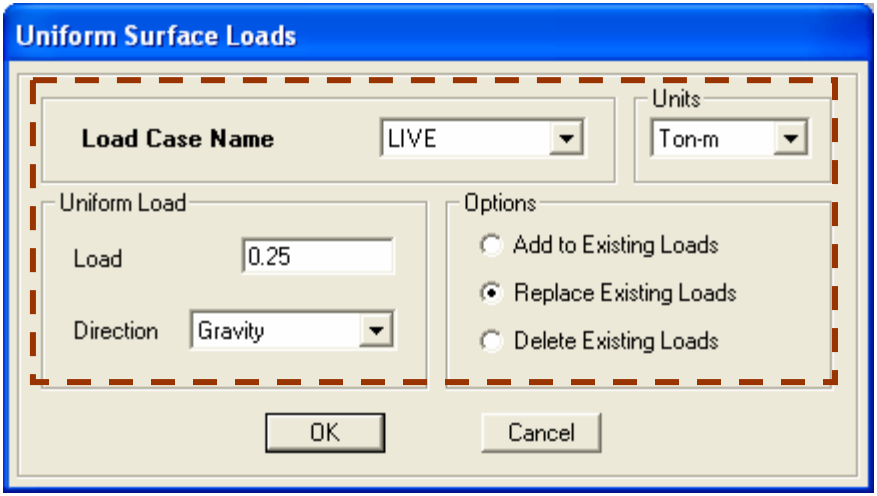
You may press “F1” key to get more information about RLLF when you are on the ”Live Load Reduction Factor” dialogue.

Step 10-4: Assign "LIVE" Uniform Load on Slab Panels

Go to **Select >> By Area Object Type** and select "Floor"



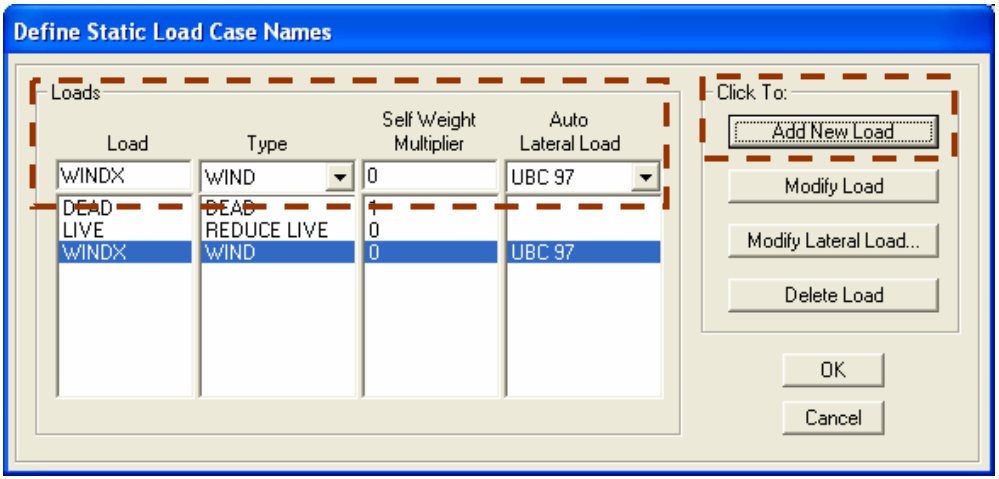
Go to **Assign >> shell/Area Loads >> Uniform** and specify parameter as shown in figure below.



11. Define and Assign Wind Load Case

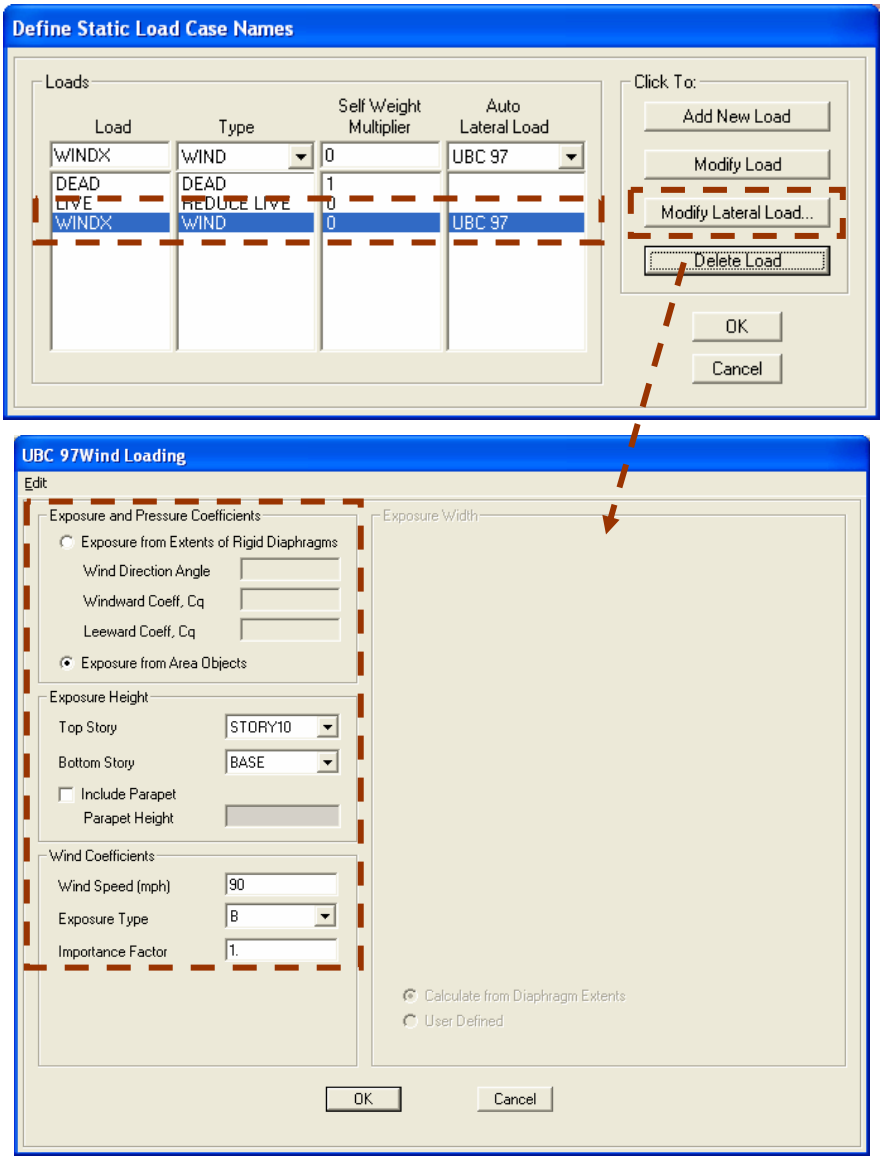
Step 11-1: Add "WINDX" Load Case

Go to **Define >> Static Load Case** and enter load case parameters for "WINDX" as shown in the figure below.



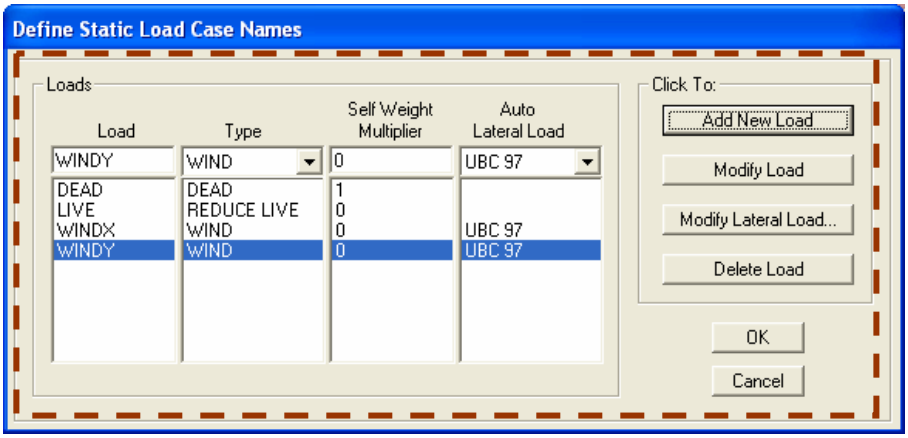
Step 11-2: Specify "WINDX" Load Case

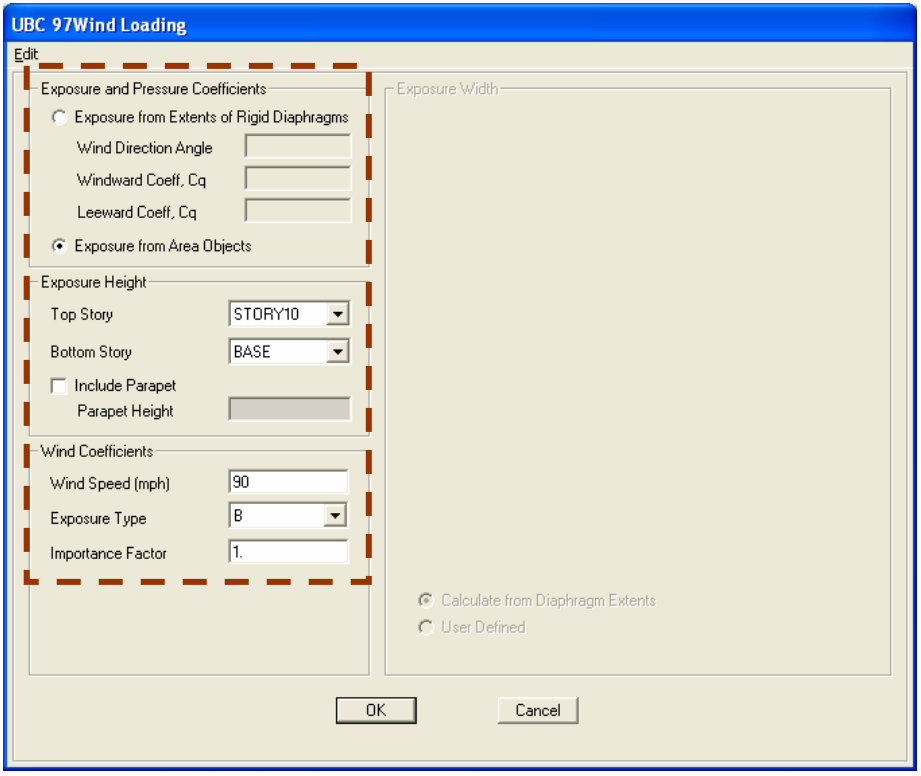
Select "WINDX" from list, click on "Modify Lateral Load" and specify parameters as shown in figure below.



Step 11-3: Add and Specify "WINDY" Load Case

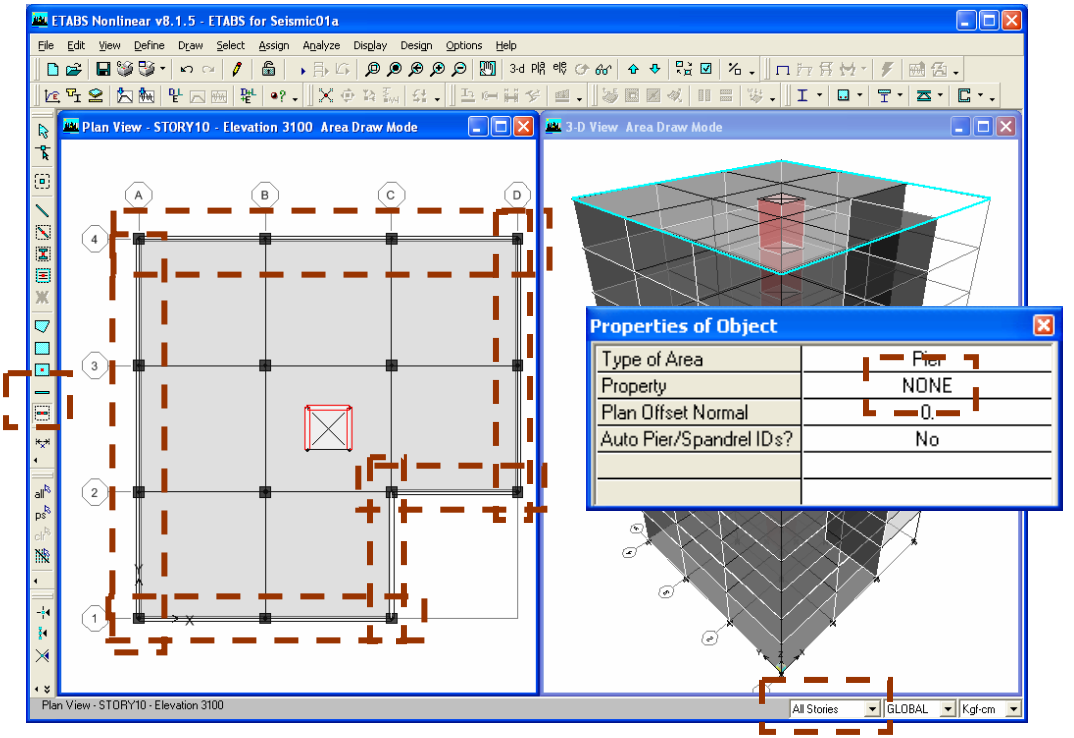
Repeat Step 11-1 to 11-2 to add "WINDY" Load Case





Step 11-4: Draw Null Areas at Side of Building for Wind Pressure Coefficient

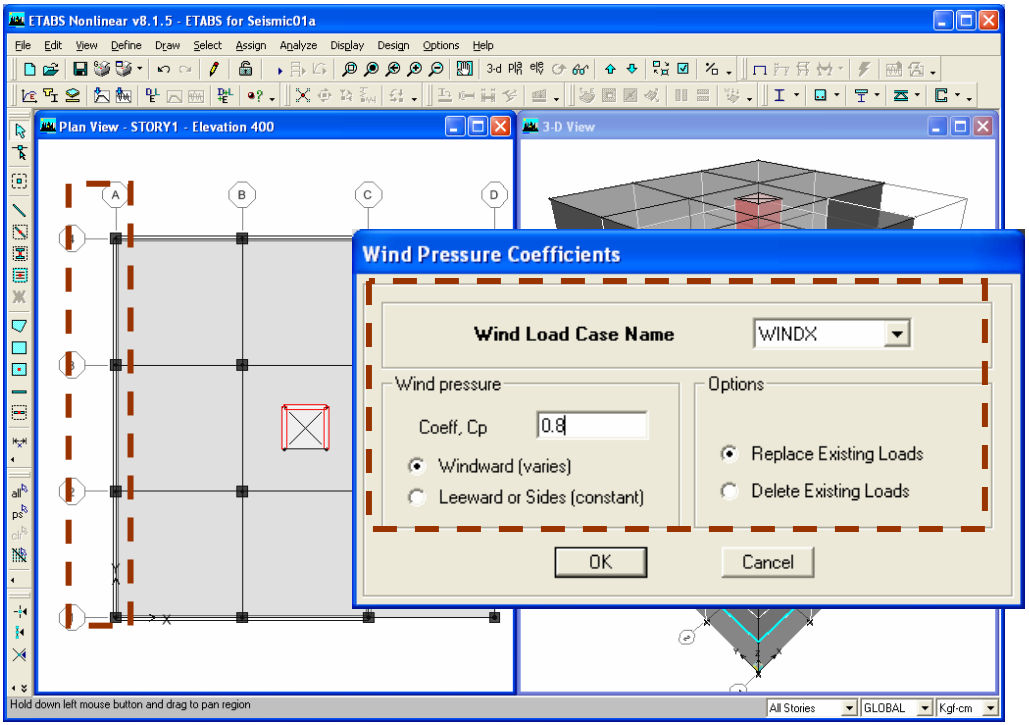
Click on *Set Plan View* button and select "STORY10" to change Plan view to "STORY10". From "STORY10" plan view, select "All Stories" from first drop-down menu, click on *Create Walls at Regions or at Clicks* button , select "NONE" and draw rectangular cover all side of plan view one by one as shown in figure below.



Note: Dummy Area (Null Area) is shell element with no stiffness to represent curtain wall or brick wall for wind pressure coefficient assignment. ETABS calculates wind load by using area of dummy area and wind pressure coefficients automatically based on selected code at step 10-1 and 10-2

Step 11-5: Assign Windward Wind Coefficient to Null Areas for "WINDX" Load Case

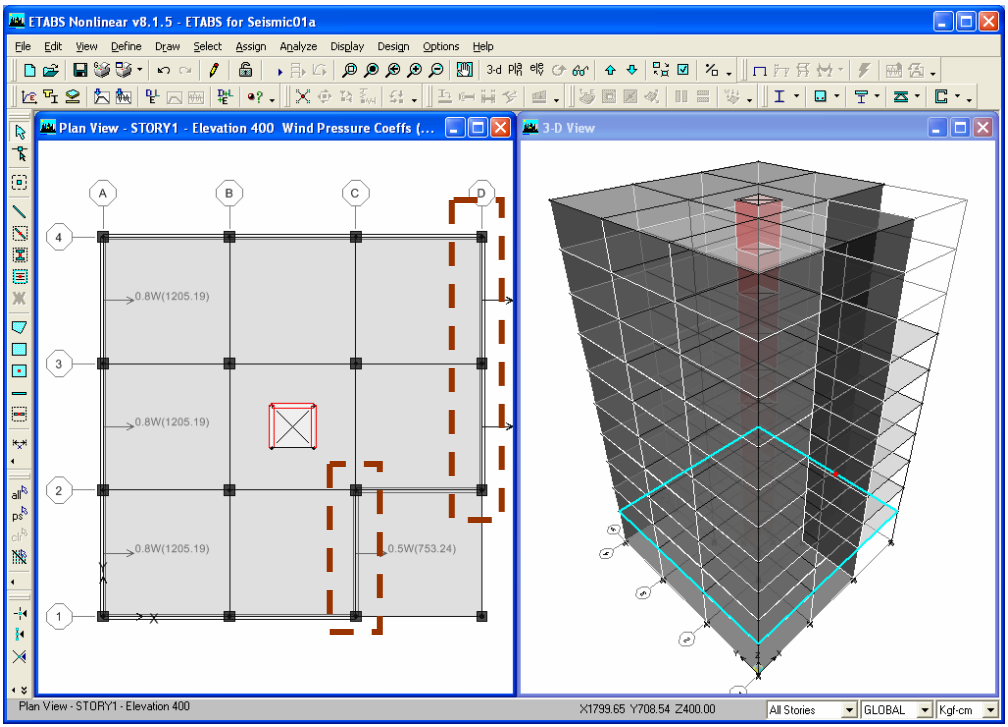
Click on *Select Object* button, select null areas on the side of the building, go to **Assign >> Shell/Area Loads >> Wind Pressure Coefficient** and specify parameters as shown in figure.

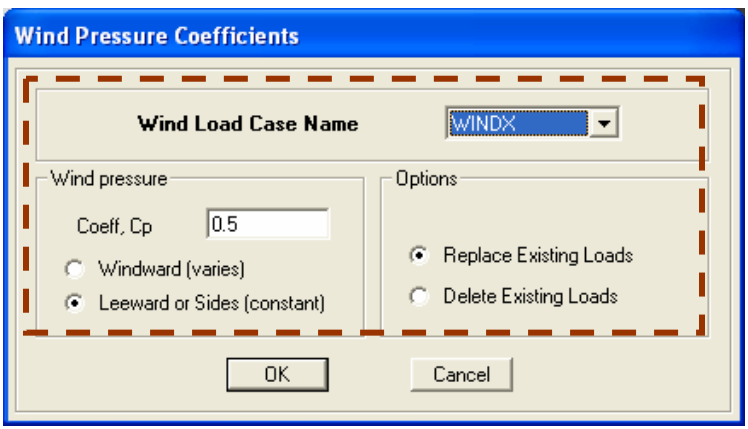


Note: Positive Direction of wind pressure is same as positive direction of local area axes 3. To check area local axes in shell area, go to **View >> Set Building View Options** or click on *Set Building View Options* button and select "Area Local Axes" in "Object View Options". ETABS will display 3 local axes in 3 color arrows (Red, white and blue to represent 1, 2 and 3 local axes).

Step 11-6: Assign Leeward Wind Coefficient to Null Areas for "WINDX" Load Case

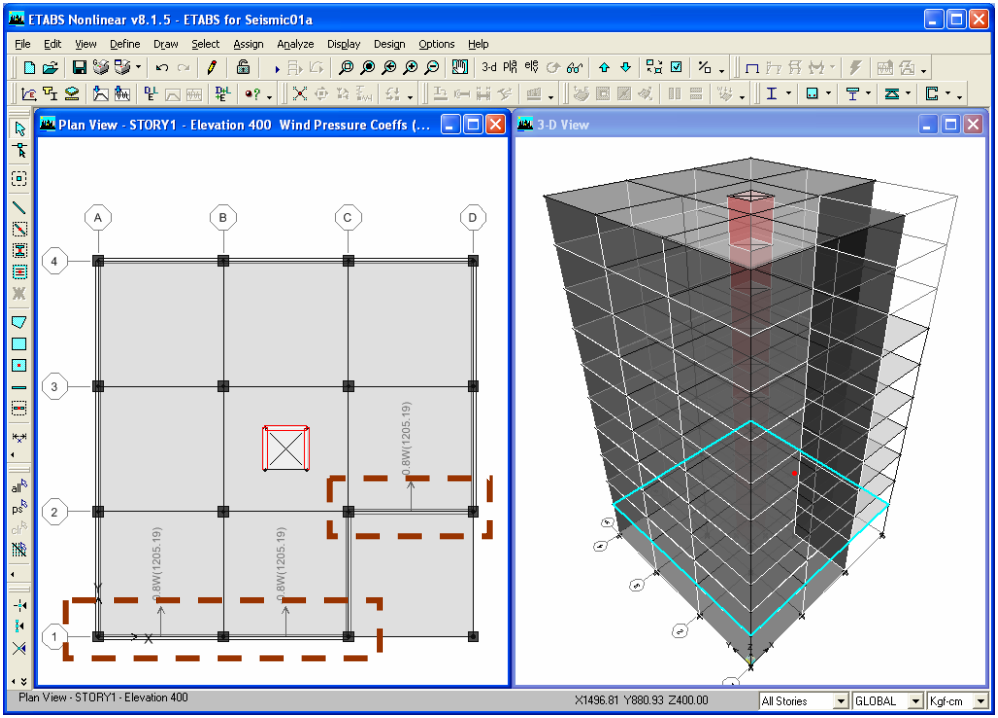
Select null areas on the side of the building, go to **Assign >> Shell/Area >> Wind Pressure Coefficient** and specify parameters as shown in figure.

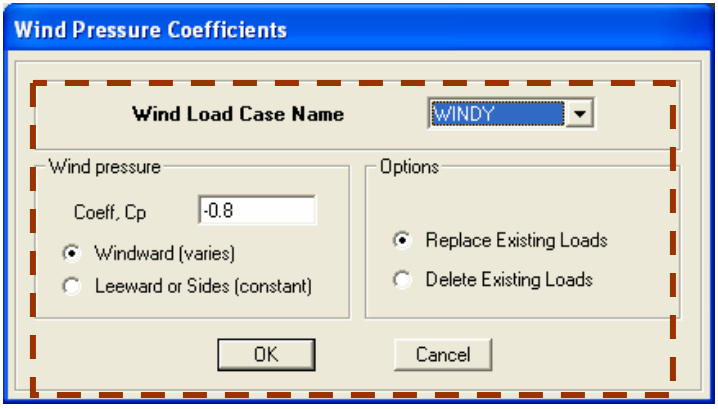




Step 11-7: Assign Windward Wind Coefficient to Null Areas for "WINDY" Load Case

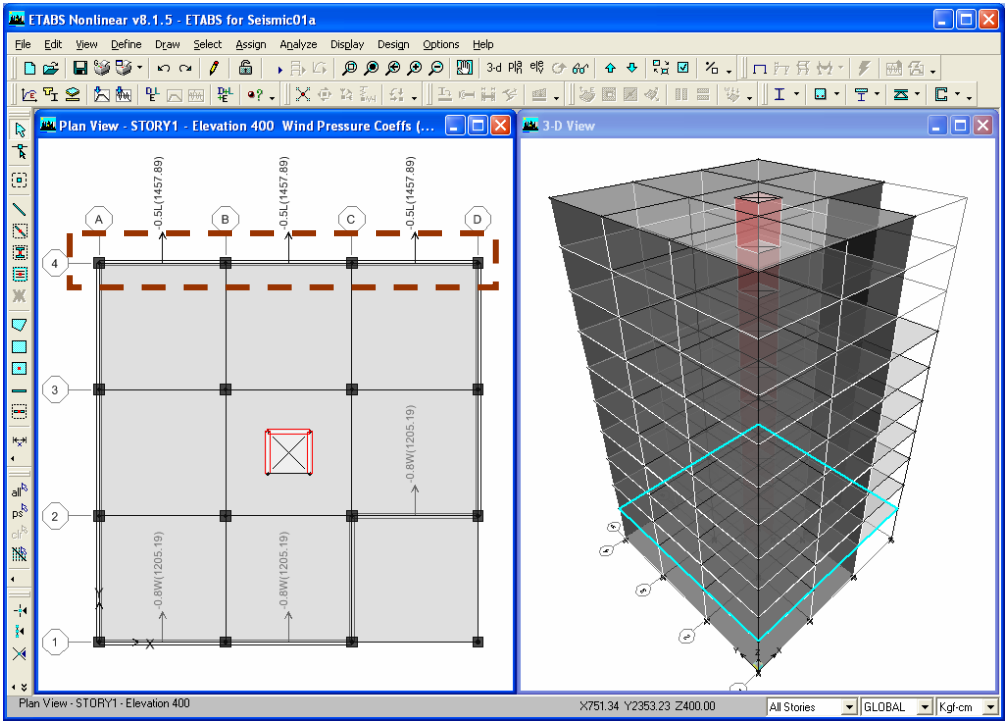
Select null areas on the side of the building, go to **Assign >> Shell/Area >> Wind Pressure Coefficient** and specify parameters as shown in figure.

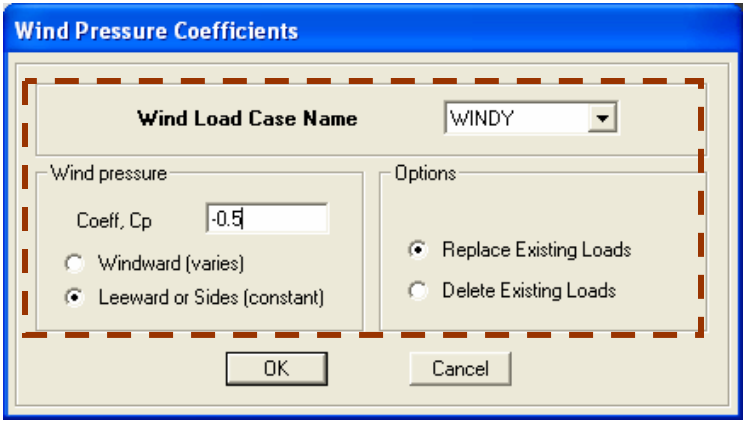




Step 11-8: Assign Leeward Wind Coefficient to Null Areas for "WINDY" Load Case

Select null areas on the side of the building, go to **Assign >> Shell/Area >> Wind Pressure Coefficient** and specify parameters as shown in figure.

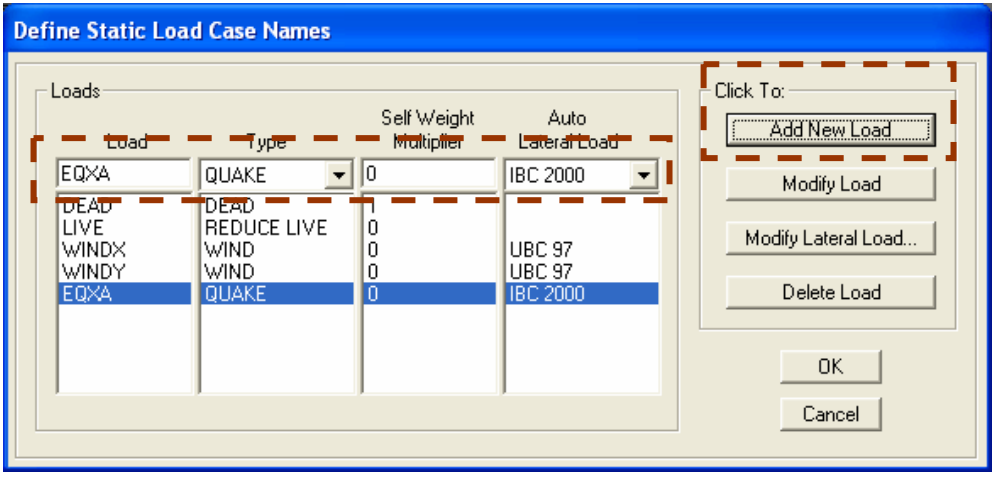




12. Define Static Load Case for Equivalent Seismic Force

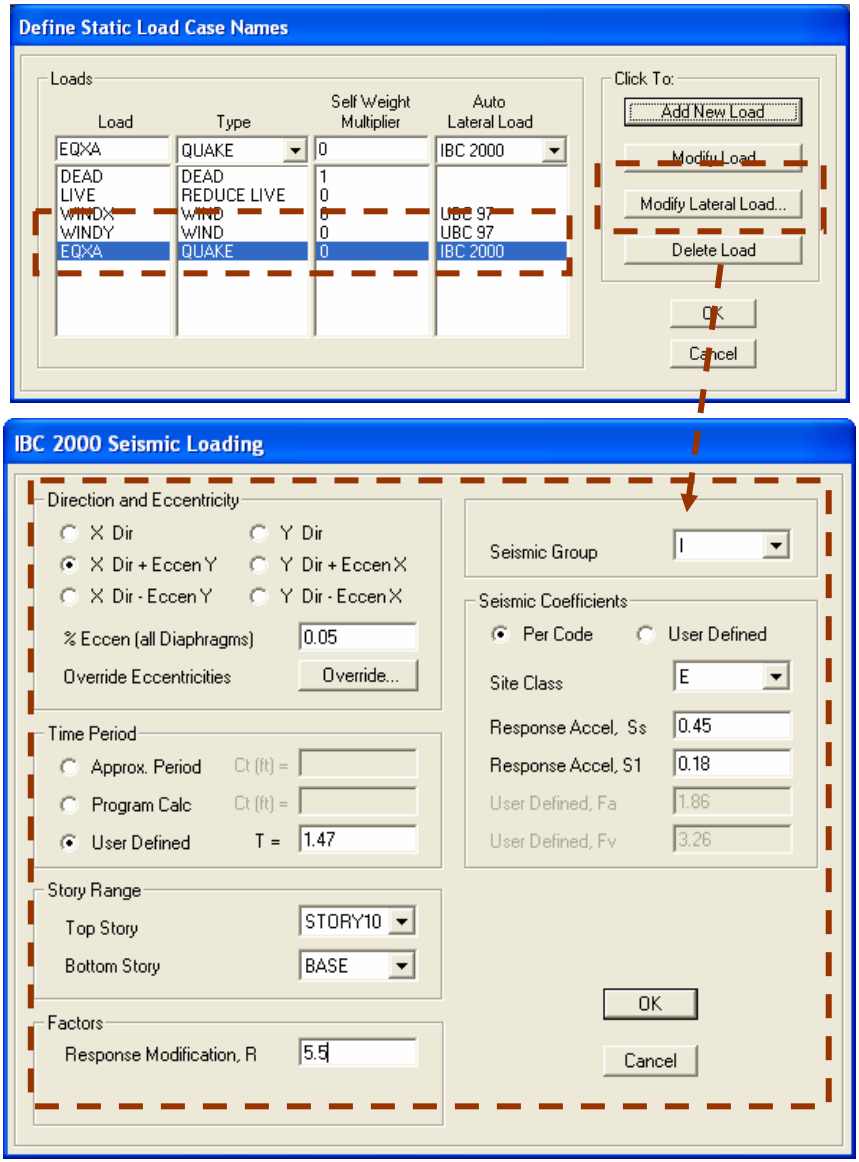
Step 12-1: Add "EQXA" Load Case

Go to **Define >> Static Load Case**, define load case parameters as shown in figure below and click on "Add New Load".



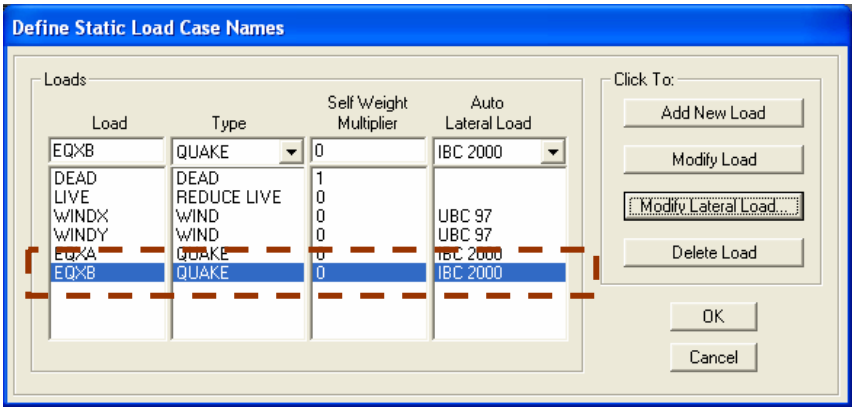
Step 12-2: Specify "EQXA" Load Case Parameters

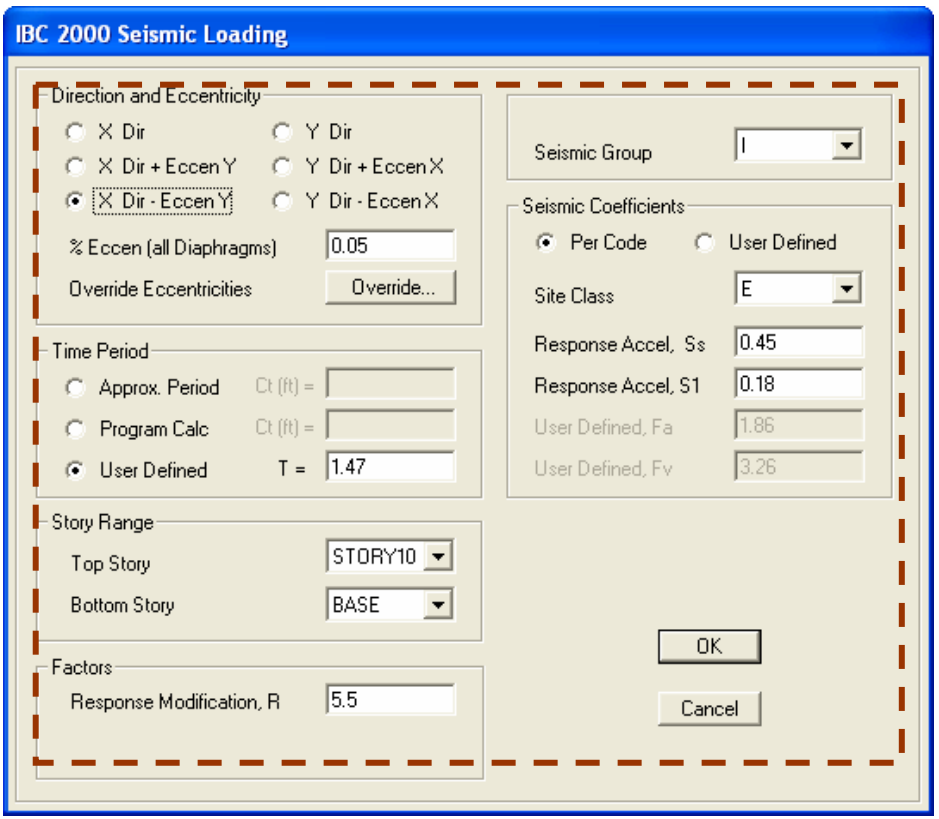
Select "EQXA" load case from list, click on "Modify Lateral Load" and specify parameters as shown in figure below.



Step 12-3: Add "EQXB" Load Case and Specify Load Case Parameters

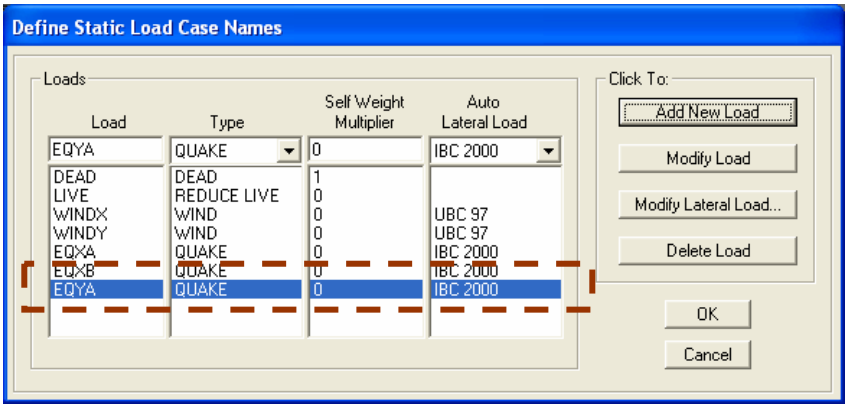
Repeat step 11-1 and 11-2 to add and specify "EQXB" load case parameters

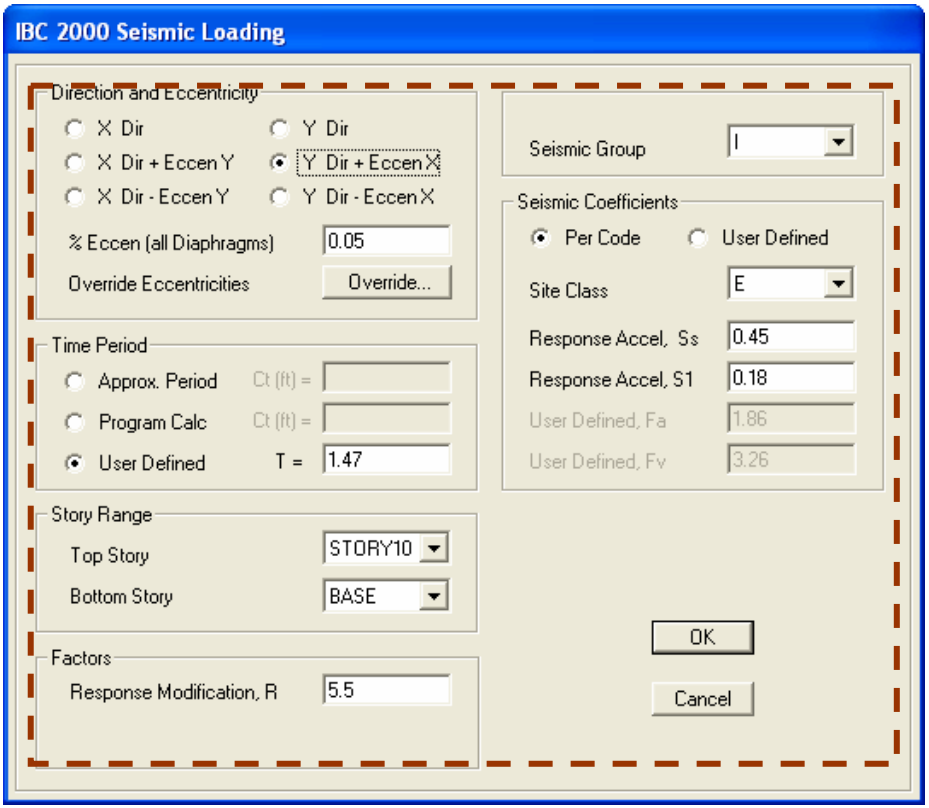




Step 12-4: Add "EQYA" Load Case and Specify Load Case Parameters

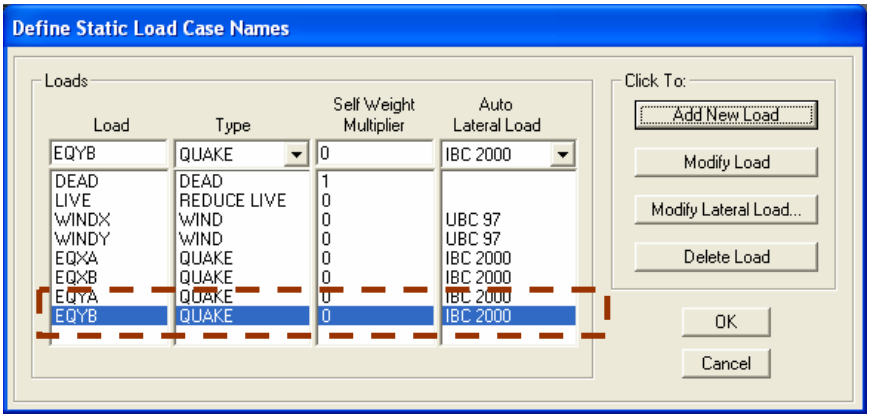
Repeat step 11-1 and 11-2 to add and specify "EQYA" load case parameters

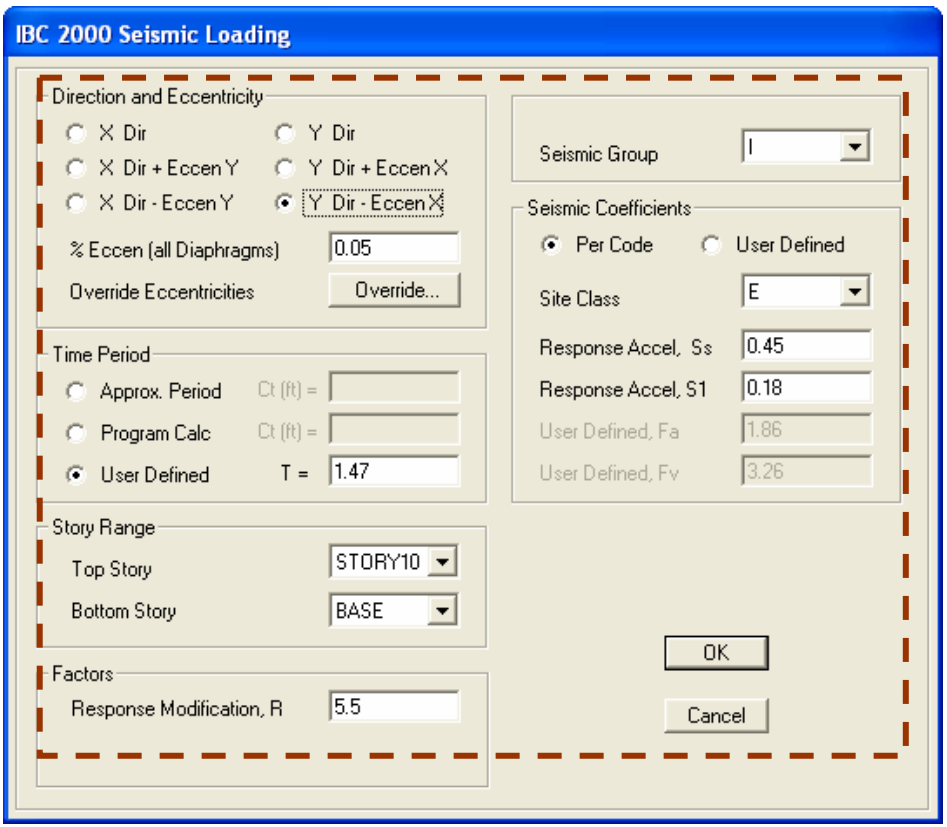




Step 12-5: Add "EQYB" Load Case and Specify Load Case Parameters

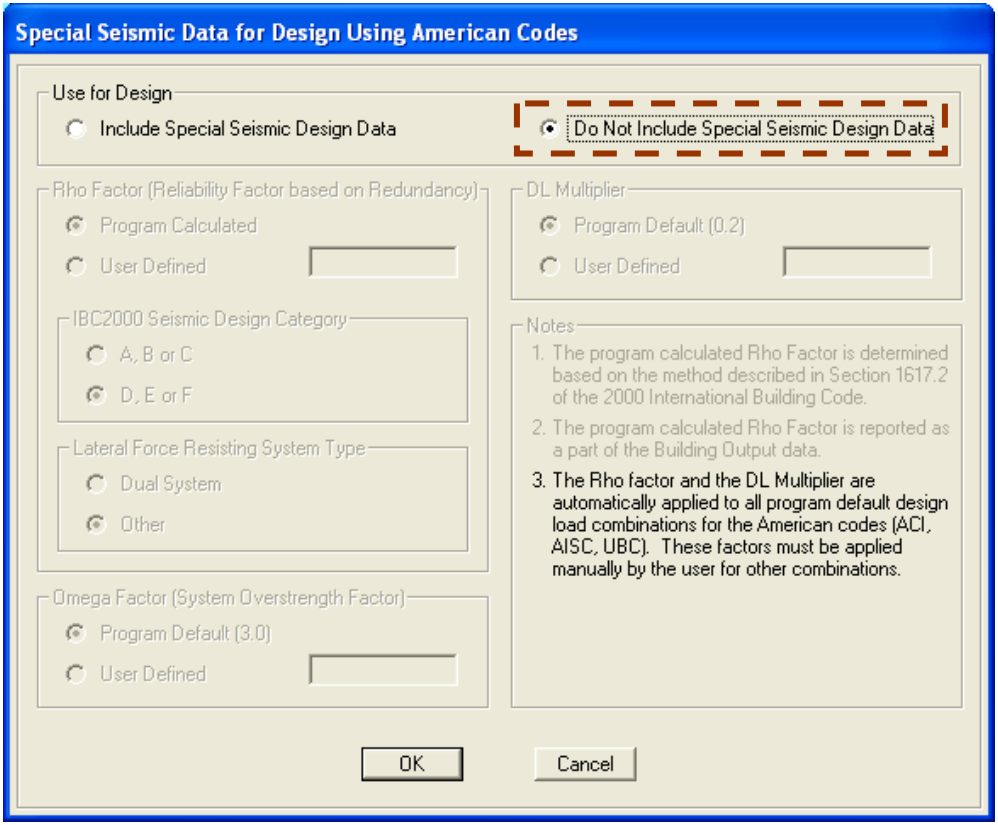
Repeat step 11-1 and 11-2 to add and specify "EQYB" load case parameters





Step 12-6: Deactivate Special Seismic Load Effect

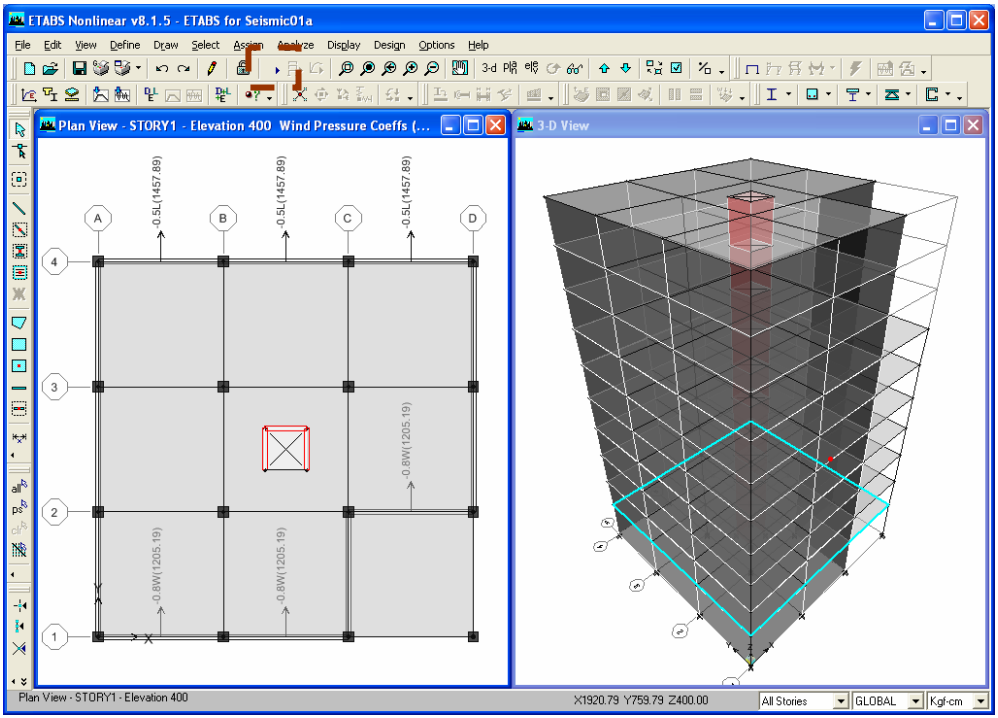
Go to **Define >> Special Seismic Load Effect** and select "Do Not Include Special Seismic Design Data"



13. Run Analysis and View Results

Step 13-1: Start Analysis

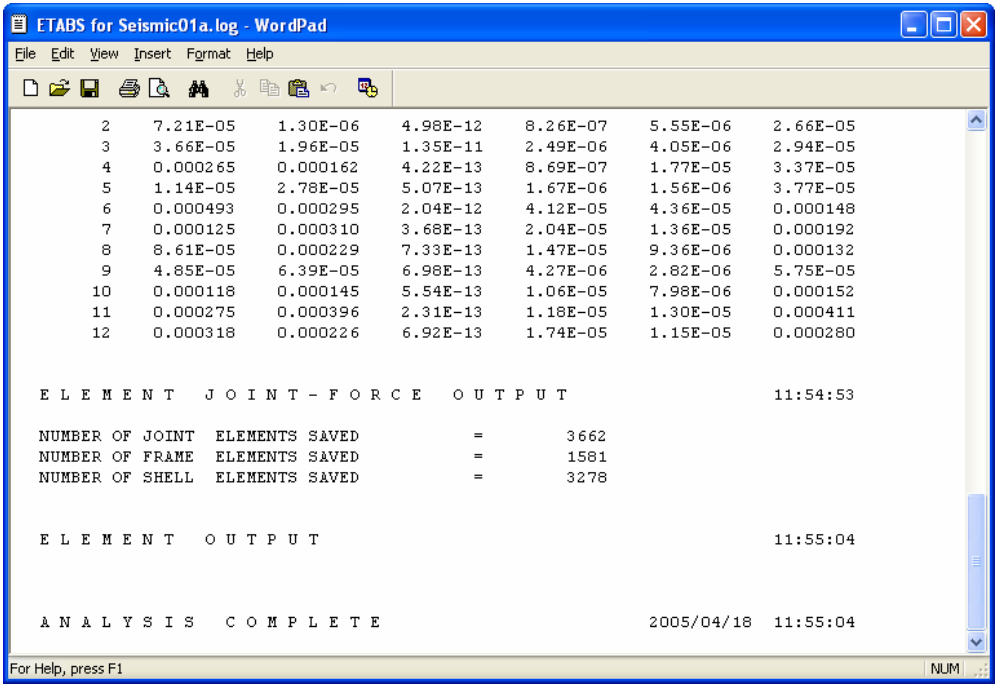
Go to **Analyze >> Run Analysis** or click on *Run Analysis* button to start analysis.
ETABS will display deformed shape of model when analysis complete.



Note: ETABS will lock the model automatically from undesired changes. Model will be unlocked by clicking on *Unlock Model* button . ETABS will delete all analysis and design results after unlock.

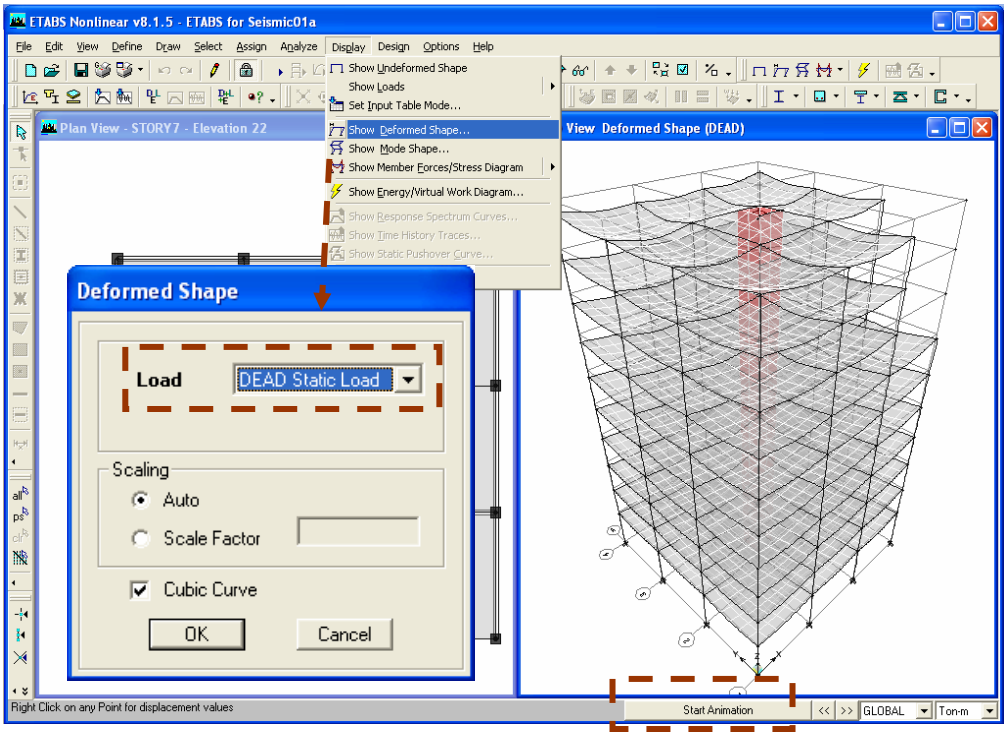
Step 13-2: Check Error from Analysis Run Record

Go to **File >> Last Analysis Run Log** and scroll down to check error message.



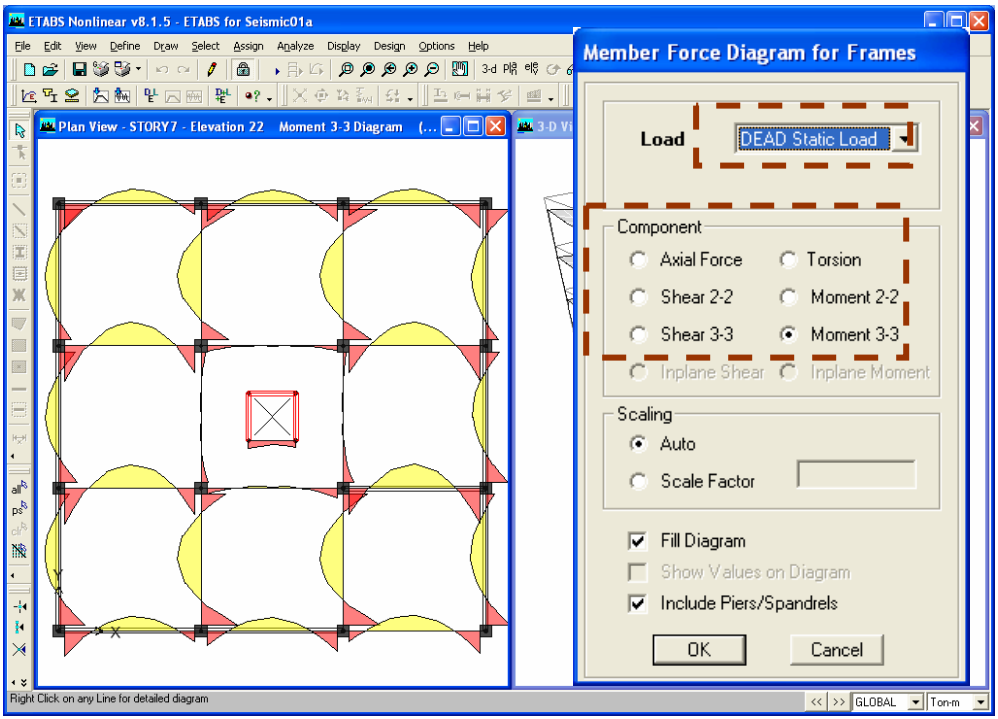
Step 13-3: Display Deformed Shape in 3D View

Select 3D view window, go to **Display >> Show Deformed Shape** and select desired load from drop-down menu. To view deformed shape in animation, click on "Start Animation".

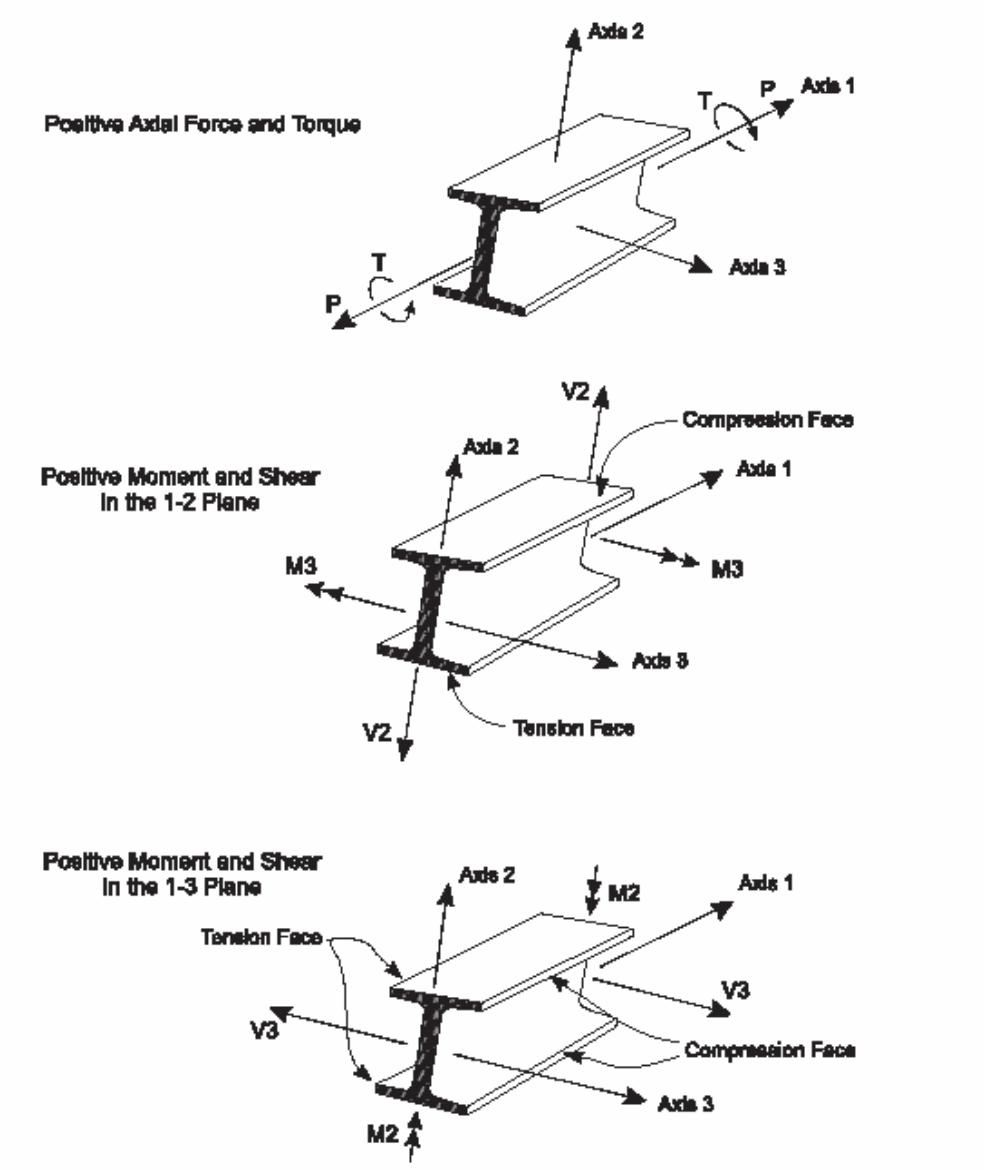


Step 13-4: View Analysis Result Diagrams of Frame Elements (Beam or Column)

Go to **Display >> Show Member Forces/Stress Diagram >> Frame/ Pier/Spandrel Forces** and select "Load" and "Component"

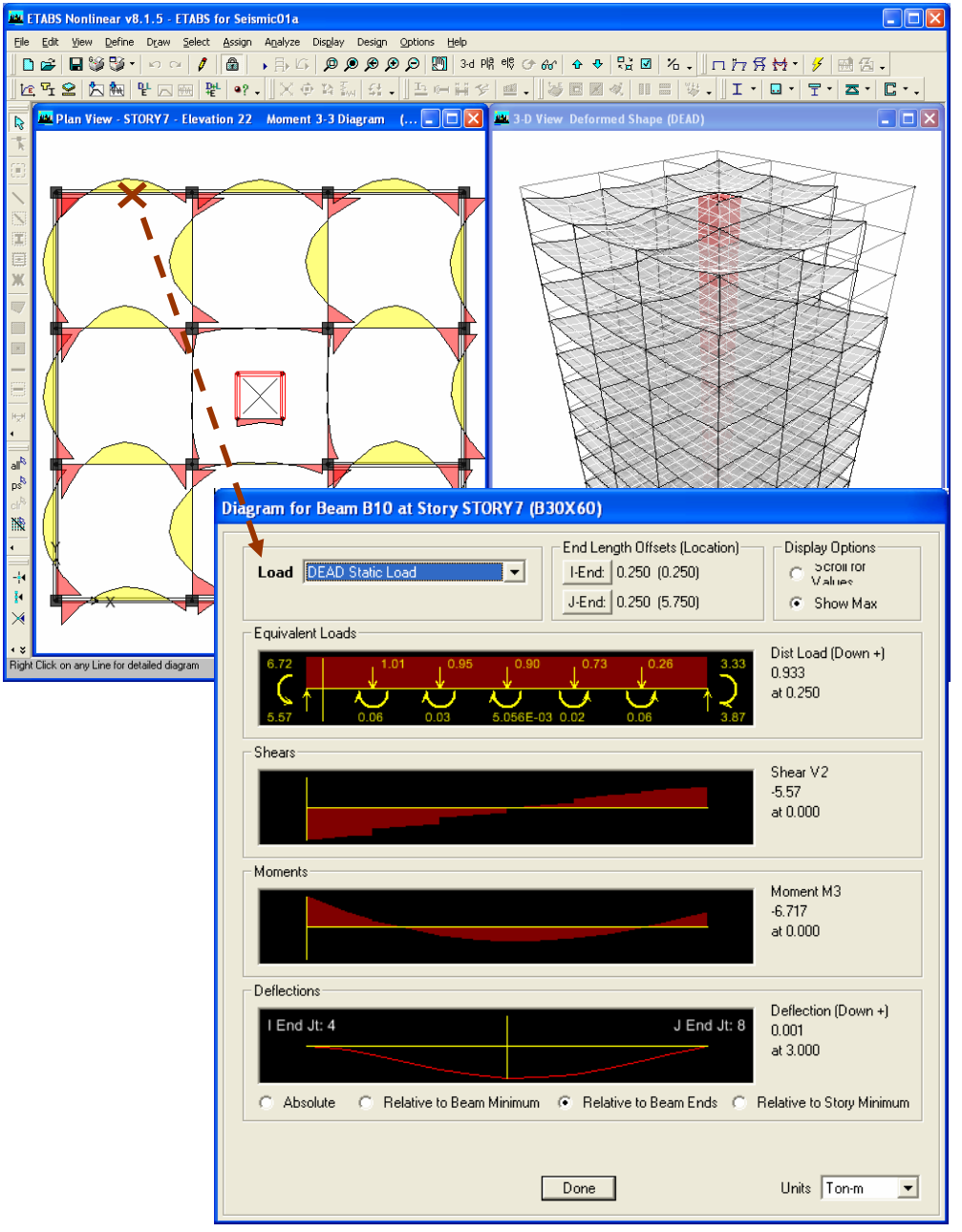


Note: Sign Convention for Frame Element



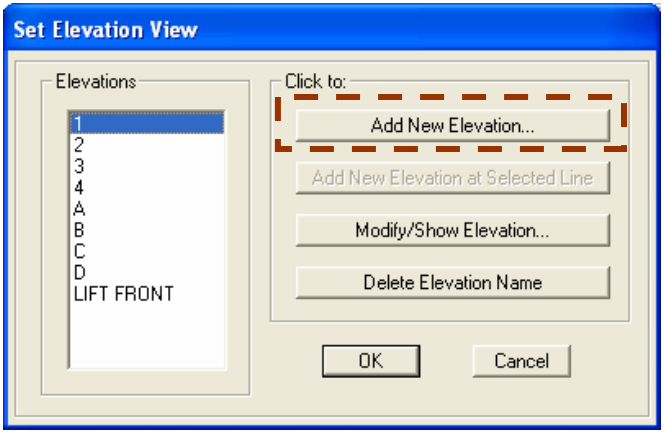
Step 13-5: View Analysis Result Diagram at Particular Frame Element

Right click on desired beam to display particular analysis result diagram

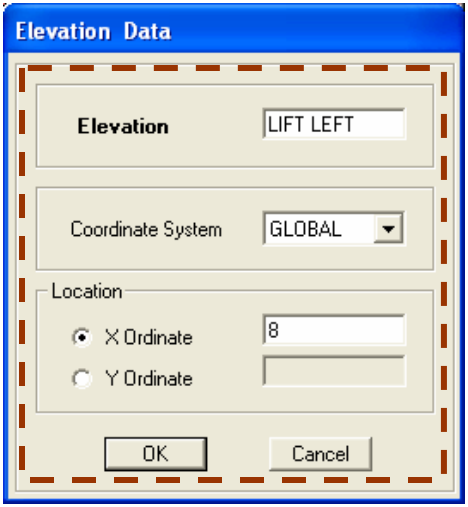


Step 13-6: Create Elevation View for Display Analysis Results in Wall Panels

Click on *Set Elevation View* button and click on "Add New Elevation"



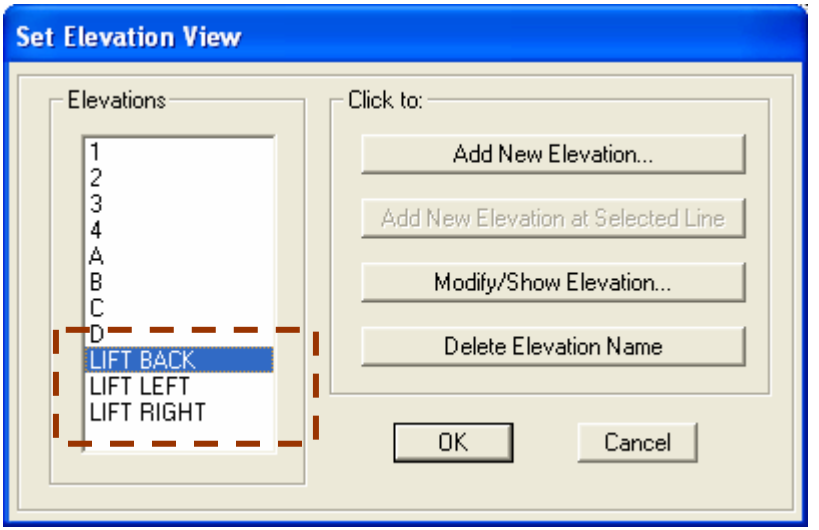
Specify "Location" at wall panels around elevators as shown in the following figure and table.



Elevation Name	X Ordinate	Y Ordinate
LIFT LEFT	8	-
LIFT RIGHT	10	-
LIFT BACK	-	10

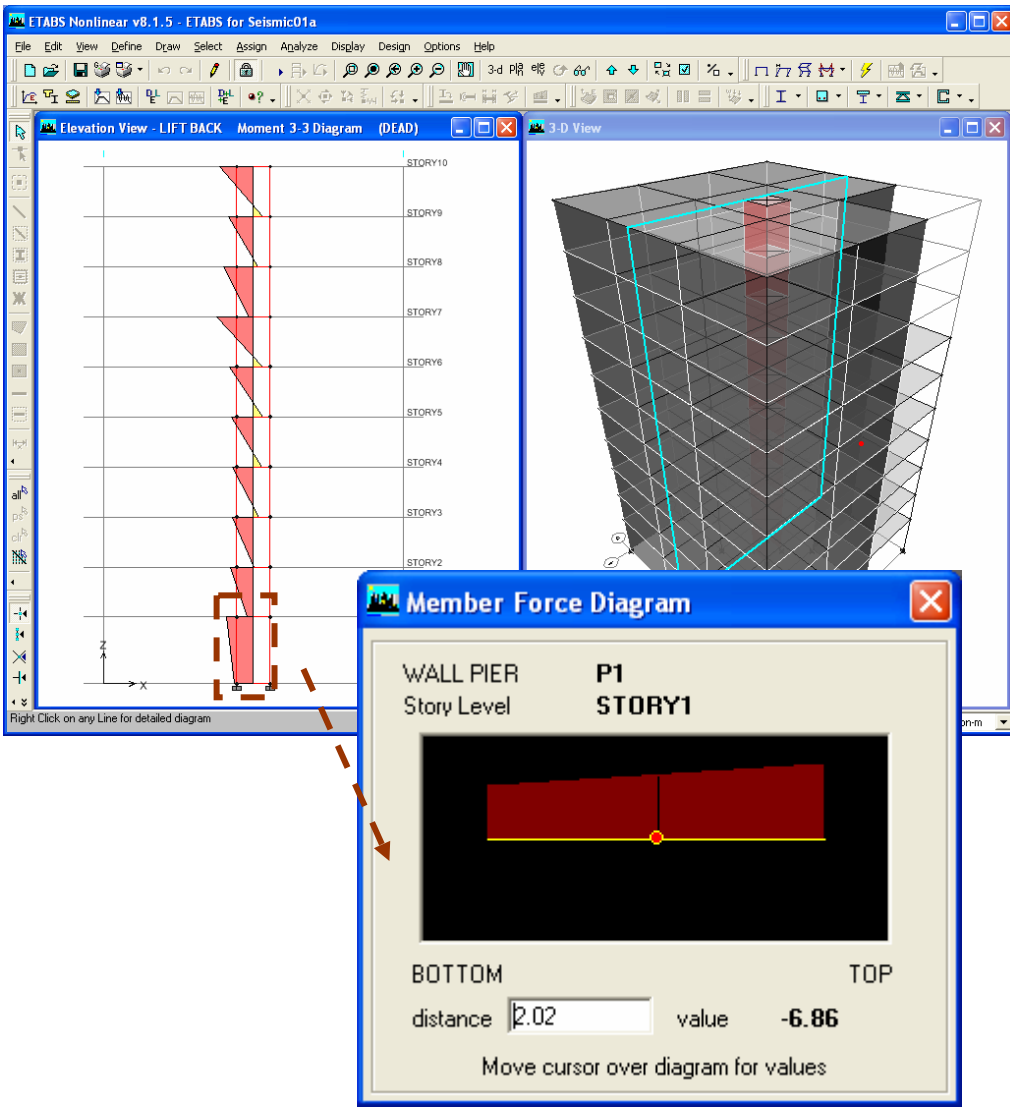
Step 13-7: Change View to Elevation View at Elevator Location

Click on *Set Elevation View* button and select elevation view at elevator location



Step 13-8: View Analysis Result Diagrams of Shear Wall (Pier)

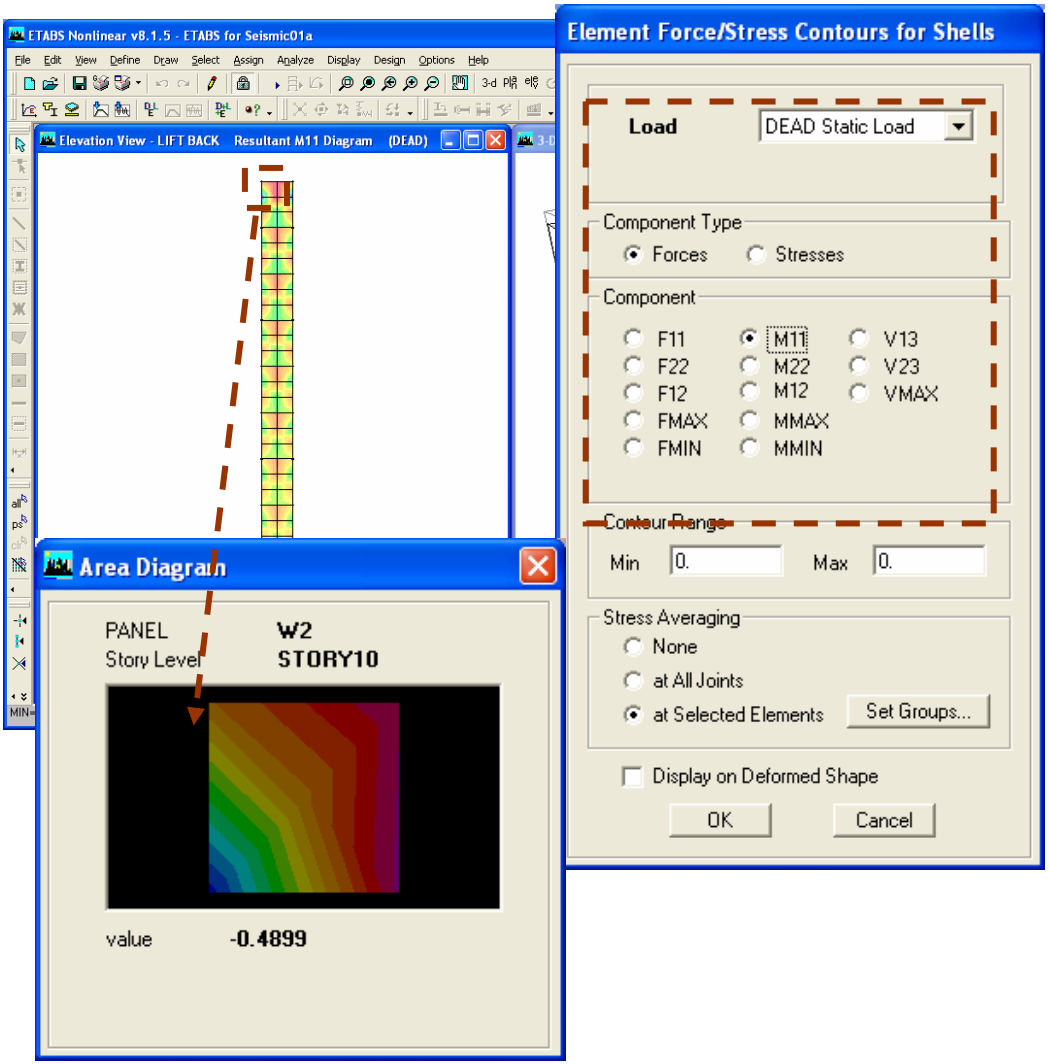
Right click on desired shear wall panel to view particular diagram



Note: Same as frame element, move mouse cursor over this diagram and see value at bottom of this window to check analysis results in particular location

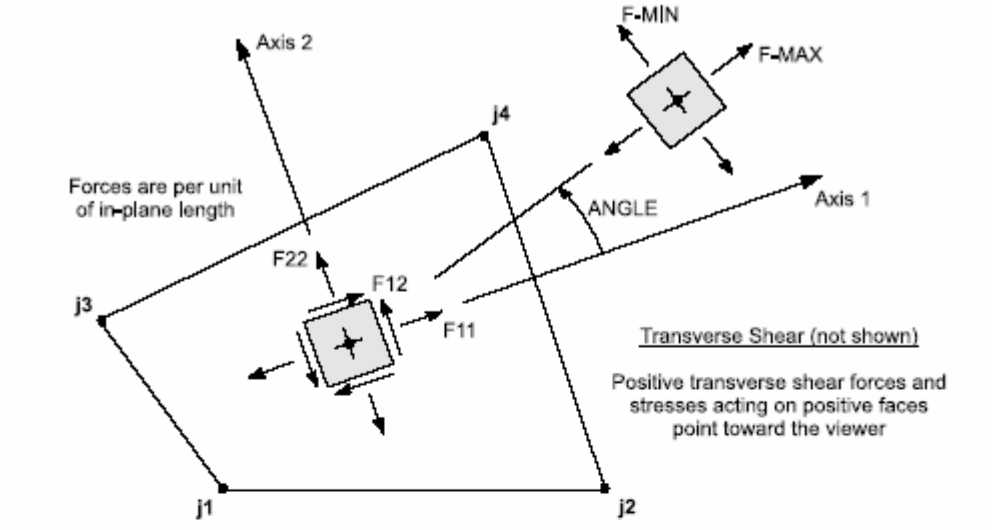
Step 13-9: View Analysis Result Contour in Shear Wall Panels (Elevation View)

Change "Plan View" to "Elevation View" by clicking on *Set Elevation View* button and selecting desired elevation for elevator location, go to **Display >> Show Member Forces/Stress Diagram >> Shell Stresses/Forces**, select "Load" and "Component". Right click on desired wall panel to view particular analysis result.



Note: Analysis results at particular location will display at the bottom of window when move mouse cursor over this diagram.

Note: Sign Convention for Shell Element



STRESSES AND MEMBRANE FORCES

Stress Sij Has Same Definition as Force Fij

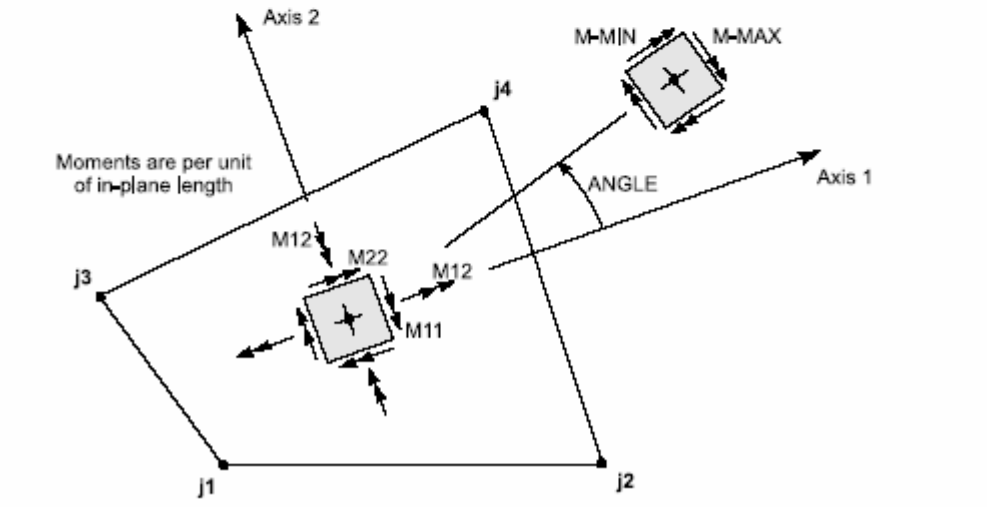
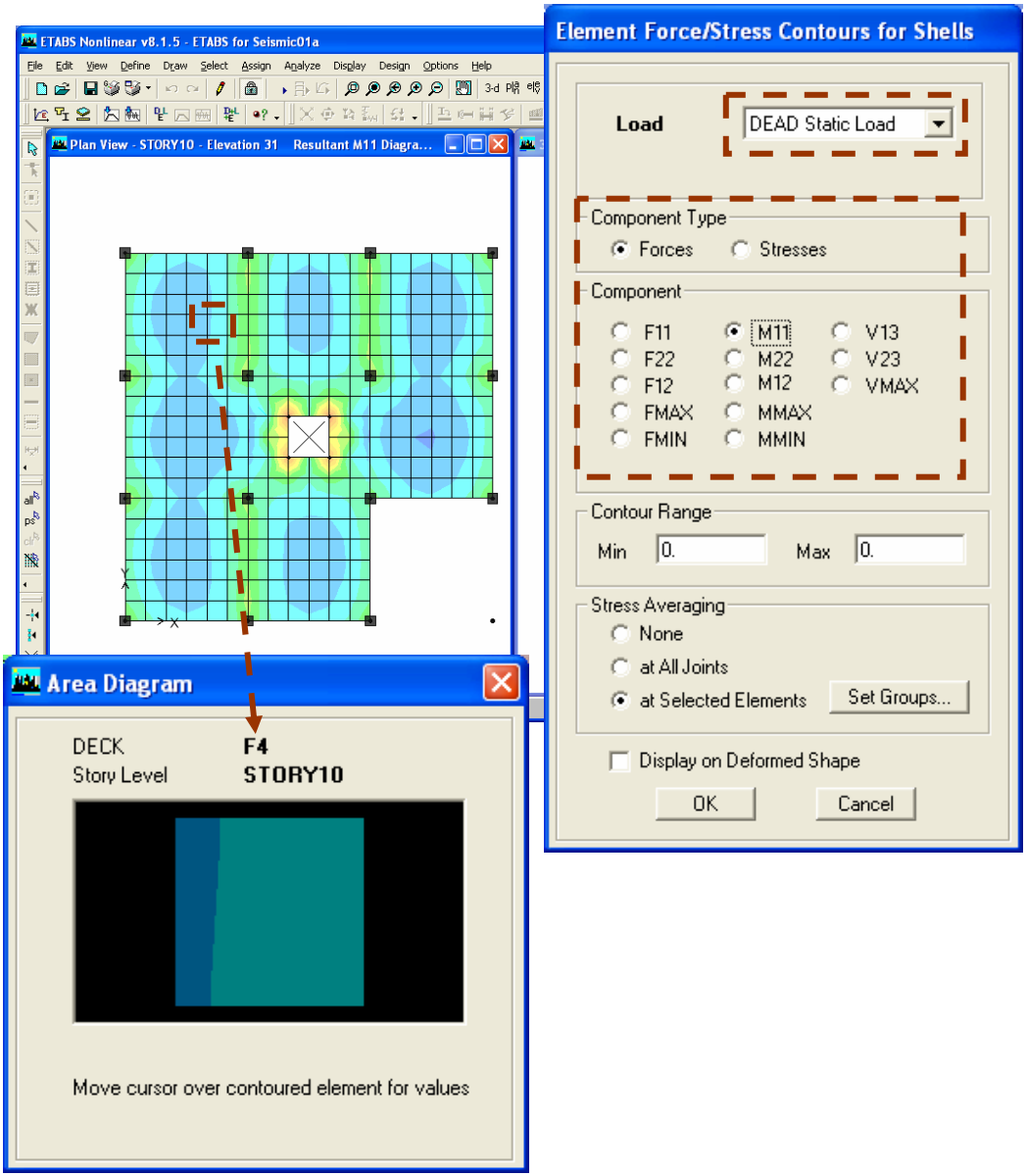


PLATE BENDING AND TWISTING MOMENTS

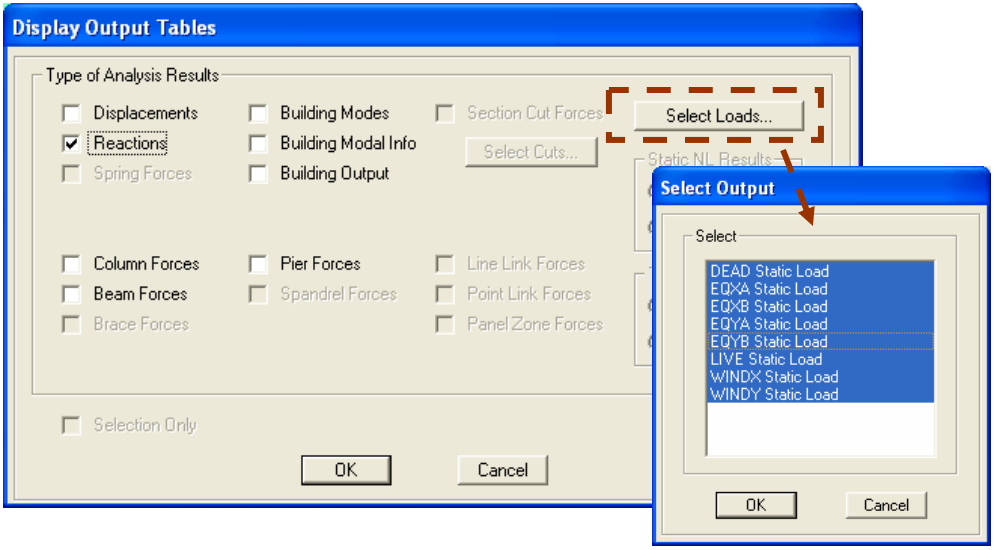
Step 13-10: View Analysis Result Contour in Slab Panels (Plan View)

Change to "Plan View" by clicking on *Set Plan View* button and selecting desired floor, go to **Display >> Show Member Forces/Stress Diagram >> Shell Stresses/Forces** and select "Load" and "Component". Same as shear wall panel, right click on desired wall panel to view particular analysis result.

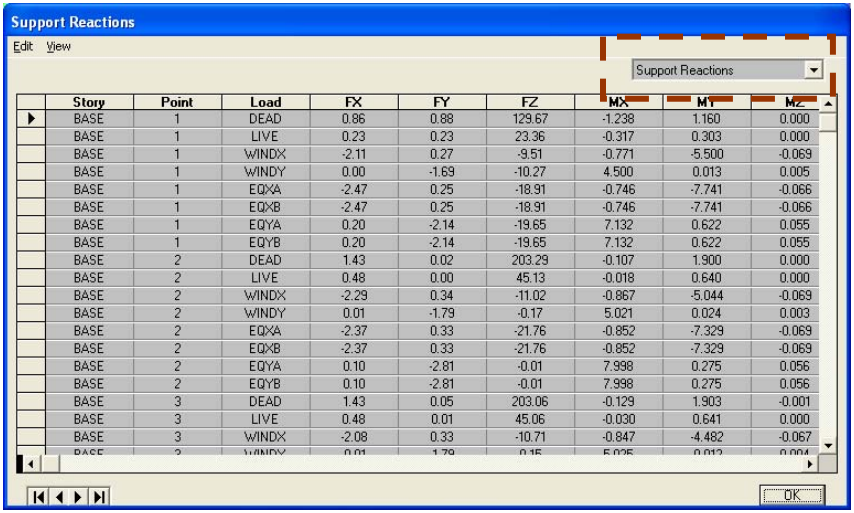


Step 13-11: View Analysis Results in Tabular Form

Go to **Display >> Show Output table Mode**, select desired items and click on "Select Loads" to specify load case/combination.



Select analysis results from drop-down menu at top-right of screen



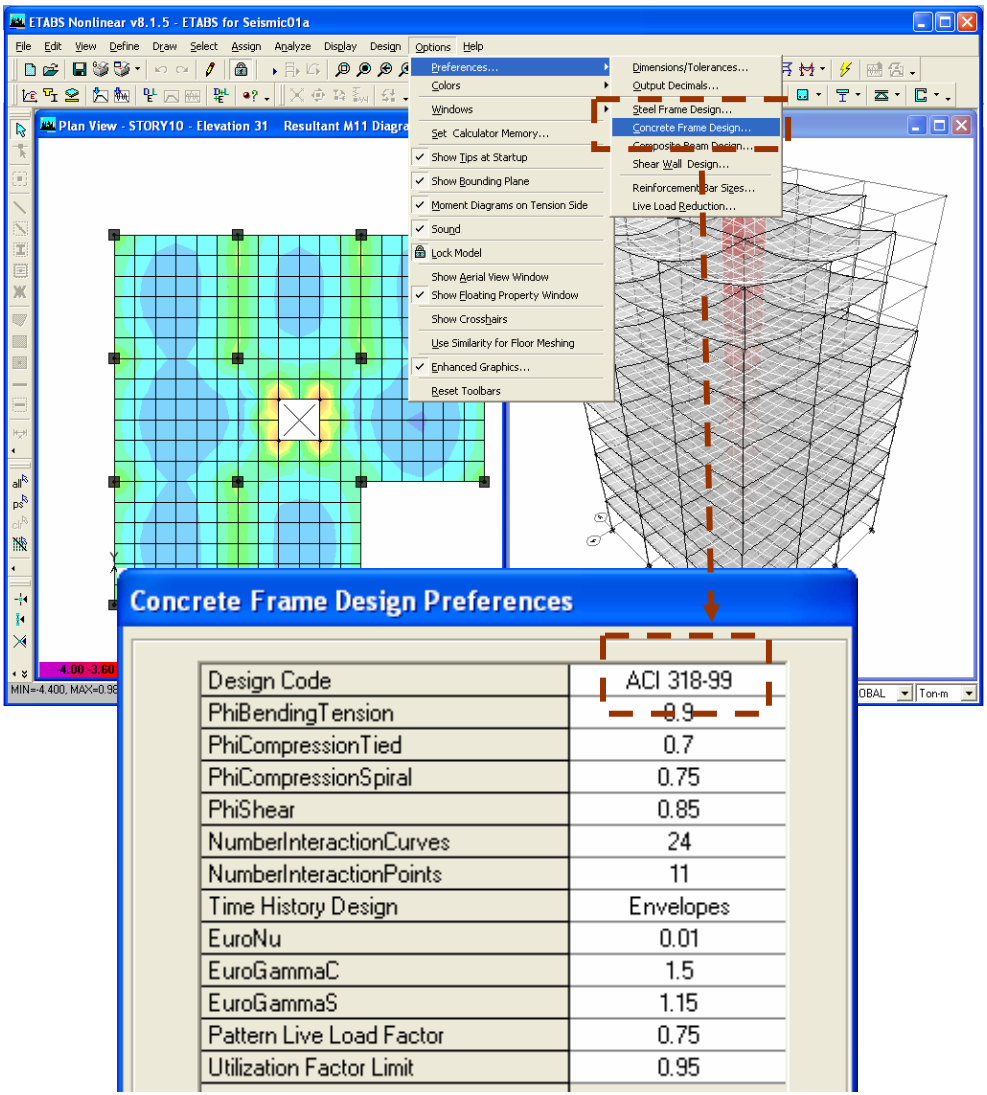
Story	Point	Load	FX	FY	FZ	MX	MY	MZ
BASE	1	DEAD	0.86	0.88	129.67	-1.238	1.160	0.000
BASE	1	LIVE	0.23	0.23	23.36	-0.317	0.303	0.000
BASE	1	WINDX	-2.11	0.27	-9.51	-0.771	-5.500	-0.069
BASE	1	WINDY	0.00	-1.69	-10.27	4.500	0.013	0.005
BASE	1	EQXA	-2.47	0.25	-18.91	-0.746	-7.741	-0.066
BASE	1	EQXB	-2.47	0.25	-18.91	-0.746	-7.741	-0.066
BASE	1	EQYA	0.20	-2.14	-19.65	7.132	0.622	0.055
BASE	1	EQYB	0.20	-2.14	-19.65	7.132	0.622	0.055
BASE	2	DEAD	1.43	0.02	203.29	-0.107	1.900	0.000
BASE	2	LIVE	0.48	0.00	45.13	-0.018	0.640	0.000
BASE	2	WINDX	-2.29	0.34	-11.02	-0.867	-5.044	-0.069
BASE	2	WINDY	0.01	-1.79	-0.17	5.021	0.024	0.003
BASE	2	EQXA	-2.37	0.33	-21.76	-0.852	-7.329	-0.069
BASE	2	EQXB	-2.37	0.33	-21.76	-0.852	-7.329	-0.069
BASE	2	EQYA	0.10	-2.81	-0.01	7.998	0.275	0.056
BASE	2	EQYB	0.10	-2.81	-0.01	7.998	0.275	0.056
BASE	3	DEAD	1.43	0.05	203.06	-0.129	1.903	-0.001
BASE	3	LIVE	0.48	0.01	45.06	-0.030	0.641	0.000
BASE	3	WINDX	-2.08	0.33	-10.71	-0.847	-4.482	-0.067

Note: This table can be copied to MS Excel by using **Edit >> Copy** menu in this window (Not main menu).

14. Run Concrete Frame Design and View Results

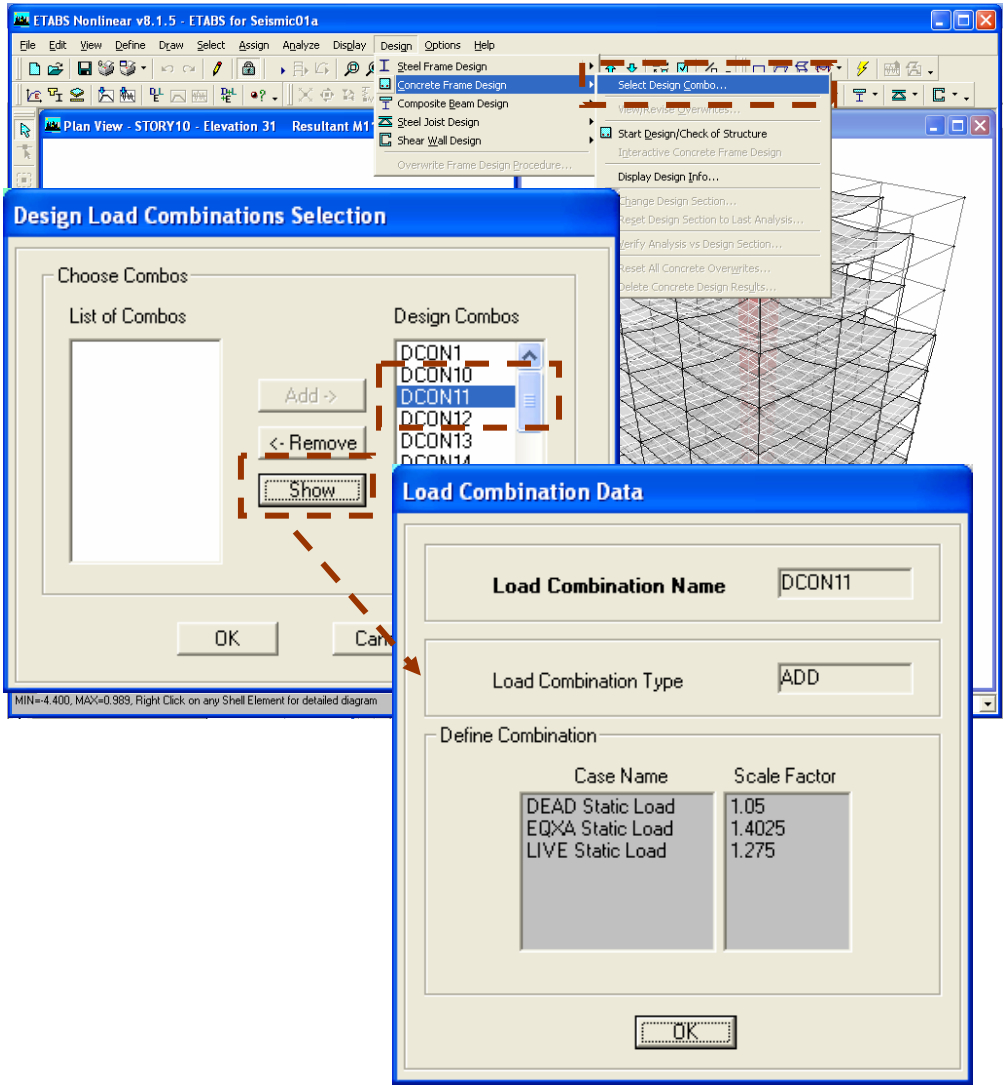
Step 14-1: Select Design Code

Go to **Options >> Preference >> Concrete Frame Design** and select “ACI 318-99” from “Design Code”



Step 14-2: View Load Combination for Concrete Frame Design

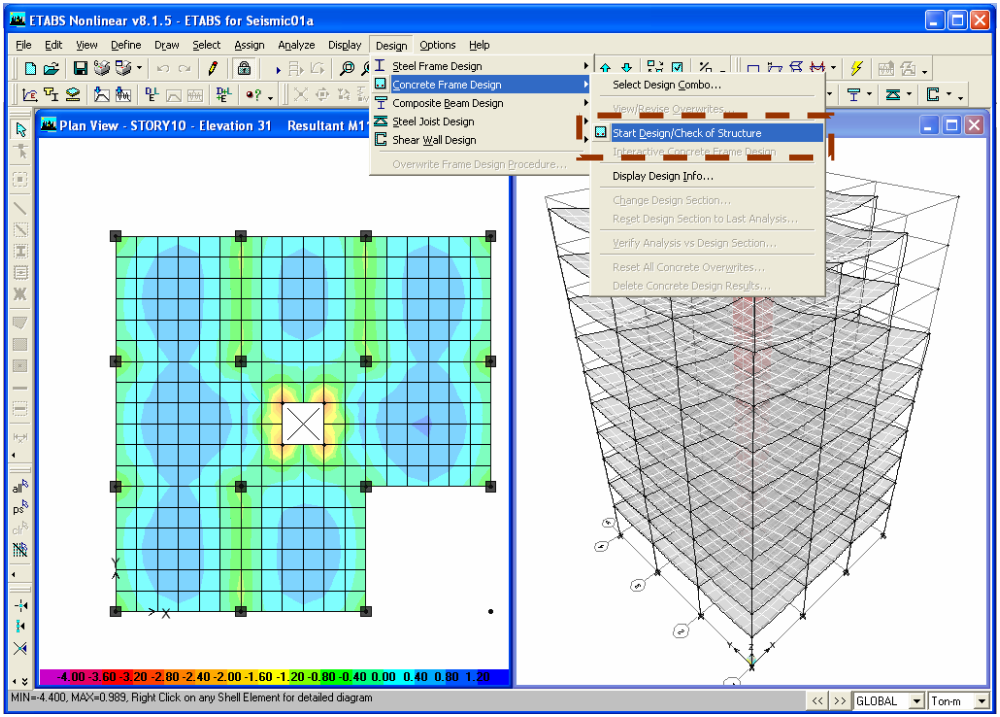
Go to **Design >> Concrete Frame Design >> Select Design Combo** to view load combination for concrete frame design. Load combinations have been defined as selected code from previous step. Select desired load combination from "Design Combos" column and click on "Show" to view load combination parameters (load factors and details)



Note: ETABS will define load combination automatically based on selected design from previous step.

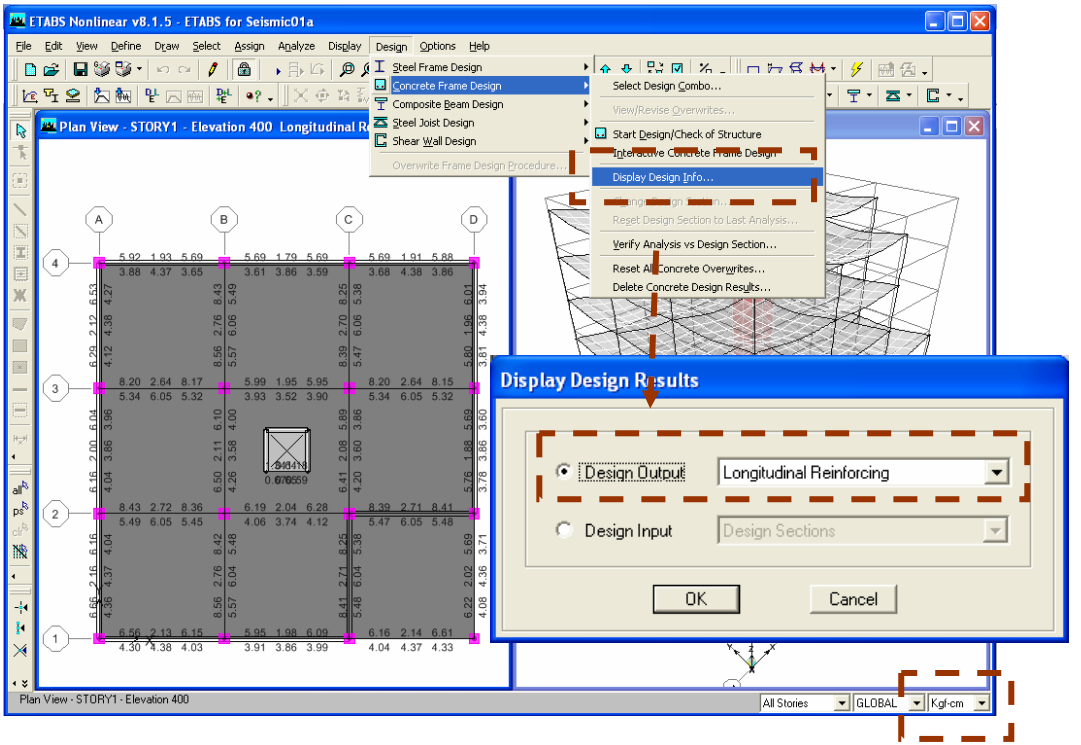
Step 14-3: Start Concrete Frame Design

Go to **Design >> Concrete Frame Design >> Start Design/Check Structure**



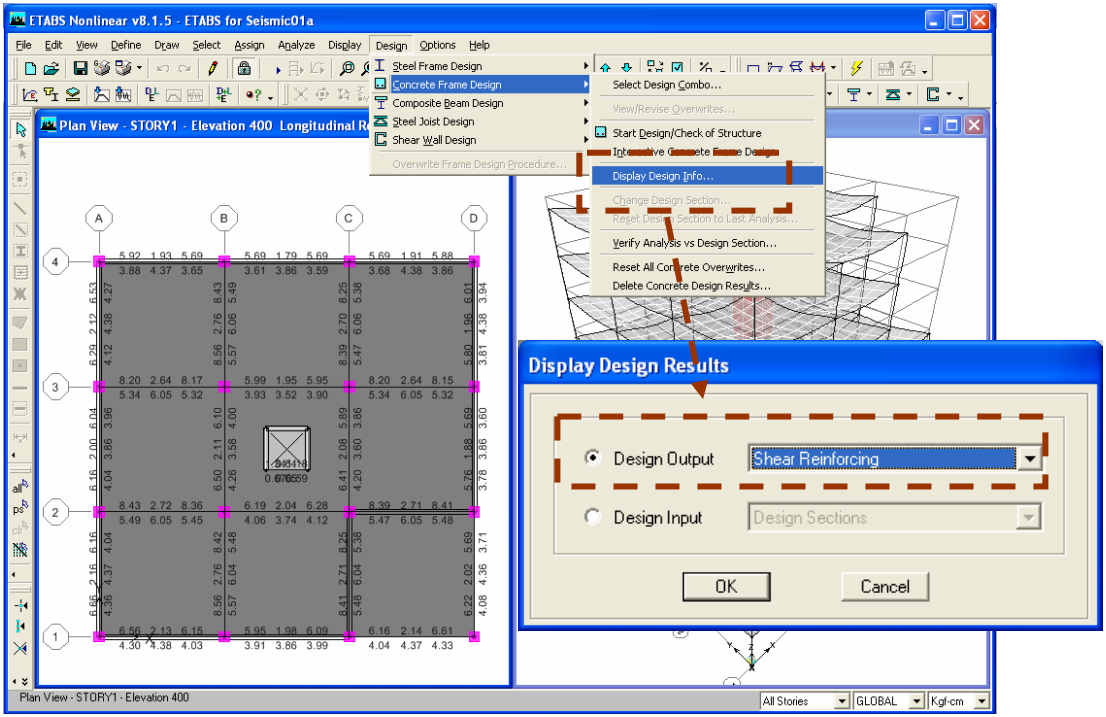
Step 14-4: Display Longitudinal Reinforcing for Concrete Frame Design

Select “kg-cm”, go to **Design >> Concrete Frame Design >> Display Design Info**, click on “Design Output” and select “Longitudinal Reinforcing” from first drop-down menu.



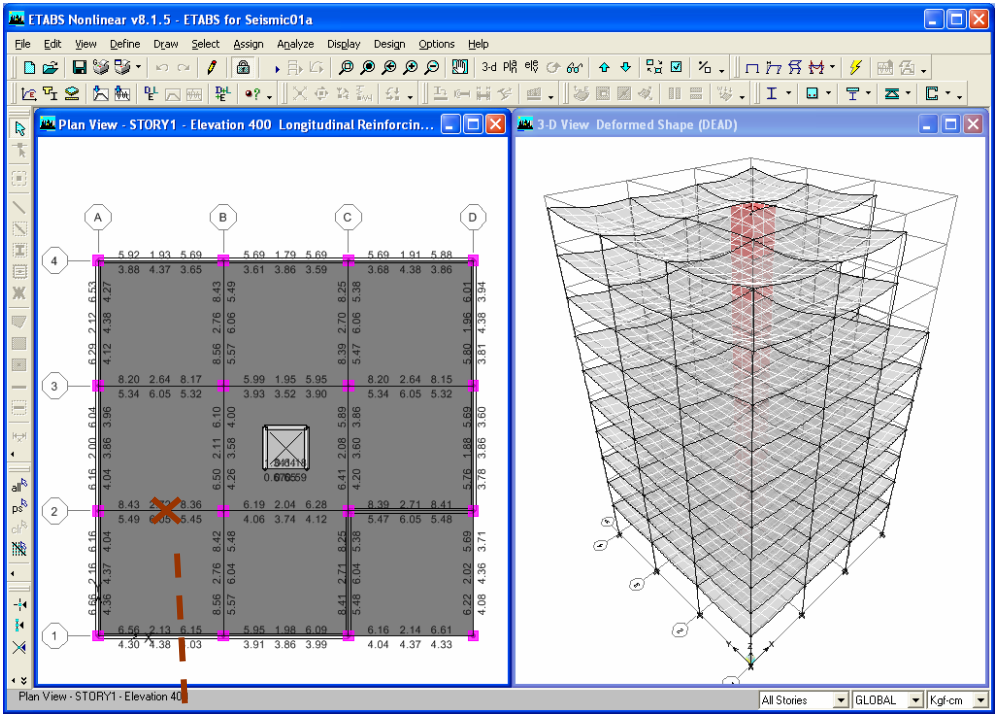
Step 14-5: Display Shear Reinforcing for Concrete Frame Design

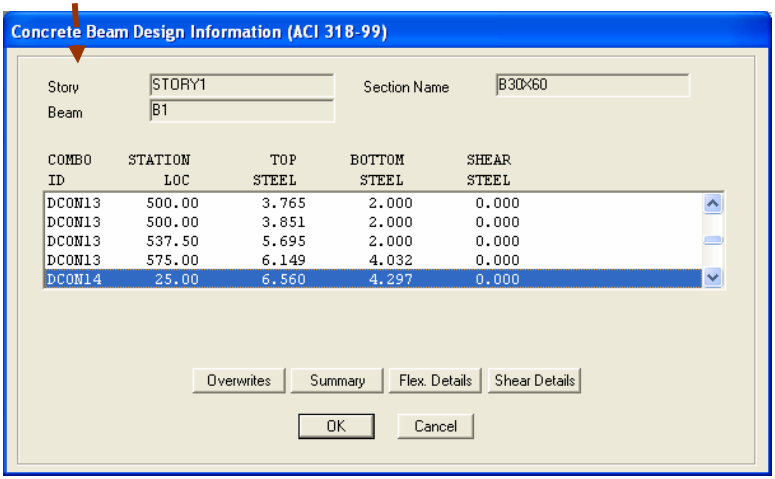
Go to **Design >> Concrete Frame Design >> Display Design Info**, click on "Design Output" and select "Shear Reinforcing" from first drop-down menu.



Step 14-6: Display Concrete Frame Design in Details

To see concrete frame design in details, right mouse click on desired element. The highlighted row is 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.





15. Run Shear Wall Design and View Results

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. After create the building model 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. Run the building analysis using the **Analyze menu > Run Analysis** command.
3. Assign the wall pier and wall spandrel labels. Use the **Assign menu > Frame/Line > Pier Label**, the **Assign menu > Shell/Area > Pier Label**, 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.

4. 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.

5. 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.
6. Click the **Design menu > Shear Wall Design > Start Design/Check of Structure** command to run the shear wall design.
7. Review the shear wall design results. To do this you might do one of the following:

    a. Click the **Design menu > Shear Wall Design > Display Design Info** command to display design information on the model.

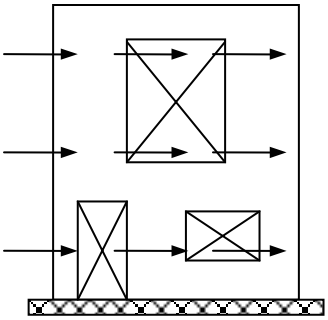
    b. Right click on a pier or spandrel while the design results are displayed on it to enter the interactive wall design mode. Note that while you are in this mode you can revise overwrites and immediately see the new design results.

    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.

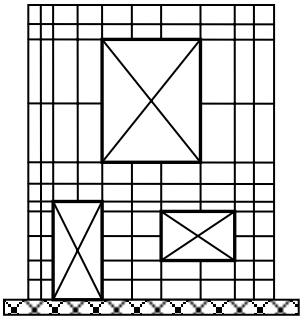
1. Use the **File menu > Print Tables > Shear Wall Design** command to print shear wall design data. If you select a few piers or spandrels before using this command then data is printed only for the selected elements.
2. If desired, revise the wall pier and/or spandrel overwrites, rerun the shear wall 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.

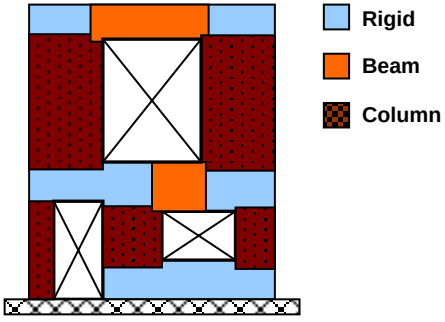
   Note that shear wall design is performed as an iterative process. You can change your wall design dimensions and reinforcing during the design process without rerunning the analysis. However, you always want to be sure that your final design is based on analysis properties (wall dimensions) that are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



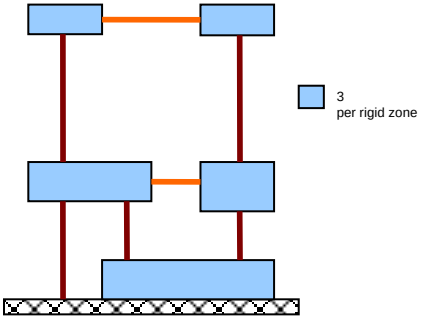
A: Shear Wall with Line Loads



B: Finite Element Model



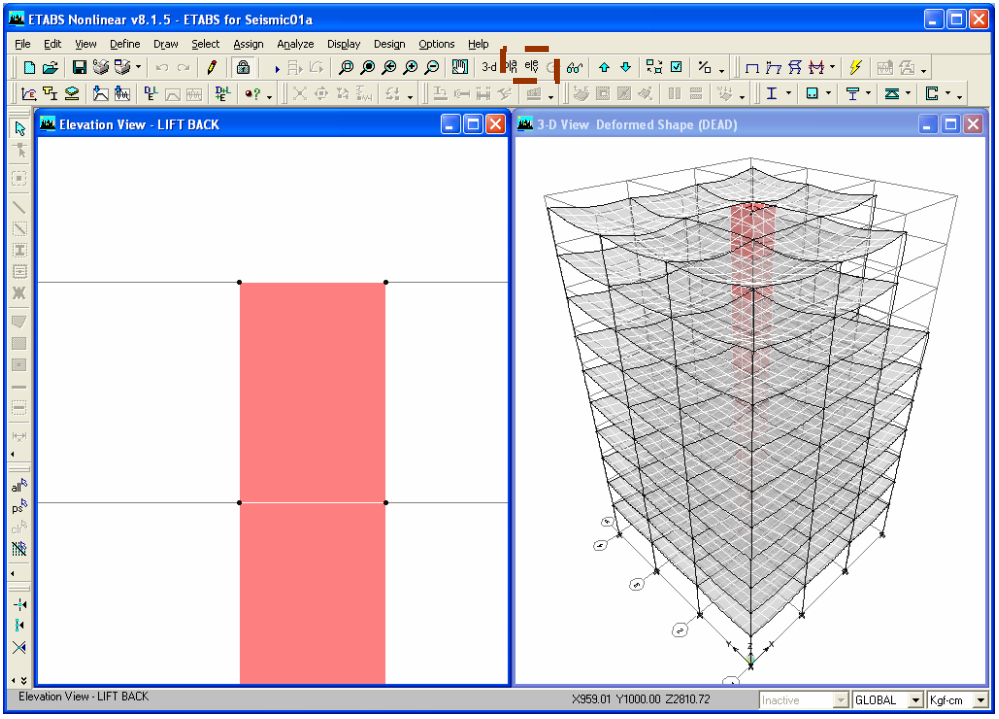
C: Define Beams & Columns

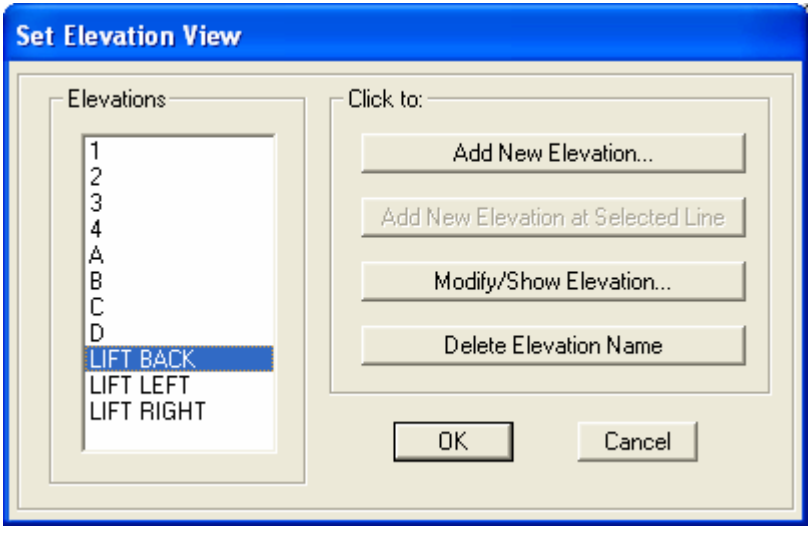


D: Beam-Column Model

Step 15-1: Change view to "Elevation View"

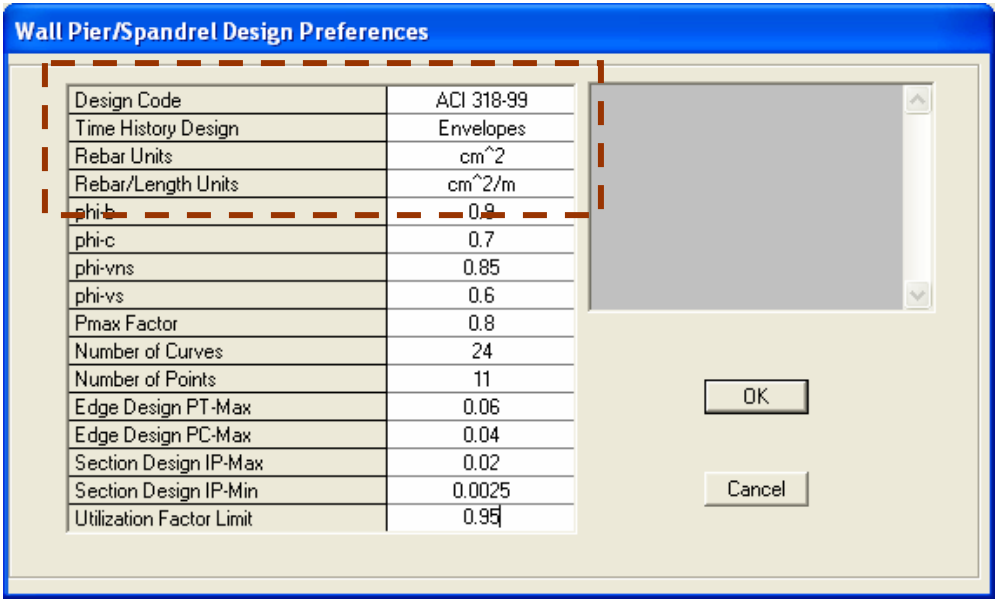
Click on *Set Elevation View* button, select desired elevation and use *Rubber Band Zoom* button to zoom shear wall view.





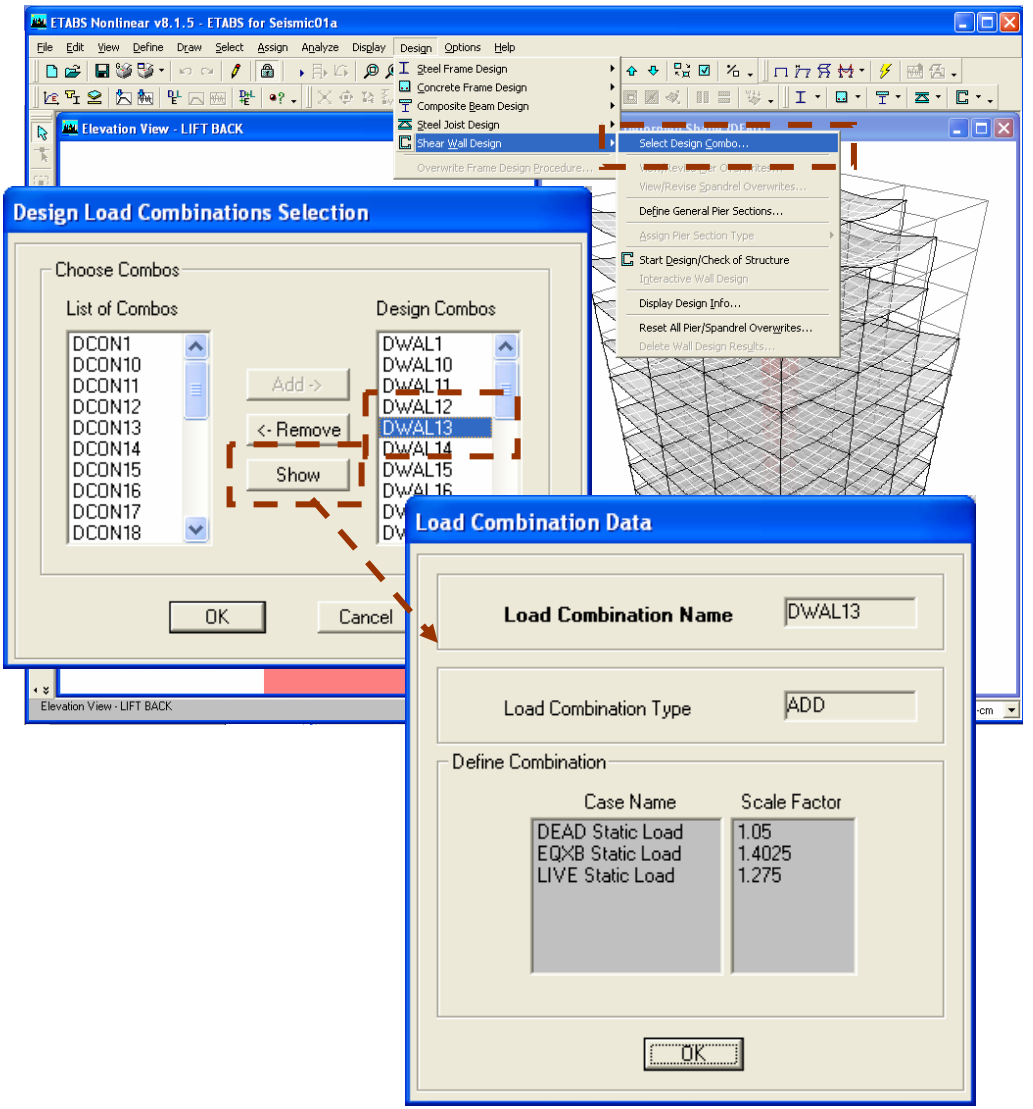
Step 15-2: Select Design Code for Shear Wall Design

Go to **Options >> Preference >> Shear Wall Design** and select parameters as shown in figure below.



Step 15-3: View Load Combination for Shear Wall Design

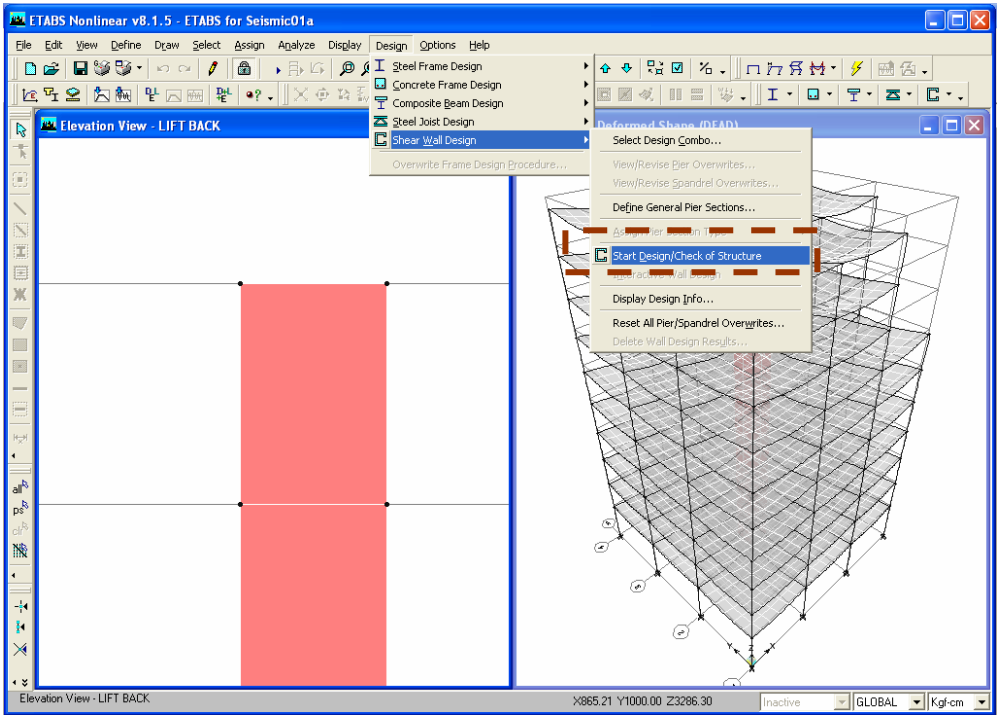
Go to **Design >> Shear Wall Design >> Select Design Combo** to view load combination for shear wall design. Load combinations have been defined as selected code from previous step. Select desired load combination from "Design Combos" column and click on "Show" to view load combination parameters (load factors and details)



Note: ETABS will define load combination automatically based on selected design from previous step.

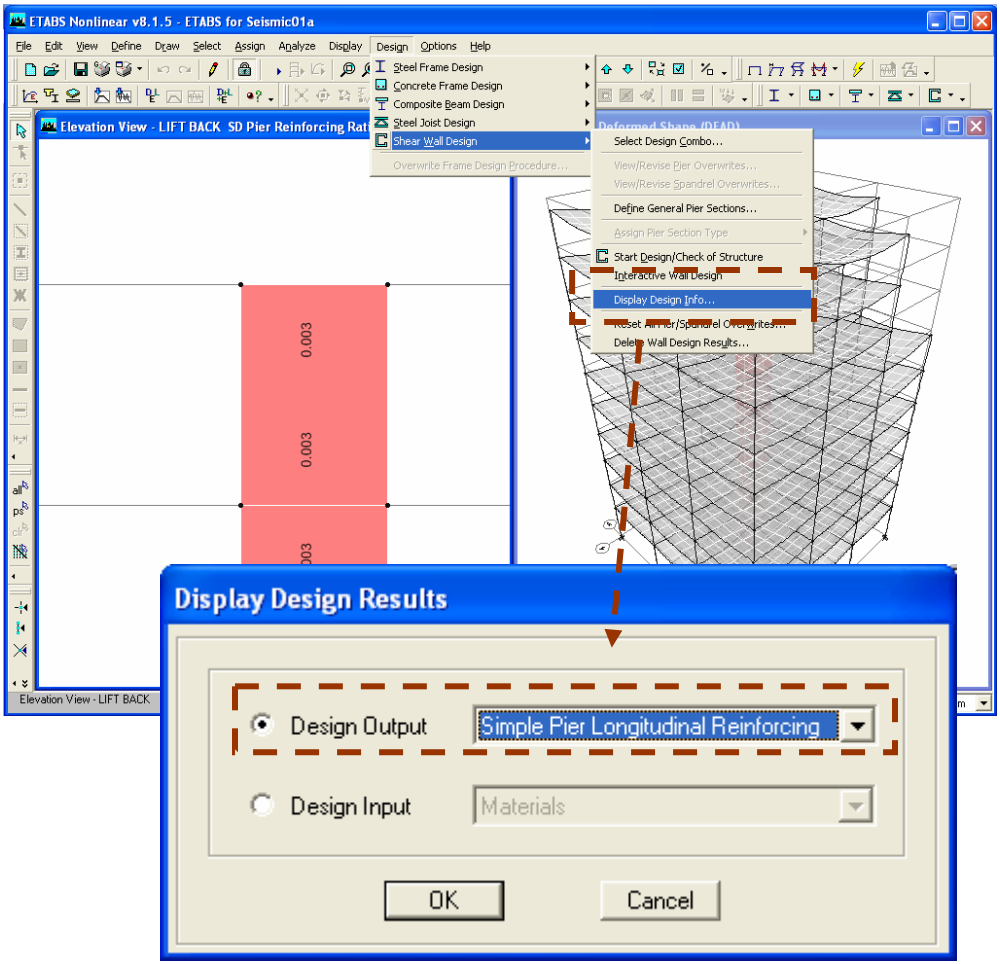
Step 15-4: Start Shear Wall Design

Go to **Design >> Shear Wall Design >> Start Design/Check Structure**



Step 15-5: Display Pier Design Information for Shear Wall Design

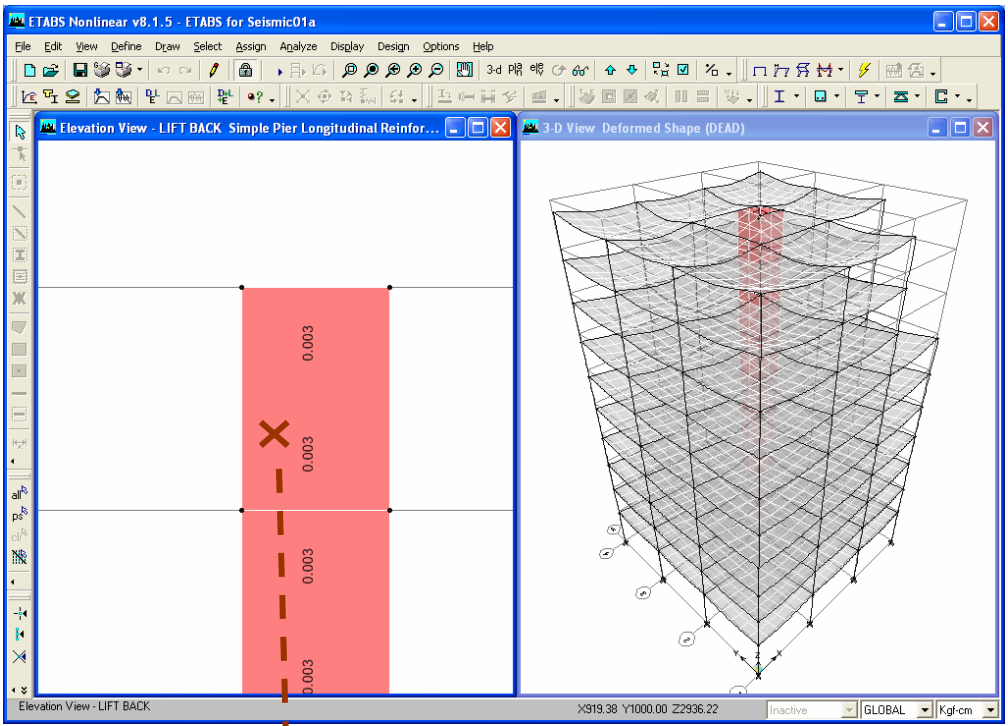
Go to **Design >> Shear Wall Design >> Display Design Info**, click on "Design Output" and select "Pier Longitudinal Reinforcing".



Note: Longitudinal reinforcing area displayed in above figure is for all 3 shear wall panels because all of them have been assigned in same pier label (P1) as specified in Step 4-5

Step 15-6: Display Pier Design Details for Shear Wall Design

To see pier design in details, right mouse click on desired pier panel.



Uniform Reinforcing Pier Section - Flexural Design (ACI 318-99)

Story ID: STORY10 Pier ID: P1 X Loc: 900 Y Loc: 933.3333 Units: Kgf-cm

Flexural Design for P-M2-M3 (RLLF = 0.786)

Station Location	Required Reinf Ratio	Current Reinf Ratio	Flexural Combo	Pu	M2u	M3u	Pier Ag
Top	0.0025	0.0085	DWAL26	30860.433	47242.666	697611.023	12000.036
Bottom	0.0025	0.0085	DWAL26	38646.153	-296411.501	7591063.467	12000.036

Shear Design - First Inadequate Leg or Leg Requiring Most Rebar per Unit Length

Station Location	Rebar cm^2/m	Shear Combo	Pu	Mu	Vu	Capacity phi Vc	Capacity phi Vn
Top Leg 1	5.000	DWAL18	12128.962	-369711.475	14327.102	29567.580	41087.580
Bot Leg 1	5.000	DWAL18	3864.629	2246934.643	12404.651	28575.860	40095.860

Boundary Element Check - First Inadequate Leg or Leg Requiring Longest Boundary Zone

Station Location	B-Zone Length	B-Zone Combo	Pu	Mu	Vu	Pu/Po
Top Leg 1	Not Needed	DWAL14	27626.869	-456846.979	-2089.517	0.0093
Bot Leg 1	Not Needed	DWAL14	49261.162	303761.354	-3445.744	0.0166

Reinforcement Location for Pier

Uniform Reinforcing Pier Section - Flexural Design (ACI 318-99)

Story ID: STORY10 Pier ID: P1 X Loc: 900 Y Loc: 933.3333 Units: Kgf-cm

Flexural Design for P-M2-M3 (RLLF = 0.786)

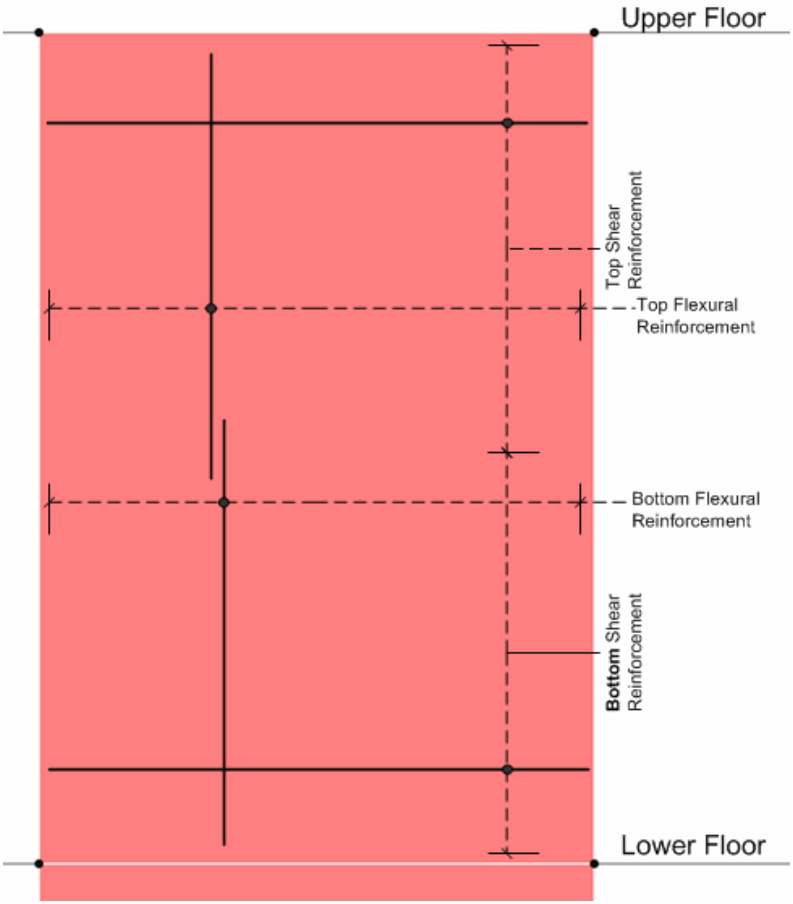
Station Location	Required Reinf Ratio	Current Reinf Ratio	Flexural Combo	Pu	M2u	M3u	Pier Ag
Top	0.0025	0.0085	DWAL26	30860.433	47242.666	697611.023	12000.036
Bottom	0.0025	0.0085	DWAL26	38646.153	-296411.501	7591063.467	12000.036

Shear Design - First Inadequate Leg or Leg Requiring Most Rebar per Unit Length

Station Location	Rebar cm^2/m	Shear Combo	Pu	Mu	Vu	Capacity phi Vc	Capacity phi Vn
Top Leg 1	5.000	DWAL18	12128.962	-369711.475	14327.102	29567.580	41087.580
Bot Leg 1	5.000	DWAL18	3864.629	2246934.643	12404.651	28575.860	40095.860

Boundary Element Check - First Inadequate Leg or Leg Requiring Longest Boundary Zone

Station Location	B-Zone Length	B-Zone Combo	Pu	Mu	Vu	Pu/Po
Top Leg 1	Not Needed	DWAL14	27626.869	-456846.979	-2089.517	0.0093
Bot Leg 1	Not Needed	DWAL14	49261.162	303761.354	-3445.744	0.0166



Note: Typical Detailing of Shear Wall