



**Andhra Pradesh State Skill  
Development Corporation**



# **Extended Three-Dimensional Analysis of Building System**

## **ETABS**

### **Response Spectrum Analysis**



## RESPONSE SPECTRUM ANALYSIS

### Objective

This chapter describes the step by step process of Response Spectrum Analysis of a Structure in ETABS.

### EXERCISE:

**Create a Model; perform Linear Static Seismic Analysis and Design with the following considerations**

- 10 Story Building – 3.0 m typical height – bottom story – 3m
- X-direction : 6 Spans – 3.2, 3.5, 5.3, 4.2, 4.5, 3.0 m
- Y –direction : 5 Spans – 5.0, 5.3, 3.6, 4.2, 3.1 m
- $F_{ck}$  – 30 MPa;  $F_y$  – 500 Mpa
- Beam Section – 300 x 450 mm
- Column Section – 450 x 450 mm
- Slab – 125 mm thick
- Additional floor load; Floor finish load –  $1.5 \text{ KN/M}^2$  ; Live load –  $2.5 \text{ KN/M}^2$
- Main Wall loads on beams (only on periphery) – 11.14 KN/M.
- Partition Wall loads (on intermediate beams) – 5.57 KN/M
- Mass source: Dead – 100%

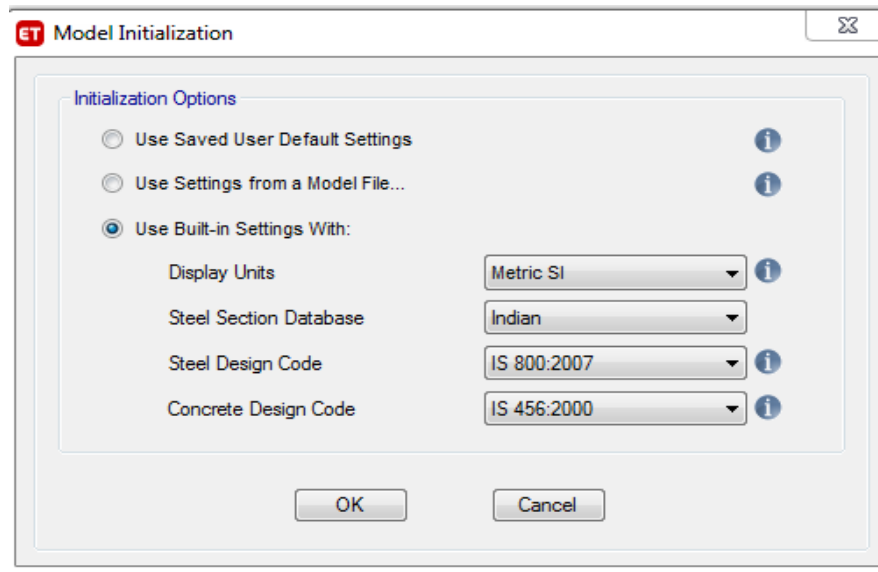
Live – 25%

Seismic:  $z = 0.16$ ,  $R = 5$ ,  $I = 1$ , Site type = I, Time period = Programme calculated

### Procedure:

#### Level-1: New Model Creation

1. Create a new model by using **Use Built in Settings with** option from the **Model Initialization** form as shown below and click on **OK**



**ET Model Initialization**

Initialization Options

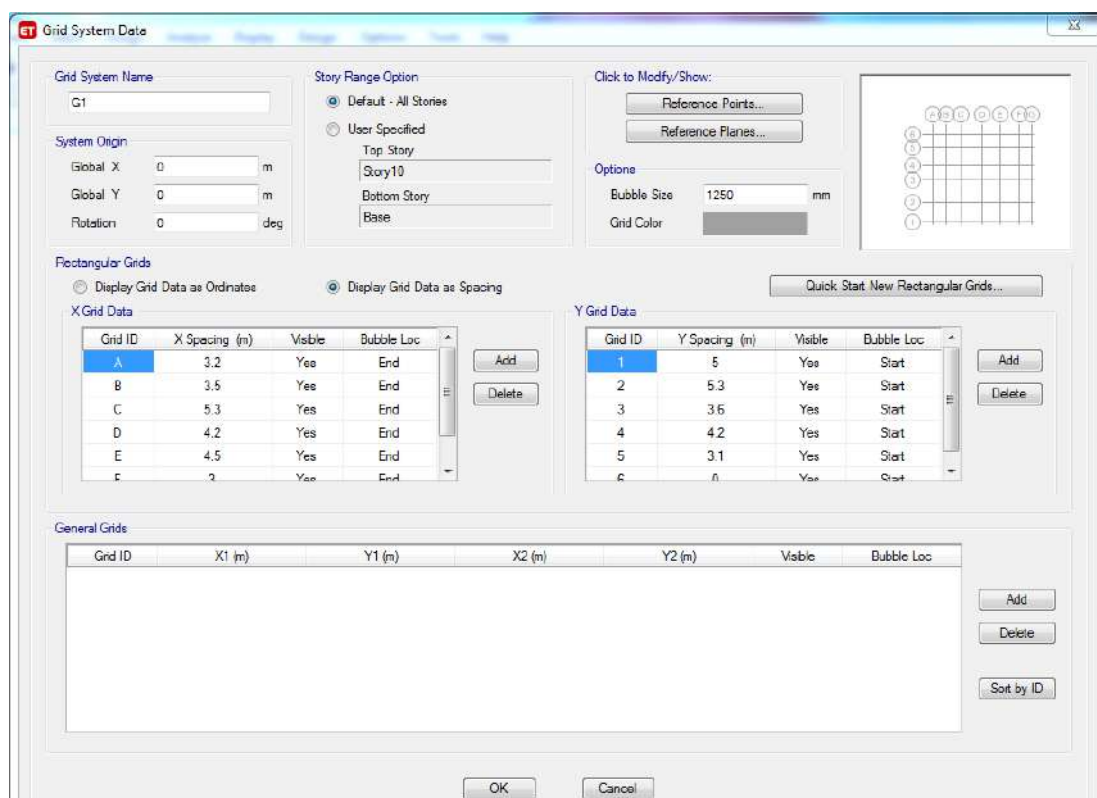
☐ Use Saved User Default Settings  
☐ Use Settings from a Model File...  
☒ Use Built-in Settings With:

Display Units: Metric SI  
 Steel Section Database: Indian  
 Steel Design Code: IS 800:2007  
 Concrete Design Code: IS 456:2000

OK Cancel

**Fig: Model Initialization form**

2. **New Model Quick Template:** In this step you have to specify the Grid Dimension, Story Dimensions and the Structural Template
3. Specify the grid dimensions as 7-grids along X-axis, 6- grid along Y-axis and then enter spacing's as per the example under **Custom Grid Spacing** by clicking on **Edit Grid Data** in the **New Model Quick Templates** form.
4. To add the grid click on **Add** and to delete the grid click on **Delete** buttons, enter the spacing's in the spacing column.



**ET Grid System Data**

Grid System Name: G1

System Origin: Global X: 0 m, Global Y: 0 m, Rotation: 0 deg

Story Range Option:
   
☒ Default - All Stories
   
☐ User Specified
   
Top Story: Story10
   
Bottom Story:
   
Base:

Click to Modify/Show:
   
Reference Points...
   
Reference Planes...

Options:
   
Bubble Size: 1250 mm
   
Grid Color:

Rectangular Grids:
   
☐ Display Grid Data as Ordinates
   
☒ Display Grid Data as Spacing

Quick Start New Rectangular Grids...

X Grid Data:

Grid ID	X Spacing (m)	Visible	Bubble Loc
A	3.2	Yes	End
B	3.5	Yes	End
C	5.3	Yes	End
D	4.2	Yes	End
E	4.5	Yes	End
F	2	Yes	End

Add Delete

Y Grid Data:

Grid ID	Y Spacing (m)	Visible	Bubble Loc
1	5	Yes	Start
2	5.3	Yes	Start
3	3.5	Yes	Start
4	4.2	Yes	Start
5	3.1	Yes	Start
6	0	Yes	Start

Add Delete

General Grids:

Grid ID	X1 (m)	Y1 (m)	X2 (m)	Y2 (m)	Visible	Bubble Loc

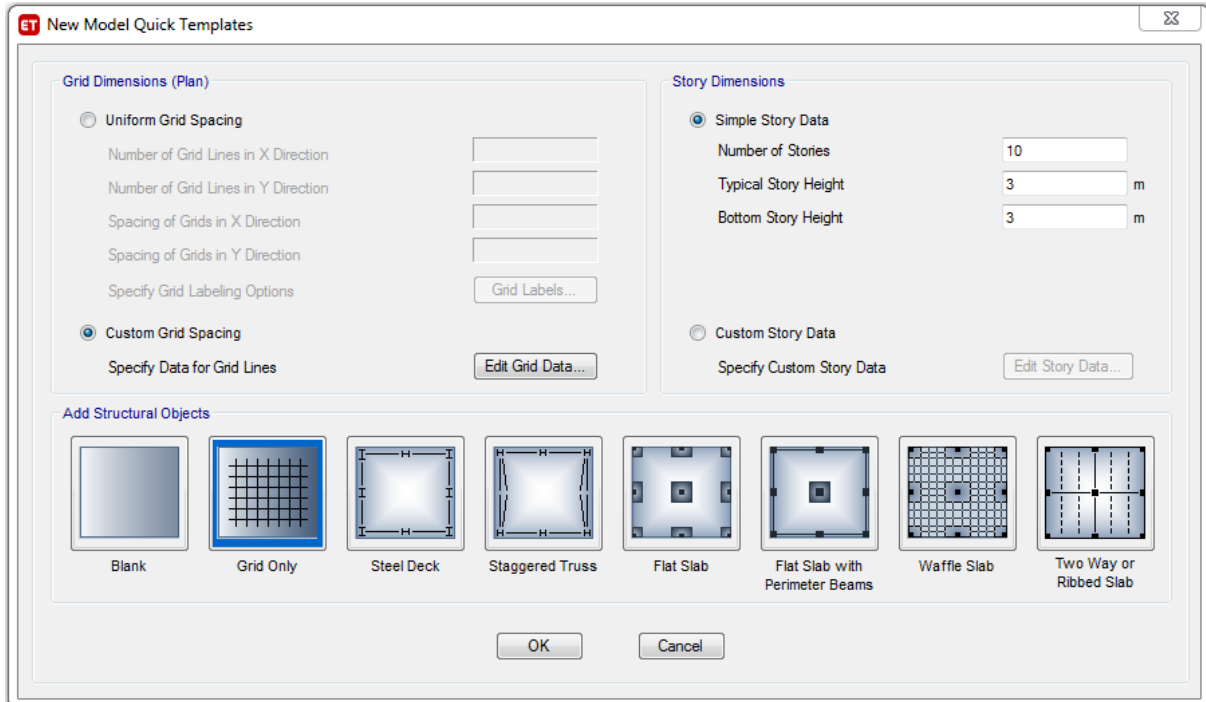
Add Delete Sort by ID

OK Cancel

**Fig: Grid System Data form**



5. Specify the number of stories as 10, Typical Story height and Bottom Story height as 3m under **Simple Story Data** in **New Model Quick Templates** form.
6. Select **Grid only** template from **Add Structural Template**.



The image shows the 'New Model Quick Templates' dialog box. It has three main sections: 'Grid Dimensions (Plan)', 'Story Dimensions', and 'Add Structural Objects'.

- Grid Dimensions (Plan):**
  - ☐ Uniform Grid Spacing: Includes input fields for 'Number of Grid Lines in X Direction', 'Number of Grid Lines in Y Direction', 'Spacing of Grids in X Direction', and 'Spacing of Grids in Y Direction'. There is a 'Grid Labels...' button.
  - ☒ Custom Grid Spacing: Includes a 'Specify Data for Grid Lines' button and an 'Edit Grid Data...' button.
- Story Dimensions:**
  - ☒ Simple Story Data: Includes input fields for 'Number of Stories' (10), 'Typical Story Height' (3 m), and 'Bottom Story Height' (3 m).
  - ☐ Custom Story Data: Includes a 'Specify Custom Story Data' button and an 'Edit Story Data...' button.
- Add Structural Objects:** A row of eight icons representing different structural templates: Blank, Grid Only (selected), Steel Deck, Staggered Truss, Flat Slab, Flat Slab with Perimeter Beams, Waffle Slab, and Two Way or Ribbed Slab.

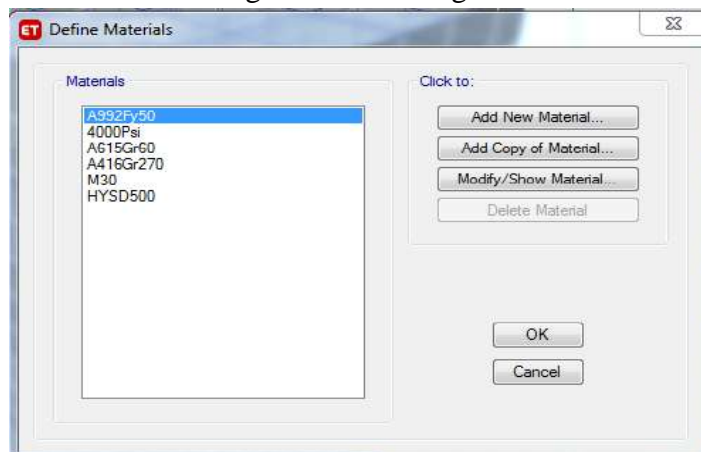
At the bottom are 'OK' and 'Cancel' buttons.

**Fig: New Model Quick Templates form**

7. By keeping remaining settings as default click **Ok**.

## Level-2: Defining Materials

Go to **Define** menu > **Material Properties**, Click on **Add New Material** option and add M30 grade concrete and HYSD500 grade rebar using **Add New Material Property** form.



The image shows the 'Define Materials' dialog box. It has two main sections: 'Materials' and 'Click to:'.

- Materials:** A list box containing the following materials: A992Fy50, 4000Psi, A615Gr60, A416Gr270, M30, and HYSD500. 'A992Fy50' is currently selected.
- Click to:** A group of buttons: 'Add New Material...', 'Add Copy of Material...', 'Modify/Show Material...', and 'Delete Material'.

At the bottom are 'OK' and 'Cancel' buttons.

**Fig: Define Materials form**

### Level- 3: Defining Structural elements

1. Go to **Define menu >Section Properties > Frame Sections**, click on **Add New Property** and specify the parameters in **Frame Section Property Data** form to create the sections as follows

Beam Sections – 300 x 450 mm

Column Section – 450 x450 mm

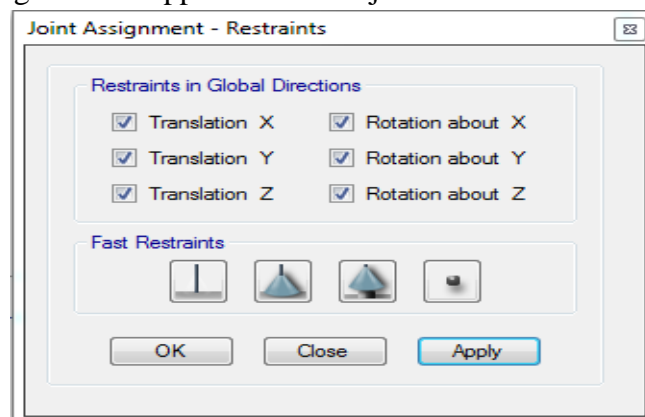
2. Click on **Modify/Show Rebar** to Specify the design type, rebar materials & cover as per the requirements
3. Click the **Define menu > Section Properties > Slab Sections** command to access the **Slab Properties** form. Click on **Add New Property** button to add a Slab of 125mm.

### Level- 4: Assigning Structural elements

Open Plan View of Story-1 > Keep the **Story Settings** as **All Stories** and assign the beam, columns & Slabs to grid by using either **Draw Beam /Column /brace (Plan, Elev, 3D)** or **Quick Draw Beams/columns (Plan,Elev,3D)** tool from **Draw** menu and **Quick Draw Floor/Wall** option

### Level-5: LOAD ASSIGNMENTS & SUPPORTS

1. Open Plan View of Story-1 > Keep the **Story Settings** as **All Stories** and Select all beams in Story-1 and assign **UDL** of 11.14 KN/M (main Wall load on beams only on periphery) & Partition Wall load(on intermediate beams) of 5.57KN/M using **Frame Loads**. Assign the Slab loads (Floor finish load & Live load) using **Shell Loads**.
2. Modify the loads on terrace story.
3. To Assign Supports keep the story settings to **one story** >open plan view of **Base Story** and select the joints at base using windows selection, go to **Assign > Joints > Restraints**, and assign Fixed support to bottom joints.





## Define Load Patterns (EQ-X & EQ-Y)

Use the **Define menu > Load Patterns** command to add, modify, or delete load patterns in the model file. An unlimited number of load patterns can be defined. Typically, separate load patterns would be defined for dead load, live load, wind load, Earthquake Load, snow load, and so on. Loads that need to vary independently, for design purposes or because of how they are applied to the structure, should be defined as a separate load pattern. ETABS uses the type of load pattern to create design load combinations automatically

Click the **Define menu > Load Patterns** command to access the Define Load Patterns form. This form has the following edit boxes and drop-down lists.

- Load** edit box. Use this edit box to specify the name of the load pattern. The previously assigned load pattern names appear in the display list below this edit box. Note that each load pattern and load combination must have a unique name. Also, the word Mode is reserved and cannot be used for a load pattern name or a load combination name.
- Type** drop-down list. The types of load patterns available appear in this drop-down list. The program uses these values when automatically creating design load combinations for the design postprocessors. The factors used in the design load combinations are different for the various types of loads. The load pattern types are assumed to be basically self-explanatory with the following further explanations:
- Reducible Live** option: This is a Reducible live load. A live load that is specified as reducible is reduced automatically by the program for use in the design postprocessors. The live load reduction parameters are specified using the **Design menu > Live Load Reduction Factors** command.
- Other** option: Use this option when specifying a pattern that does not fit into one of the other categories or when the pattern is not intended to be included in the design load combinations that are created automatically by the program.
- Self-Weight Multiplier** edit box. The self-weight of the structure is determined by multiplying the weight per unit volume of each object that has structural properties times the volume of the object. The weight per unit volume is specified in the material properties. Use the edit box to specify what portion of the self-weight is to be included in the load pattern. A self-weight multiplier of 1 means to include the full self-weight of the structure in the load pattern. A self-weight multiplier of 0.5 means to include half of the self-weight of the structure in the load pattern. **Normally a self-weight multiplier of 1 is specified in one load pattern only, usually the dead load pattern, with all other load patterns having a multiplier of zero.** Note that if a self-weight multiplier of 1 is specified in two different load patterns, and then those two load patterns are included in the same load combination, the results for the load combination are based on an analysis where double the self-weight of the building has been applied as a load.
- Auto Lateral Load** drop-down list. The *Auto Lateral Load* drop-down list becomes active when the load type is specified as *Seismic* or *Wind*.

Selecting a code from this list specifies that code-compliant loads be created automatically for the load case. Another form will appear that can be used to



review and as necessary specify the appropriate parameters for the automatic load. (See the *Modify Lateral Load* bullet below for hyperlinks to topics addressing those forms.)

When *None* is selected from the drop-down list, no automatic loads are used, and loads must be assigned using the commands available on the Assign menu.

**Note:** If a model has more than one tower, do not use a Quake type Auto Lateral Load, but perform a Response Spectrum or Time History analysis instead. Using a seismic Auto Lateral Load with multiple towers likely will result in an incorrect distribution of lateral loads.

Use the following buttons to add, modify, or delete load cases:

- **Add New Load** button.
  1. Type the name of the load pattern in the *Load* edit list.
  2. Select a load type from the *Type* drop-down list.
  3. Specify a self-weight multiplier in the *Self-Weight Multiplier* edit box. (See the *Self-Weight Multiplier* text above for cautions related to applying self-weight.)
    - If the load *Type* specified is *Quake* or *Wind*, select an option from the *Auto Lateral Load* drop-down list.
  4. Click the Add New Load button.
    - If the load type specified is *Seismic* or *Wind*, note the text for the **Modify Lateral Load** button (see below).
- **Modify Load** button.
  1. Highlight an existing load pattern in the *Loads* area. Note that the data associated with that load pattern appears in the edit boxes and drop-down lists at the top of the *Loads* area.
  2. Modify any of the data in the *Loads* area for the load pattern.
  3. Click the **Modify Load** button.
- **Modify Lateral Load** button. If a new load is being defined with a load Type of *Seismic* or *Wind* and a code (i.e., automatic load) has been selected in the *Auto Lateral Load* drop-down list, first click the Add New Load button, then click the Modify Lateral Load button to access a form specific to wind or seismic load. Use the form to specify parameters consistent with the code selected in the *Auto Lateral Load* drop-down list. The options available on the form relate to code requirements. Thus, fill in parameters on the form as appropriate. Then click the OK button to complete the operation and return to the Define Load Patterns form. Note that selecting *User Defined* for the *Auto Lateral Load* will access the User {Seismic, Wind} Loads on Diaphragms form, respectively.

**Note:** After a code specific load has been defined as described here, highlight it in the list of loads and click the Modify Lateral Load button to redisplay the form if any additional changes are needed.

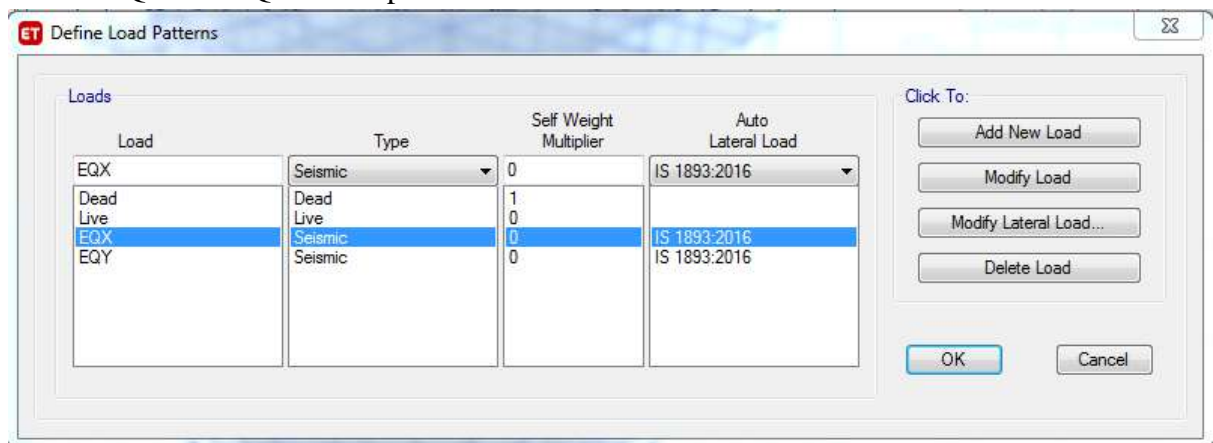
**Note:** Each automatic static lateral load must be in a separate load pattern. That is, two or more automatic static lateral loads cannot be specified in the same load pattern. However, additional user-defined loads can be added to a load pattern that includes an automatic static lateral load. A separate automatic static load pattern must be defined for each direction of wind load.

## • Delete Load

Note that when a load pattern is deleted, all of the loads that have been assigned to the model as a part of that pattern are also deleted.

- Highlight the existing load pattern in the *Loads* area.
- Click the **Delete Load** button.

## 4. Define EQ- X & EQ-Y Load patterns

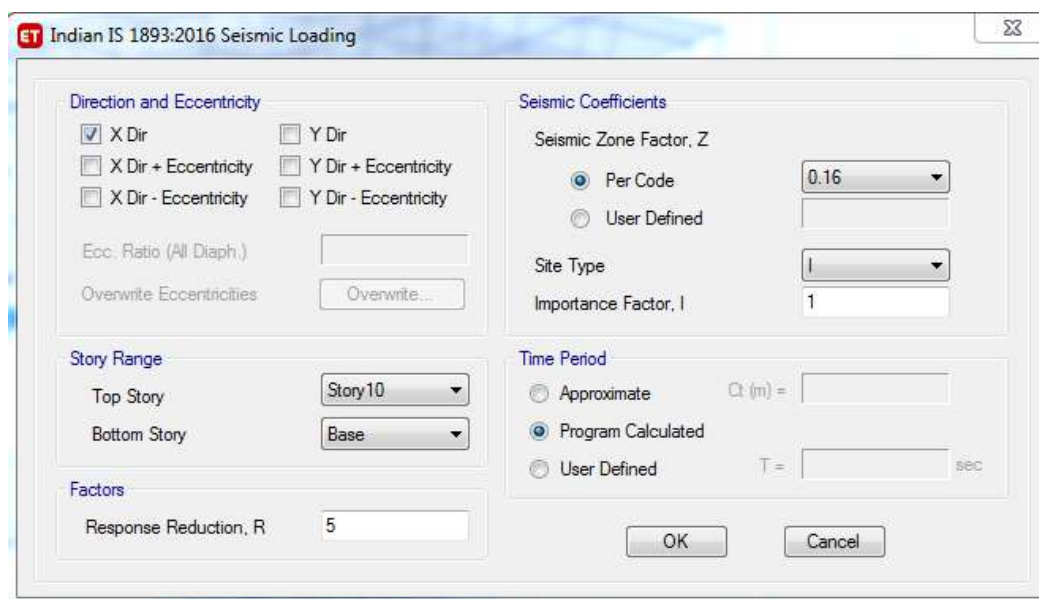


The 'Define Load Patterns' dialog box shows a table with columns: Load, Type, Self Weight Multiplier, and Auto Lateral Load. The 'EQX' load is highlighted. To the right, there are buttons for 'Add New Load', 'Modify Load', 'Modify Lateral Load...', and 'Delete Load'. At the bottom are 'OK' and 'Cancel' buttons.

Load	Type	Self Weight Multiplier	Auto Lateral Load
EQX	Seismic	0	IS 1893:2016
Dead	Dead	1	
Live	Live	0	
EQX	Seismic	0	IS 1893:2016
EQY	Seismic	0	IS 1893:2016

**Fig: Define Load Patterns form**

## 5. Click on **Modify Lateral Load** for EQ-X and change the parameters as per the requirement



The 'Indian IS 1893:2016 Seismic Loading' dialog box is divided into several sections: 'Direction and Eccentricity', 'Seismic Coefficients', 'Story Range', 'Time Period', and 'Factors'. The 'Seismic Coefficients' section shows 'Seismic Zone Factor, Z' set to 0.16, 'Site Type' set to I, and 'Importance Factor, I' set to 1. The 'Time Period' section shows 'Program Calculated' selected. The 'Factors' section shows 'Response Reduction, R' set to 5. At the bottom are 'OK' and 'Cancel' buttons.

**Fig: Modify Lateral Load form**





6. Select Direction and Eccentricity, Specify the Seismic Coefficients, Time Period as per Code i.e. IS1893 2016
7. Similarly modify the EQ-Y Load pattern.

## Mass Source

Use the **Define menu > Mass Source** command to access the **Define Mass Source** form. Choose to define the source of the mass of a building as from self-mass and additional mass, from loads, or from a combination of self-mass, additional mass, and loads. Also specify how lateral mass is to be considered and located.

This are of the form has the following options:

- **Element Self Mass** option. Each structural element has a material property associated with it; one of the items specified in the material properties is a mass per unit volume. When this checkbox is checked, ETABS determines the building mass associated with the element mass by multiplying the volume of each structural element times its specified mass per unit volume. This is the default. Also, if the link properties have been assigned to a joint or frame object, ETABS also includes the mass and weight as specified in the link property definition (see Link Properties)
- **Additional Mass** option. It is also possible to assign additional mass to account for partitions, cladding, and so forth. (See Additional Area Mass, Additional Line Mass, and Additional Point Mass.) The additional mass is applied to each joint in the structure on a tributary area basis in all three translational directions. Note that additional masses can be input as negative.
- **Specified Loads Patterns** option. Specify a load pattern combination that essentially defines the mass of the structure. The mass is equal to the weight defined by the load combination divided by the gravitational multiplier,  $g$ . Only the global Z-direction loads are considered when calculating the mass. This mass is applied to each joint in the structure on a tributary area basis in all three translational directions. Net downward loads acting on a joint are considered a positive mass. If the net load acting on a joint is upward, the mass at that joint is set to zero. Negative mass is not allowed in ETABS. By default, the mass is applied to each joint in the structure on a tributary area basis in all three translational directions.

## Mass Multipliers for Loads Patterns:

This area is active when the Specified Loads Patterns option is selected. The mass source load combination is created by specifying one or more load patterns, each with an associated multiplier.

- **Add:** Add a load pattern to the mass source load combination definition as follows.
  1. Select the load pattern name from the *Load* drop-down list.
  2. Type in an appropriate multiplier in the *Multiplier* edit box.
  3. Click the Add button.
- **Modify:** Modify the mass source load combination definition as follows.

1. Highlight the load pattern name. Note that the load pattern name and associated multiplier appear in the *Load Pattern* drop-down list and *Multiplier* edit box.
2. Select a different load, or type in the revised scale factor in the *Multiplier* edit box.
3. Click the Modify button.
- **Delete:** Delete a load pattern from the mass source load combination definition as follows.
  1. Highlight the load pattern name.
  2. Click the Delete button.

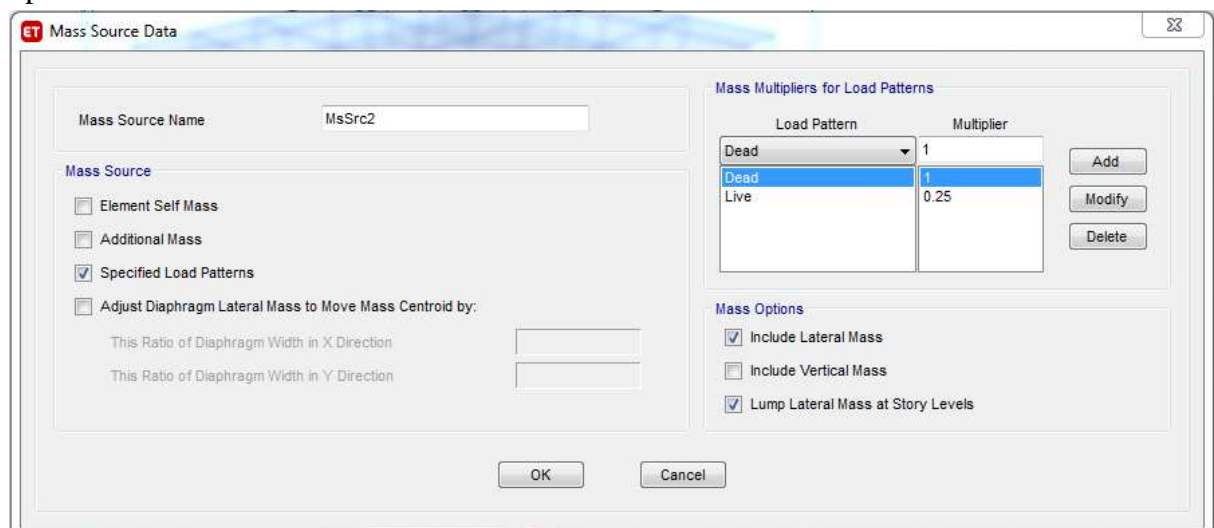
## Mass Options:

This area is of the form have the following check boxes:

**Include Lateral Mass** check box. If this checkbox is checked, assigned translational mass in the global X and Y axes directions and assigned rotational mass moments of inertia about the global Z-axis are considered in the analysis.

**Include Vertical Mass** check box. If this checkbox is checked, assigned translational mass in the global Z axis direction and assigned rotational mass moments of inertia about the global X and Y axes are considered in the analysis. Leave this box unchecked if you do not want to consider vertical dynamics.

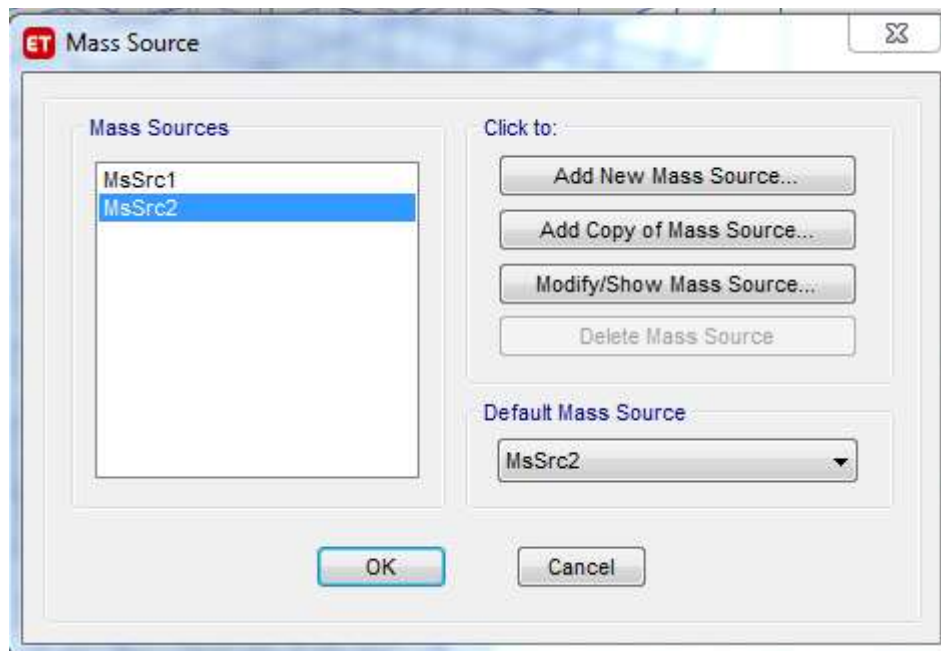
**Lump Lateral Mass at Story Levels** check box. As the name suggests, this option "moves" lateral mass that may occur between story levels to the nearest story level during analysis. This option can be used in conjunction with the default (i.e., **Element Self Mass** option for which the total mass is applied to each joint in the structure on a tributary area basis in all three translational directions) and with the **Include Lateral Mass** option.



Load Pattern	Multiplier
Dead	1
Live	0.25

**Fig: Mass Source Data form**

8. Click on Add New Mass Source , Select Specified Load Patterns under Mass Source and Specify Mass Multipliers for Load patterns as per IS 1893-2016 i.e.. Dead as 1 & Live as 0.25
9. Make the new Mass Source MsSrc2 as Default Mass Source



**Fig: Mass Source form**

## Define Response Spectrum Functions

Use the **Define menu > Functions > Response Spectrum** command to access the **Define Response Spectrum Functions** form and add, modify, or delete a response function from a text file, using code-specific parameters, or based on user specified parameters.

A response spectrum function is a list of period versus spectral acceleration values.

**Note:** In ETABS the acceleration values in the function are assumed to be normalized; that is, the functions themselves are not assumed to have units. Instead, the units are associated with a scale factor that multiplies the function and is specified when the response spectrum case is defined.

10. Click the **Define menu > Functions > Response Spectrum** command to access the Define Response Spectrum Functions form.
11. Use the *Response Spectra* display area, the drop-down list, and the buttons as appropriate to add, modify, or delete a response spectrum function. The *Response Spectra* display area lists all of the response spectrum functions currently defined in the model file.
  - a. **Add a New Function** button. Choose the type of function from the "Choose Function Type to Add" drop-down list. Then click the Add New Function button.

## Response Spectrum Function - IS1893:2002

The IS: 1893 response spectrum function is based on Figure 2 in section 6.4.5 of the IS: 1893 code. The digitization of these response spectra is based on section 6.4.5.

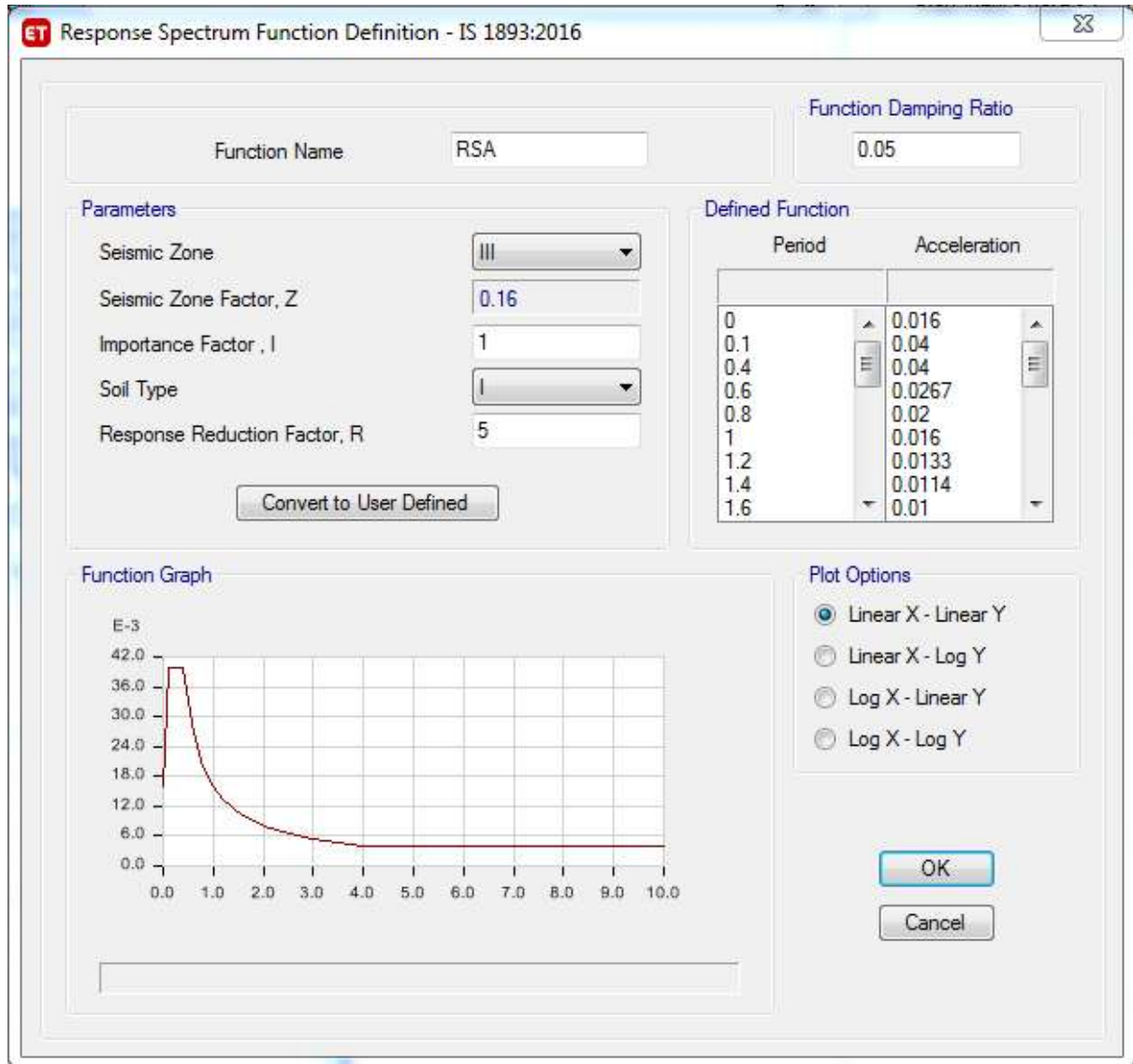
The parameters required are a seismic zone factor  $Z$ , soil type and the damping ratio of the building structure. These values can be selected from relevant sections of the IS: 1893 code.



Access the Response Spectrum Function Definition - IS1893:2002 form as follows:

1. Click the **Define menu > Functions > Response Spectrum** command to access the Define Response Spectrum Functions form. Then do one of the following:
    - Select IS1893:2002 from the *Choose Function Type to Add* drop-down list; click the Add New Function button.
    - Highlight an existing IS1893:2002 function definition in the Response Spectra display area and click the Modify/Show Spectrum button.
  2. **Function Name** edit box. Use the default or specify a new name for the response spectrum function.
  3. **Function Damping Ratio** edit box . Use the default or specify the damping ratio that will be used to generate the response spectrum curve. During analysis, the response spectrum curve will be automatically adjusted from this value to the actual damping present in the model.
  4. **Defined Function / Function Points.** This area displays the period and spectral acceleration values for the function. They cannot be edited and are provided for review purposes only. These values update when changes are made to the response spectrum-specific parameters identified in the hyperlinked topics (above). If manual changes to the *Period* and *Acceleration* values are necessary, click the Convert to User-Defined button to convert the function to a user-defined function, which will make the values editable.
  5. **Function Graph** area. The function graph area displays a graph of the function. It updates automatically the spectrum parameters are defined. Run the mouse pointer over the function graph and a dot appears along the line representing the response spectrum. The coordinates of the dot are reported in the box just below the graph. Use the **Plot Options** to review the display of the function graph.
  6. **Display Graph** button. Some form will have this button. Click it to update the Function Graph if the graph does not automatically update as changes are made to the parameters.
- **Modify or Show a Response Spectrum Function Definition**
    1. Highlight the name of the function to be modified/shown in the *Response Spectra* area of the form.
    2. Click on the Modify/Show Spectrum button.
    3. The form that was used to define the function will appear. Use the form to make any changes or modifications. The hyperlinks provided above provide access to topics specific to each type of form.
  - **Delete a Response Spectrum Definition**
    1. Highlight the name of the function to be deleted in the *Response Spectra* area of the form.
    2. Click the Delete Spectrum button.





**Response Spectrum Function Definition - IS 1893:2016**

Function Name: RSA

Function Damping Ratio: 0.05

**Parameters**

Seismic Zone: III

Seismic Zone Factor, Z: 0.16

Importance Factor, I: 1

Soil Type: I

Response Reduction Factor, R: 5

**Defined Function**

Period	Acceleration
0	0.016
0.1	0.04
0.4	0.04
0.6	0.0267
0.8	0.02
1	0.016
1.2	0.0133
1.4	0.0114
1.6	0.01

**Function Graph**

E-3

42.0  
36.0  
30.0  
24.0  
18.0  
12.0  
6.0  
0.0

0.0 1.0 2.0 3.0 4.0 5.0 6.0 7.0 8.0 9.0 10.0

**Plot Options**

☒ Linear X - Linear Y

☐ Linear X - Log Y

☐ Log X - Linear Y

☐ Log X - Log Y

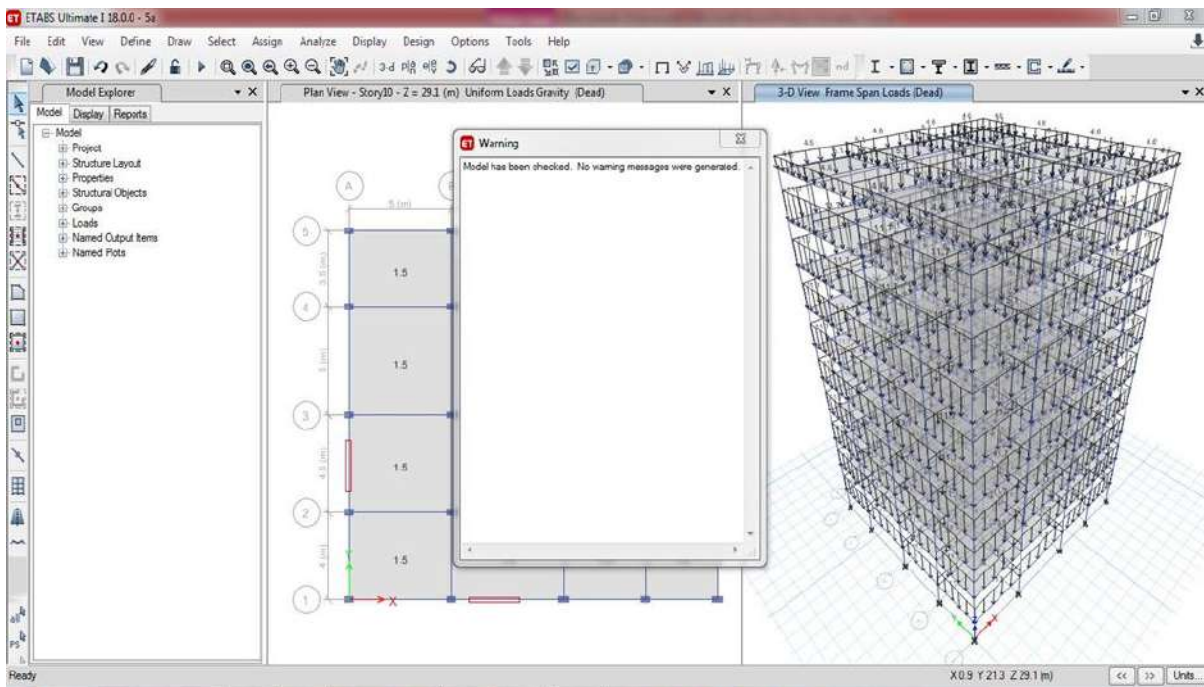
OK Cancel

**Fig: Response Spectrum Function Definition form**

12. Define the Load Cases for Spec-X and Spec-Y under Load Case option in Define Menu
13. To define the load combinations Go to **Define Menu> Load Combinations> Add Default Design Combos**, select **Concrete Frame Design** and click **OK**

## Level-6: Model Check

1. Perform **Model Check** by clicking on **Analyze** menu and select **Model Check** option from the dropdown list and select all the checks and click on **OK**.
2. As a result of **Check Model** you will receive a warning message stating the errors in model, if the model is free from errors and mistakes it display warning message as shown in below fig



## Level-7: Analysis

To perform analysis click on **Run Analysis** from the drop down list of **Analyze** menu. As the analysis completes it shows the deflection diagram initially.

## Level-8: Results Interpretation

1. To check the results like BMD or SFD click on **Display Frames/Piers/Spandrels/Links** or **F8**, select live load under load case, select Moment 3-3 or Shear 2-2 respectively and click on **OK**
2. To check the results of slabs **Display Shell Stresses/Forces** or **F9**

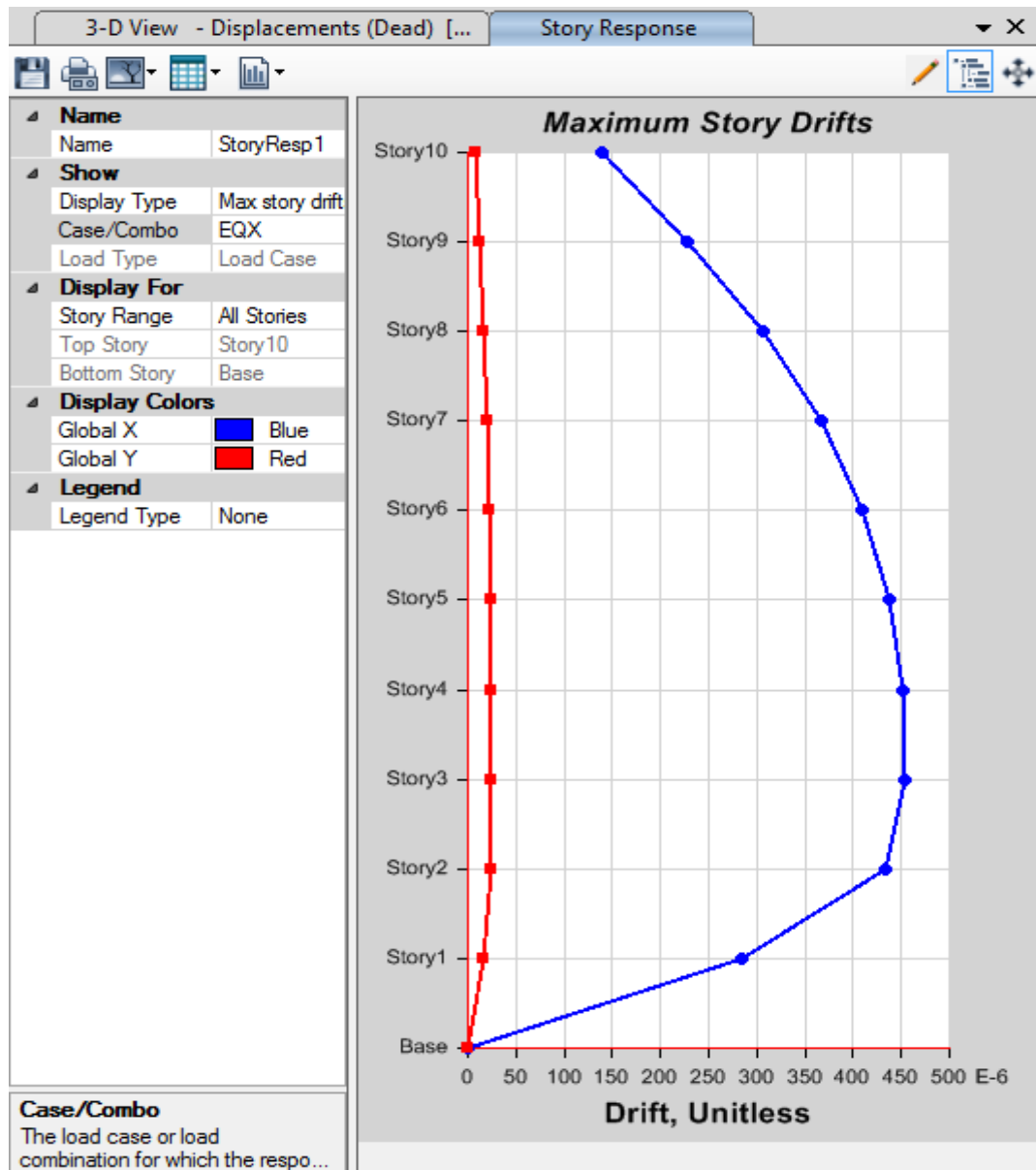
## Story Response Plots

Use the **Display menu > Story Response Plots** command to review and print vertical load, shear and story overturning plots using the Maximum Story Displacement form.

To view the form:

1. Define the load pattern and run the analysis.
2. Click the **Display menu > Story Response Plots** command. The form will display with a tab at the top, a tool bar, graphical plot and plot parameter options.
  - **Graphical Plot** area. The plot is shown in this area with appropriate labeling for the X and Y axes. When the mouse pointer is moved over the plot, the coordinates of the pointer location are shown below the plot.
  - **Plot Parameters** - Click on a parameter and an explanation of that parameter will display in the lower left-hand corner of the form. When changes are possible,

clicking in the value area will display a drop-down list of possible parameters or use the available edit boxes. If the text of the parameter name is partially hidden, use the cursor to drag the parameter area to make it larger, similar to resizing in Windows or Microsoft Excel. Similarly, if the text displays area in the lower left-hand corner is too small; drag it up to make it larger. The changes made to the parameters immediately update the graphical plot.



**Fig: Story Response Plot**

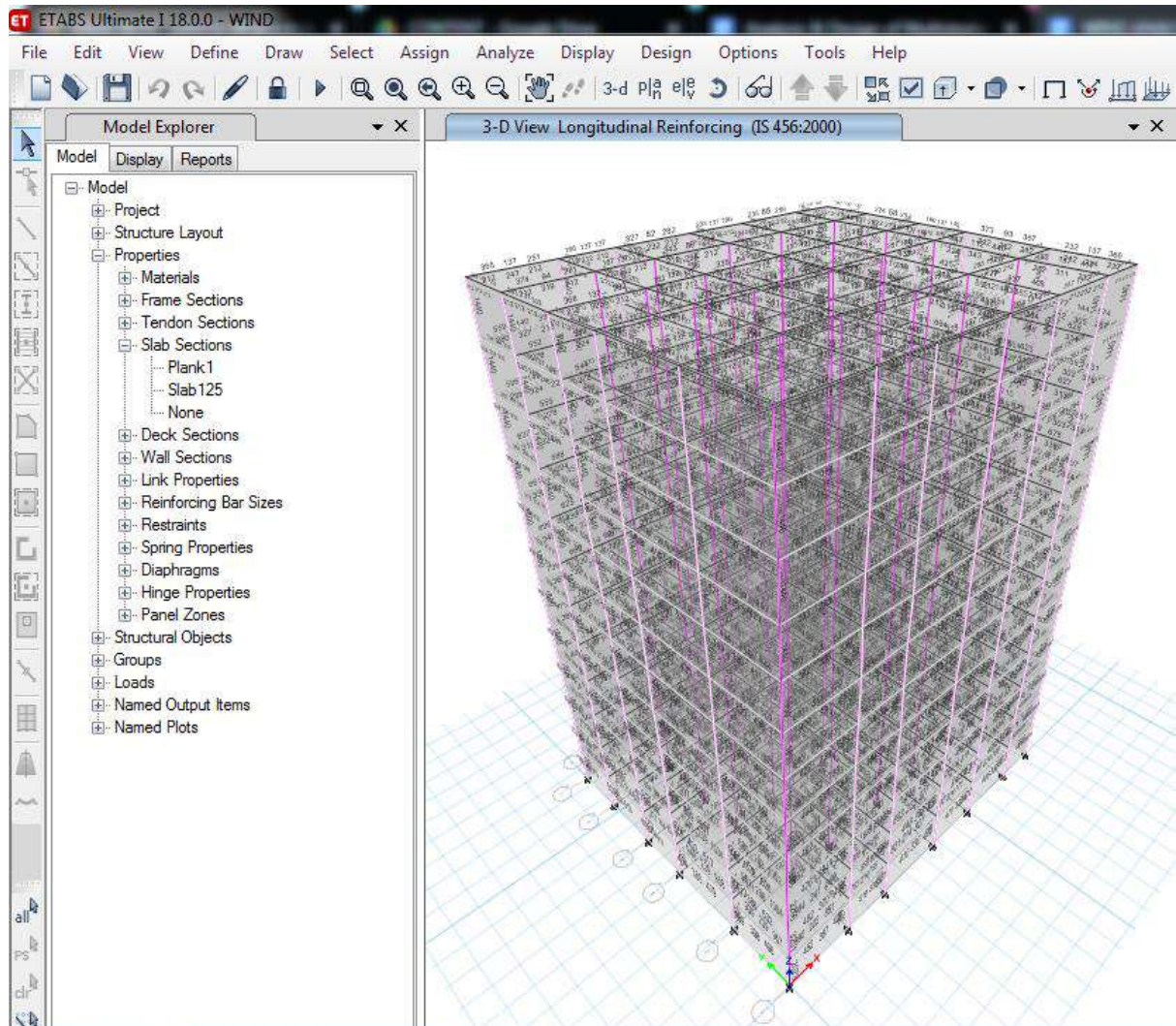
- To see the results like BMD, SFD & Deflection for any individual beam or column, select the frame and specify right click immediately it will display the form as shown below. Select the load case from the dropdown list for which you would like to view the results. And then click on the **Close** button to close the form.



## Level-9: Design Check

This step involves the design of structure for the obtained analysis results. As a result it shows the area of reinforcement required in the structural elements. And also it will detect the failure sections. Design can be performed for all the load cases and load combinations assigned to the structure.

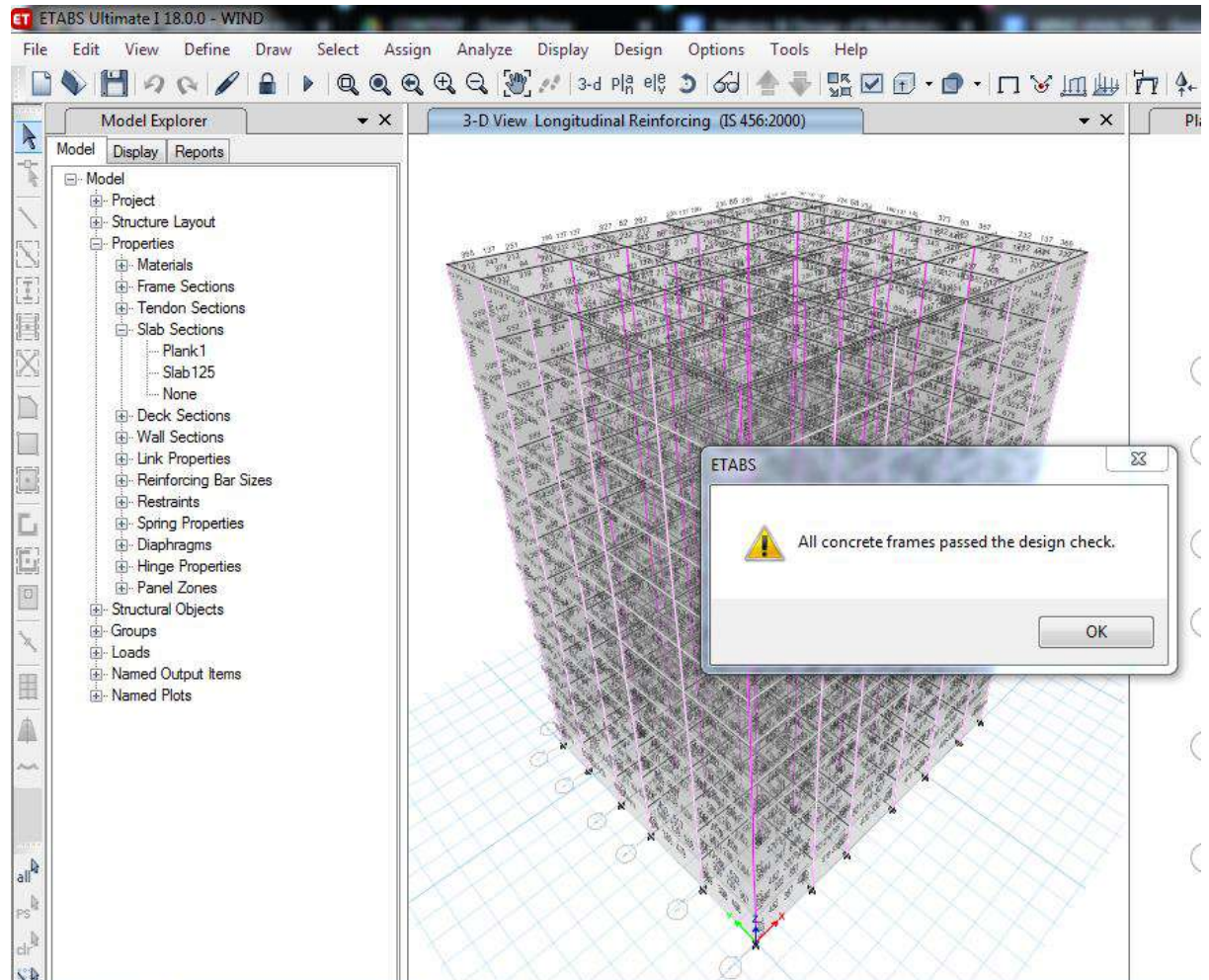
1. After analysis Go to **Design menu > Concrete Frame Design> View Preferences**, check the Design parameters.
2. Go to **Design menu > Concrete Frame Design > Start Design Check**



**Fig: Design Details**

3. To check the failure members **Design menu > Concrete Frame Design > Verify all members Passed**





**Fig: Design check**

4. To see the design details select the member and right click, it will display the design information form, to get detailed reports click on **Details** in the same form.