



# Andhra Pradesh State Skill Development Corporation



## Extended Three-Dimensional Analysis of Building System

# ETABS

### Analysis & Design of Multistoried Building

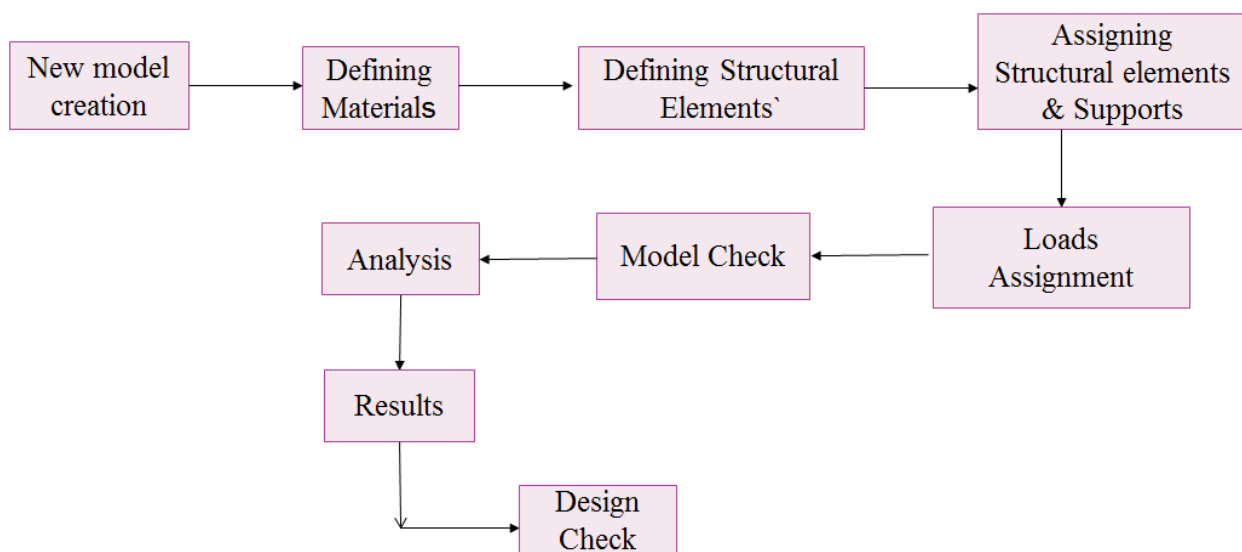
## Analysis & Design of Multistoried Building

### Objective

This chapter contains an explanation on Analysis & Design of Multistoried Building using ETABS.

### ETABS Workflow

To perform any analysis and design in ETABS a standard procedure has to be followed. The following flowchart represents the steps involved in ETABS for the analysis and design of structures.



### EXAMPLE

[Exercise Link](#)

### CONSIDERATIONS

#### Material Properties

Concrete: M30

Steel: HYSD500

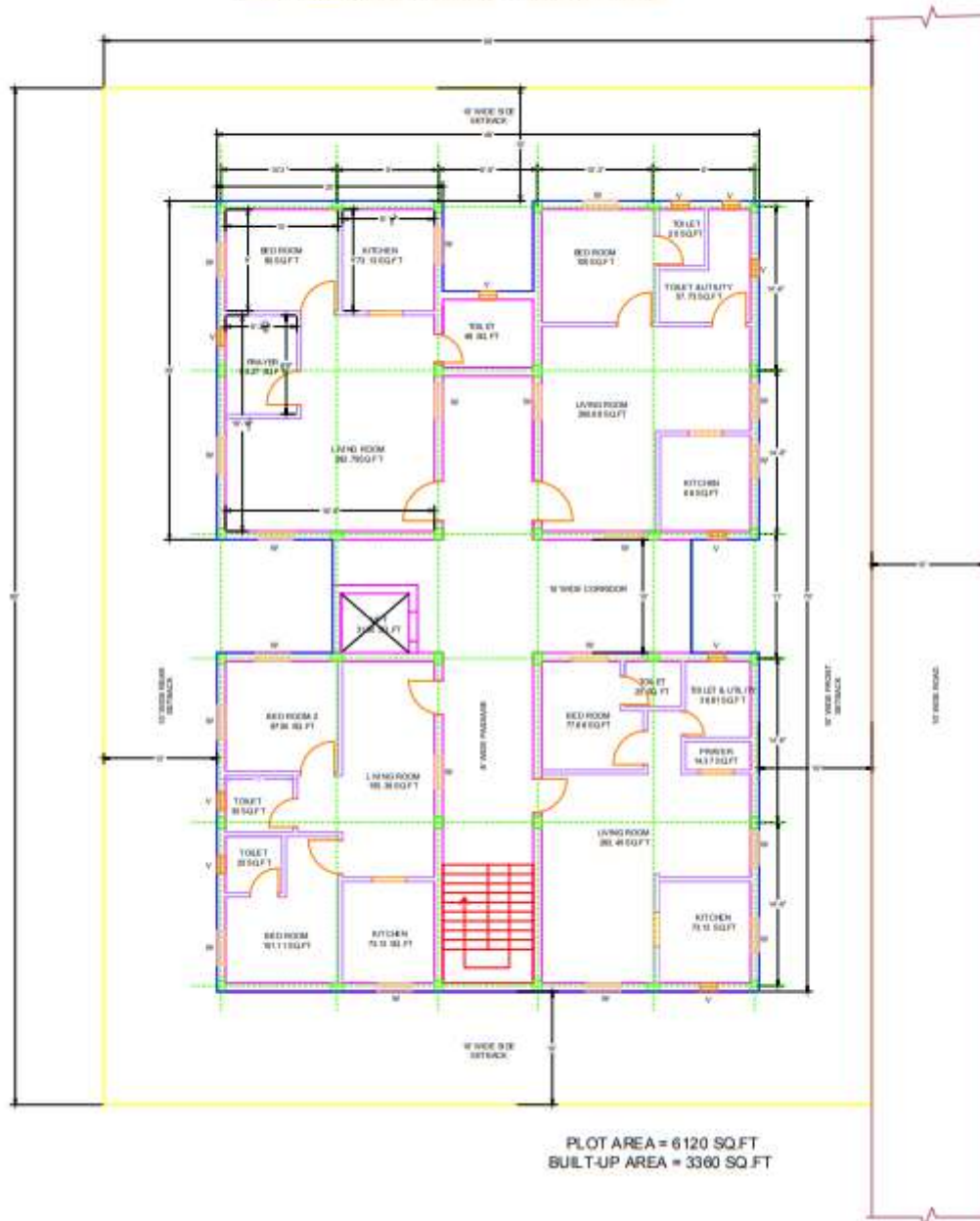
#### Section Property

Beam Section: 230mm X 350mm

Column Sections: 230mm X 300mm

Slab: 150mm

## TYPICAL FLOOR PLAN SHOWING DETAILS OF G+3 RESIDENTIAL BUILDING



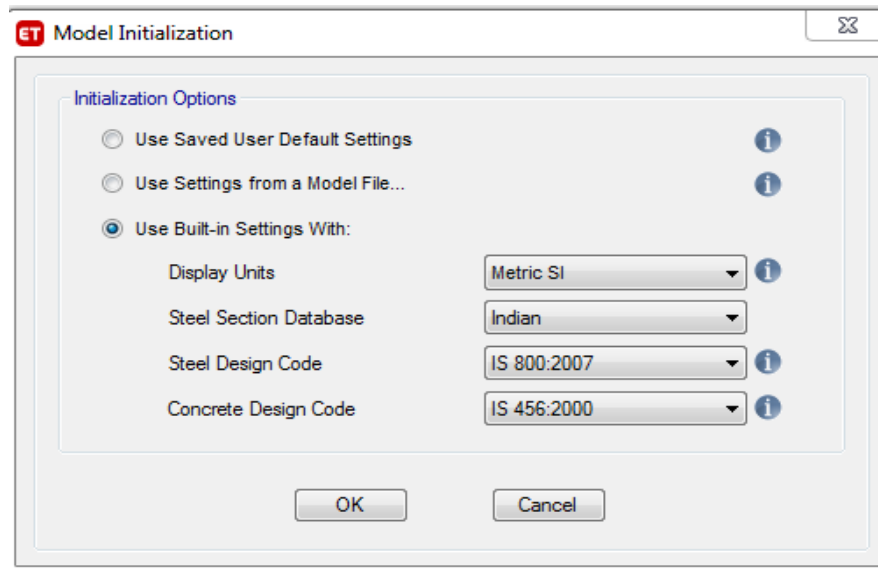
### Procedure

#### Level-1: New Model Creation

In this level you have to choose the method of opening new model and the unit's setup. This involves 2 steps

1. **Model Initialization:** 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**

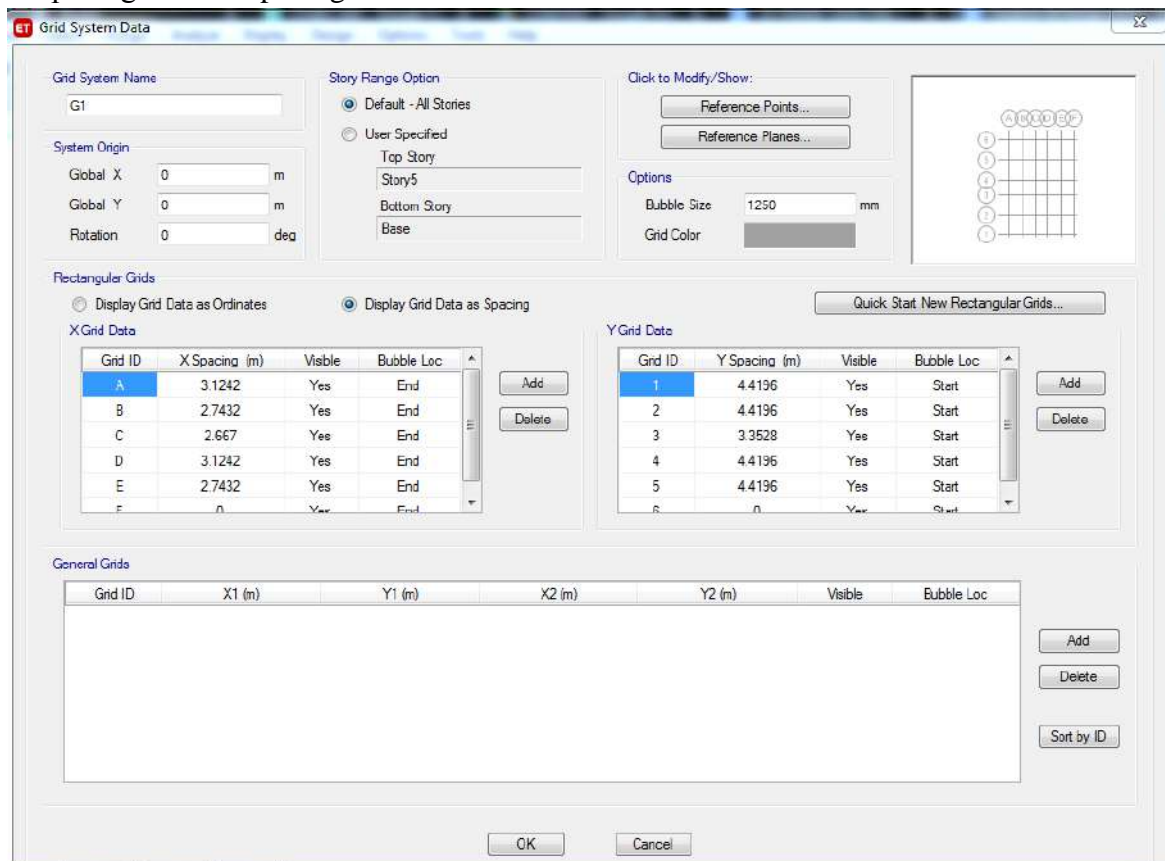
**Initialization Options**

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

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

**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 6-grids along X-axis, 6- grids 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: Story5  
     Bottom Story: Base

Click to Modify/Show:

Options  
 Bubble Size: 1250 mm  
 Grid Color:

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

**X Grid Data**

Grid ID	X Spacing (m)	Visible	Bubble Loc
A	3.1242	Yes	End
B	2.7432	Yes	End
C	2.667	Yes	End
D	3.1242	Yes	End
E	2.7432	Yes	End
F	0	Yes	End

**Y Grid Data**

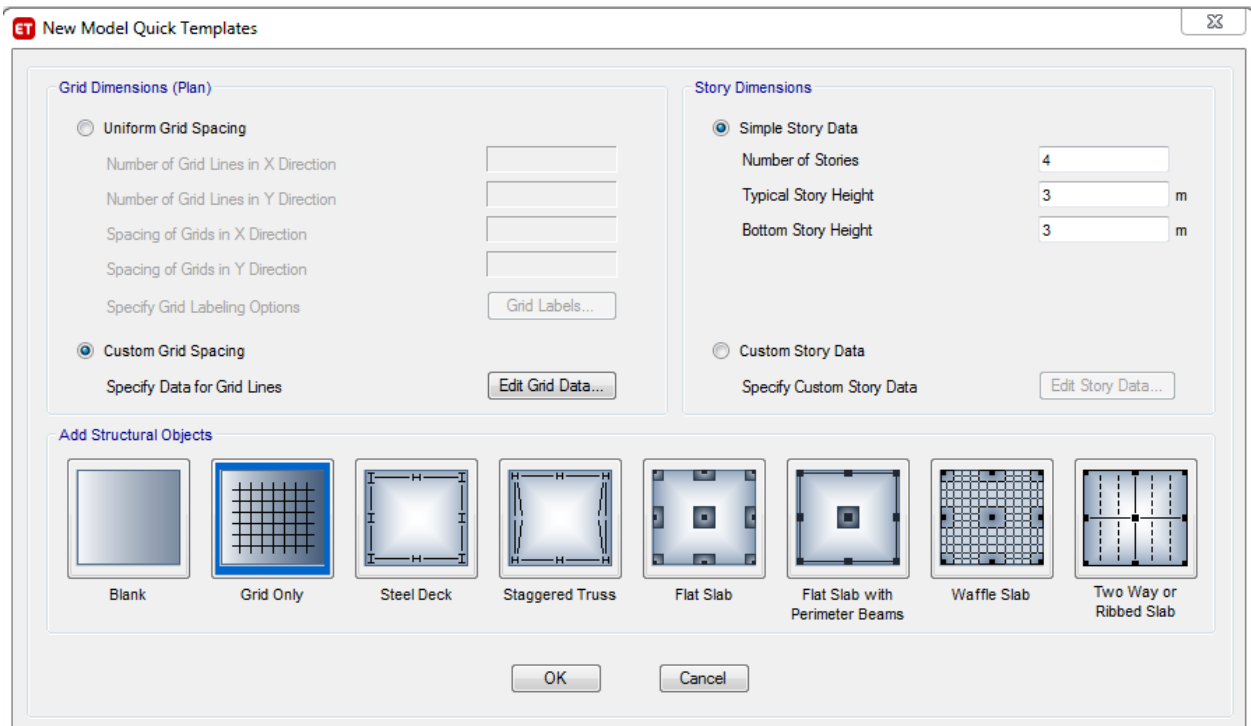
Grid ID	Y Spacing (m)	Visible	Bubble Loc
1	4.4196	Yes	Start
2	4.4196	Yes	Start
3	3.3528	Yes	Start
4	4.4196	Yes	Start
5	4.4196	Yes	Start
6	0	Yes	Start

**General Grids**

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

**Fig: Grid System Data form**

- Specify the number of stories as 4, Typical Story height and Bottom Story height as 3m under **Simple Story Data** in New Model Quick Templates form.
- Select **Grid only** template from **Add Structural Template**. By keeping remaining settings as default click **Ok**.



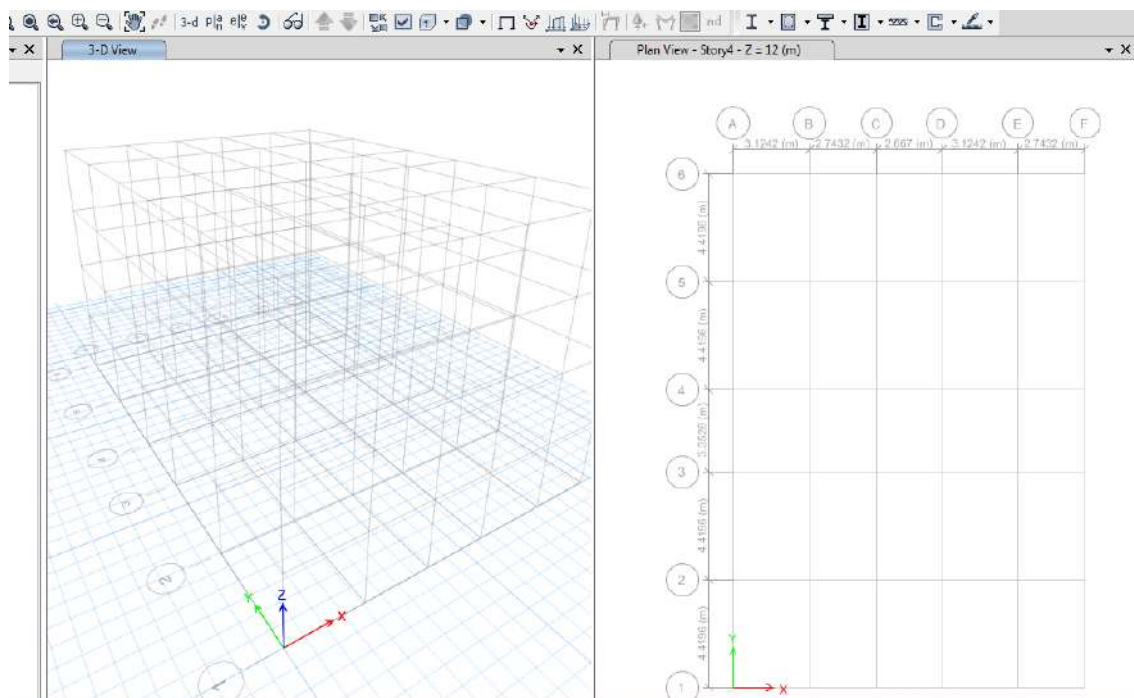
The dialog box is titled "New Model Quick Templates". It contains three main sections:

- 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. A "Grid Labels..." button is also present.
  - ☒ 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 (4), Typical Story Height (3 m), and Bottom Story Height (3 m). An "Edit Story Data..." button is also present.
  - ☐ Custom Story Data: Includes a "Specify Custom 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 Template Form**

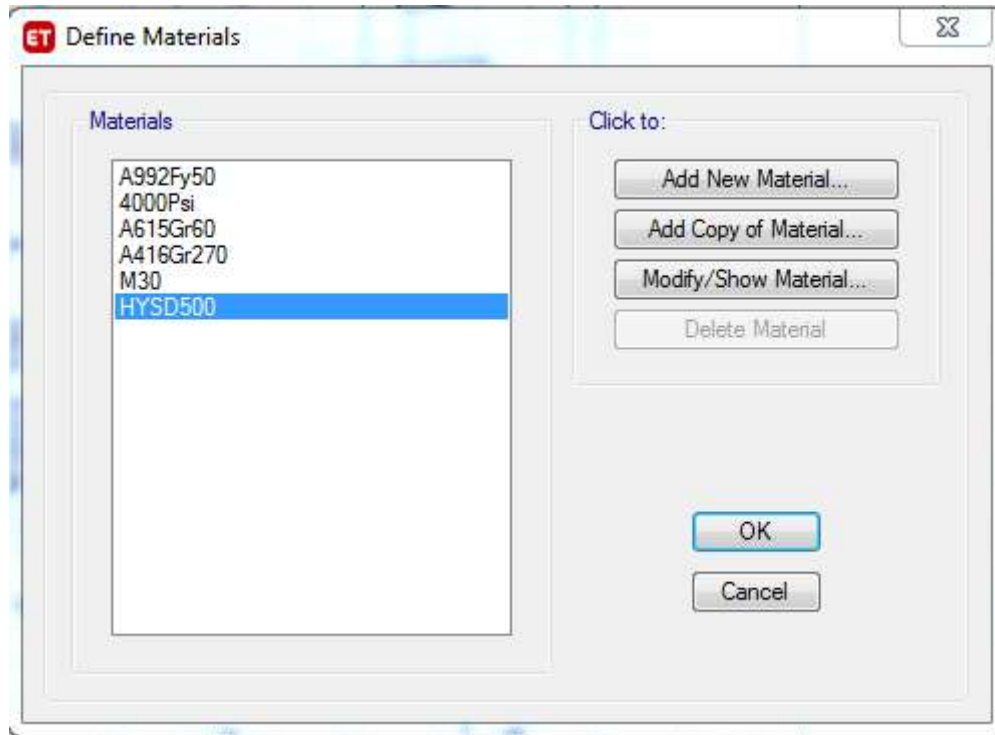
- As you specify grid dimensions and story dimensions, as a result you will get a 3D grid generated, which resembles the positions of columns and beams.



## Level-2: Defining Materials

In this step we will define the materials (conc & steel) as per the requirement i.e Concrete: M30 & Rebar: HYSD500

1. 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.



**Fig: Define Material Form**

## Level- 3: Defining Structural elements

In this step you have to create the sections for beams, columns and slab that you are going to use in the project.

To define beams and columns **Frame Sections** option is used, similarly to define slab sections **Slab Sections** option is used.

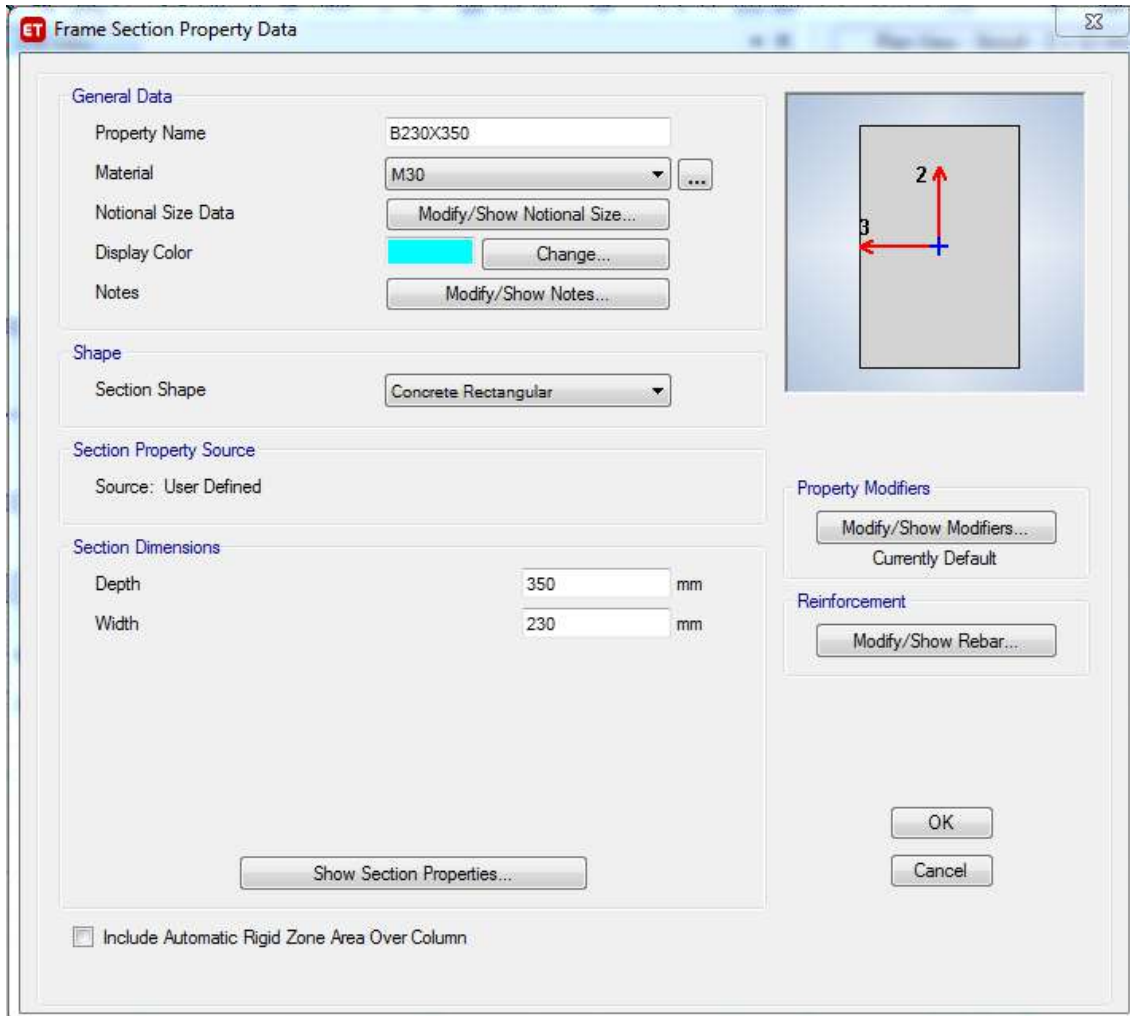
**Beam Section:** 230mm X 350mm

**Column Sections:** 230mm X 300mm

**Slab:** 150mm

### Procedure:

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



**ET Frame Section Property Data**

**General Data**

Property Name: B230X350

Material: M30

Notional Size Data: Modify/Show Notional Size...

Display Color: Change...

Notes: Modify/Show Notes...

**Shape**

Section Shape: Concrete Rectangular

**Section Property Source**

Source: User Defined

**Section Dimensions**

Depth: 350 mm

Width: 230 mm

**Property Modifiers**

Modify/Show Modifiers...  
Currently Default

**Reinforcement**

Modify/Show Rebar...

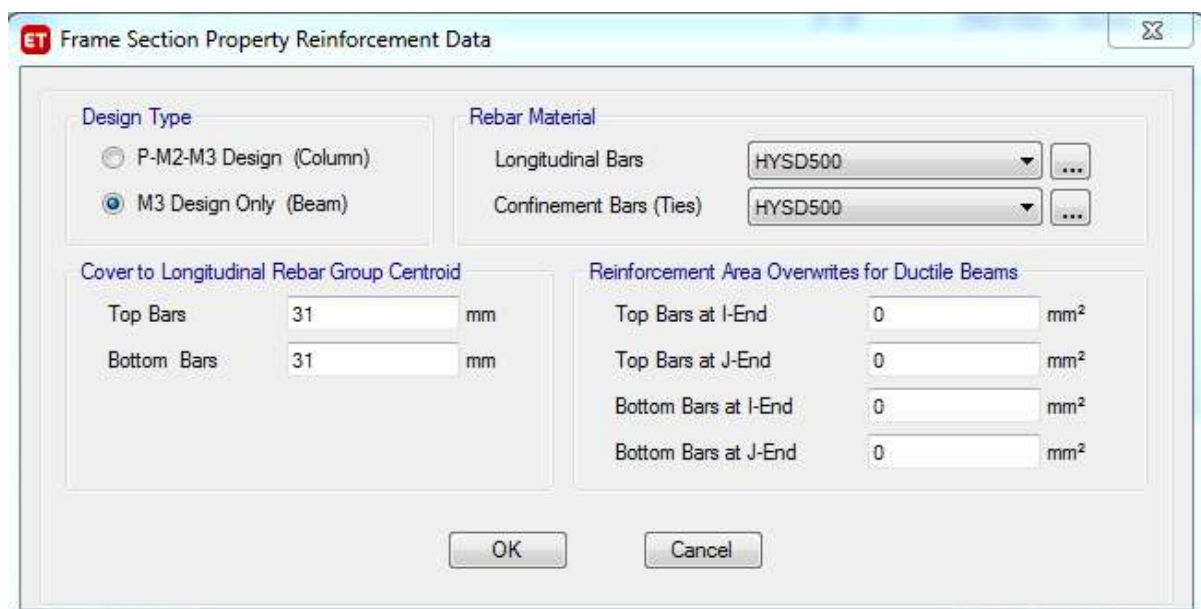
OK Cancel

Show Section Properties...

☐ Include Automatic Rigid Zone Area Over Column

**Fig: Frame Section Property Data form (Beam Definition)**

- Click on **Modify/Show Rebar** to Specify the design type, rebar materials & cover as per the requirements as shown in following figure



**ET Frame Section Property Reinforcement Data**

**Design Type**

☐ P-M2-M3 Design (Column)

☒ M3 Design Only (Beam)

**Rebar Material**

Longitudinal Bars: HYSD500

Confinement Bars (Ties): HYSD500

**Cover to Longitudinal Rebar Group Centroid**

Top Bars: 31 mm

Bottom Bars: 31 mm

**Reinforcement Area Overwrites for Ductile Beams**

Top Bars at I-End: 0 mm<sup>2</sup>

Top Bars at J-End: 0 mm<sup>2</sup>

Bottom Bars at I-End: 0 mm<sup>2</sup>

Bottom Bars at J-End: 0 mm<sup>2</sup>

OK Cancel

**Fig: Frame Section Property Reinforcement Data form (Beam Definition)**

**ET Frame Section Property Data**

**General Data**

Property Name: C230X300

Material: M30

Notional Size Data: Modify/Show Notional Size...

Display Color:   Change...

Notes: Modify/Show Notes...

**Shape**

Section Shape: Concrete Rectangular

**Section Property Source**

Source: User Defined

**Section Dimensions**

Depth: 300 mm

Width: 230 mm

**Property Modifiers**

Modify/Show Modifiers...  
Currently Default

**Reinforcement**

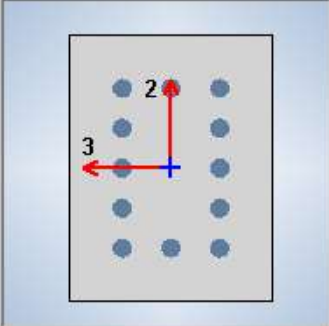
Modify/Show Rebar...

OK

Cancel

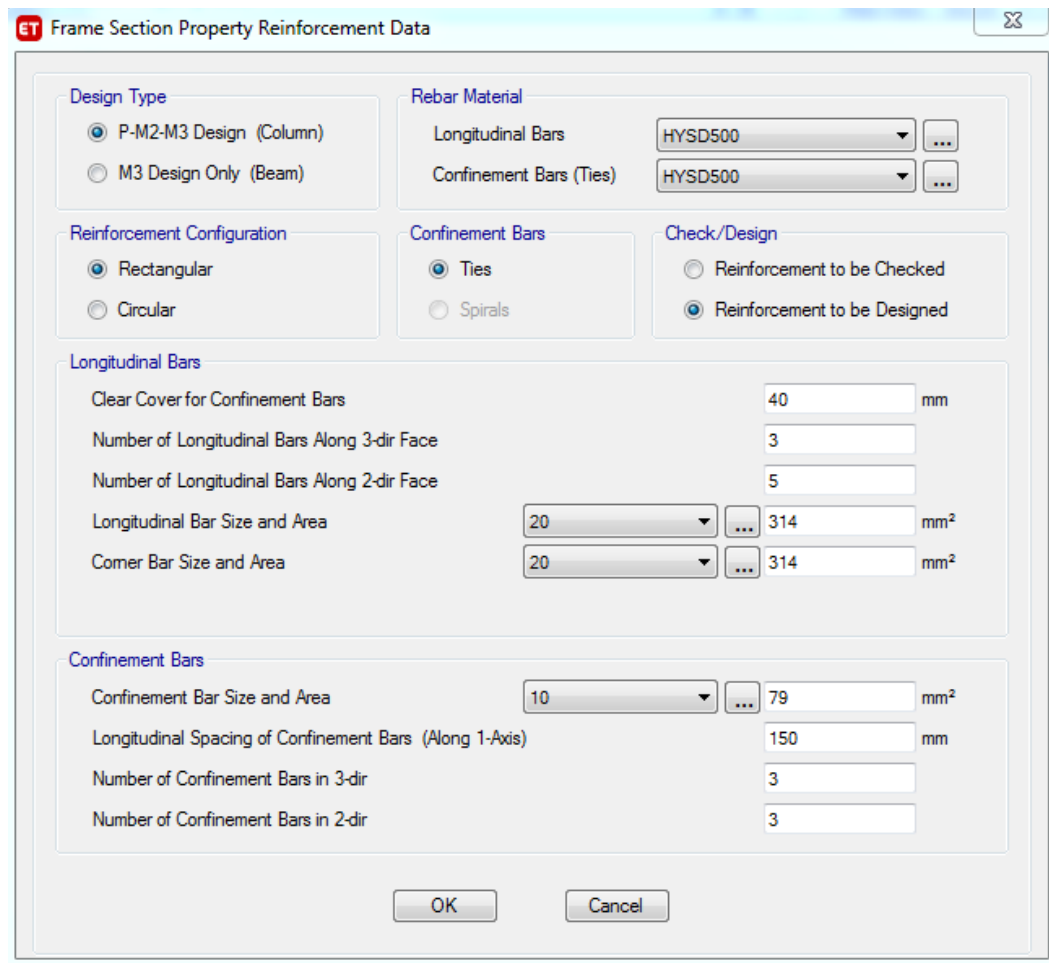
Show Section Properties...

☐ Include Automatic Rigid Zone Area Over Column



**Fig: Frame Section Property Data form (Column Definition)**





**ET Frame Section Property Reinforcement Data**

**Design Type**

- ☒ P-M2-M3 Design (Column)
- ☐ M3 Design Only (Beam)

**Rebar Material**

Longitudinal Bars: HYSD500

Confinement Bars (Ties): HYSD500

**Reinforcement Configuration**

- ☒ Rectangular
- ☐ Circular

**Confinement Bars**

- ☒ Ties
- ☐ Spirals

**Check/Design**

- ☐ Reinforcement to be Checked
- ☒ Reinforcement to be Designed

**Longitudinal Bars**

Clear Cover for Confinement Bars: 40 mm

Number of Longitudinal Bars Along 3-dir Face: 3

Number of Longitudinal Bars Along 2-dir Face: 5

Longitudinal Bar Size and Area: 20 ... 314 mm<sup>2</sup>

Comer Bar Size and Area: 20 ... 314 mm<sup>2</sup>

**Confinement Bars**

Confinement Bar Size and Area: 10 ... 79 mm<sup>2</sup>

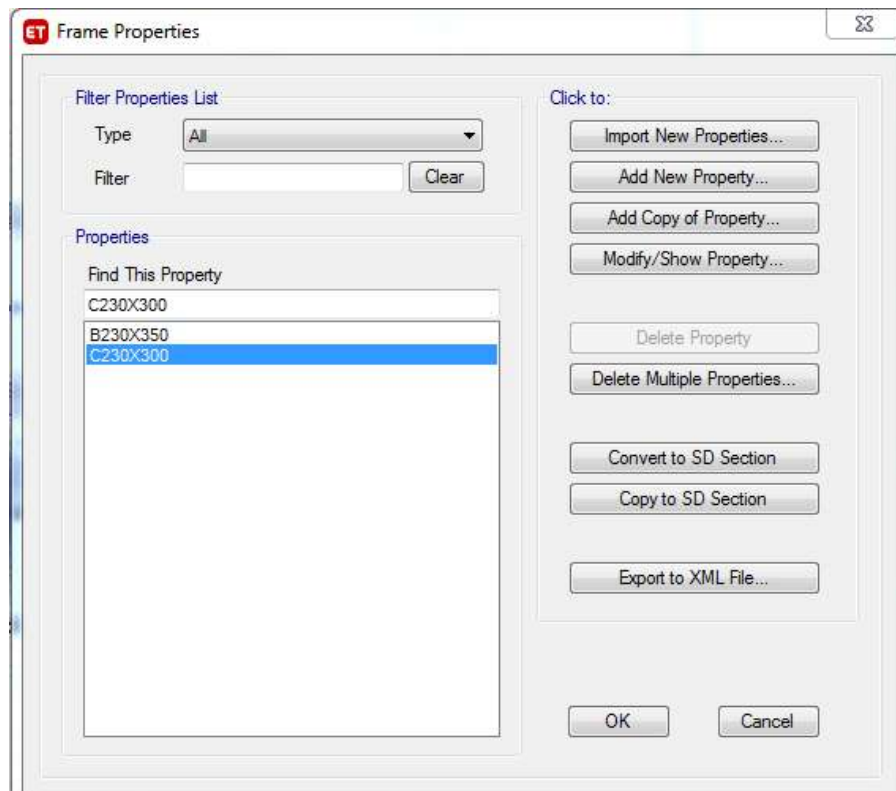
Longitudinal Spacing of Confinement Bars (Along 1-Axis): 150 mm

Number of Confinement Bars in 3-dir: 3

Number of Confinement Bars in 2-dir: 3

OK Cancel

**Fig: Frame Section Property Reinforcement Data form (Column Definition)**



**ET Frame Properties**

**Filter Properties List**

Type: All

Filter: Clear

**Properties**

Find This Property

- C230X300
- B230X350
- C230X300**

**Click to:**

- Import New Properties...
- Add New Property...
- Add Copy of Property...
- Modify/Show Property...
- Delete Property
- Delete Multiple Properties...
- Convert to SD Section
- Copy to SD Section
- Export to XML File...

OK Cancel

**Fig: Frame Properties form**

3. Click the **Define menu > Section Properties > Slab Sections** command to access the **Slab Properties** form. Click on **Add New Property** button to add Slab of 150mm thickness as shown below

**Fig: Slab Property Data form**

**Fig: Slab Properties form**

## Level- 4: Assigning Structural elements & Supports

### 1. Assigning Structural elements :

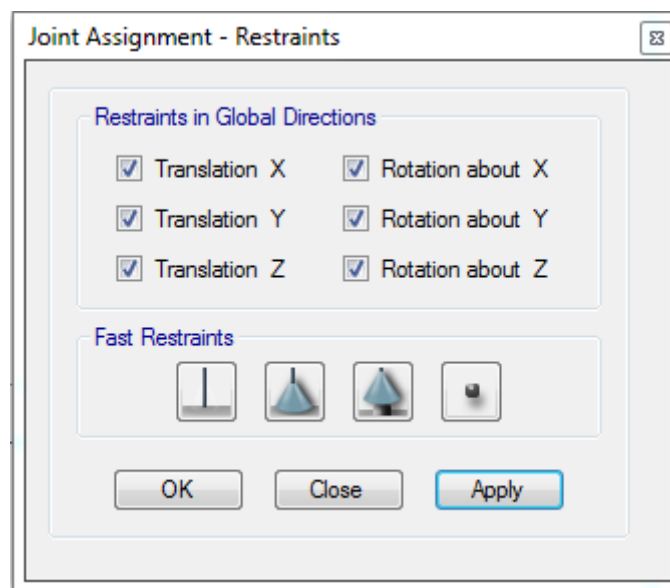
This step involves assigning defined structural elements of **Level:3** to the Empty grid generated in **level-1** by using various draw tools.

- Keep the **Story Settings** as **All Stories** and Assign the beams & columns to grid by using either **Draw Beam/Column/brace (Plan,Elev,3D)** or **Quick Draw beam Beams/columns(Plan,Elev,3D)** tool from **Draw** menu.
- Similarly assign the slabs in all stories by using **Quick Draw Floor/Wall** option

### 2. Assigning Supports:

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.

Ex: Generally for a building at bottom fixed supports has to be assigned to indicate footings.



**Fig: Joint Assignment- Restraints form**

## Level-5: Load Assignments

This step involves defining and assigning various loads on to the structure like dead load, live load and lateral loads. Along with loads Load combinations are also defined by using the **Define** menu.

### •Wall load calculations

Unit weight of brick = 19 kN/m<sup>3</sup>

#### Main wall load

Thickness of wall = 230 mm

$$\begin{aligned} \text{DL} &= \text{unit weight of brick} \times \text{thickness of wall} \times (\text{floor height} - \text{beam depth}) \\ &= 19 \times 0.23 \times (3 - 0.35) \\ &= 11.58 \text{ kN/m} \end{aligned}$$

## Partition wall load

$$\begin{aligned} \text{Thickness of wall} &= 115 \text{ mm} \\ &= 19 \times 0.115 \times (3 - 0.35) \\ &= 5.79 \text{ kN/m} \end{aligned}$$

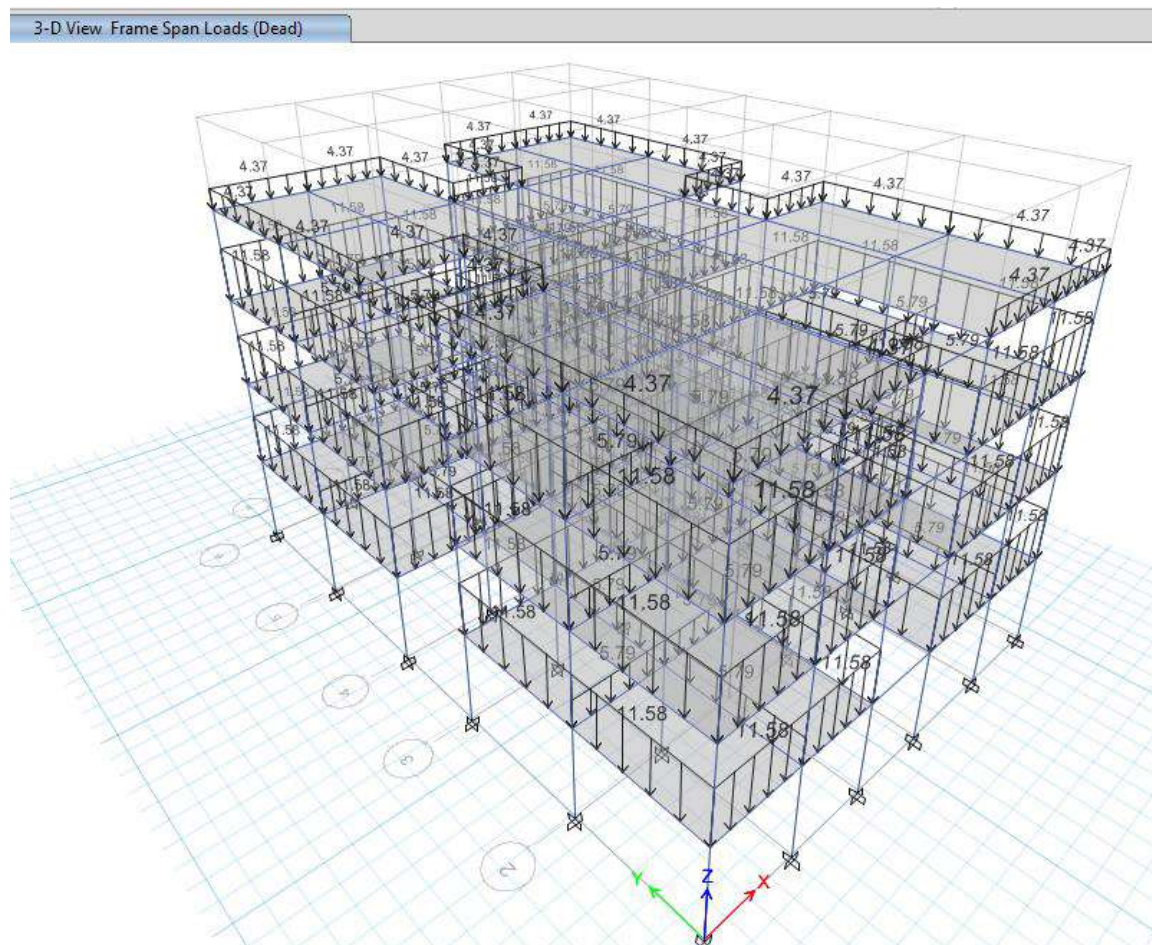
## ●Slab load calculations

Floor finish (Dead Load) = 1.5kN/m<sup>2</sup>(as per IS 875)

Live load can be considered as per IS 875 part 2

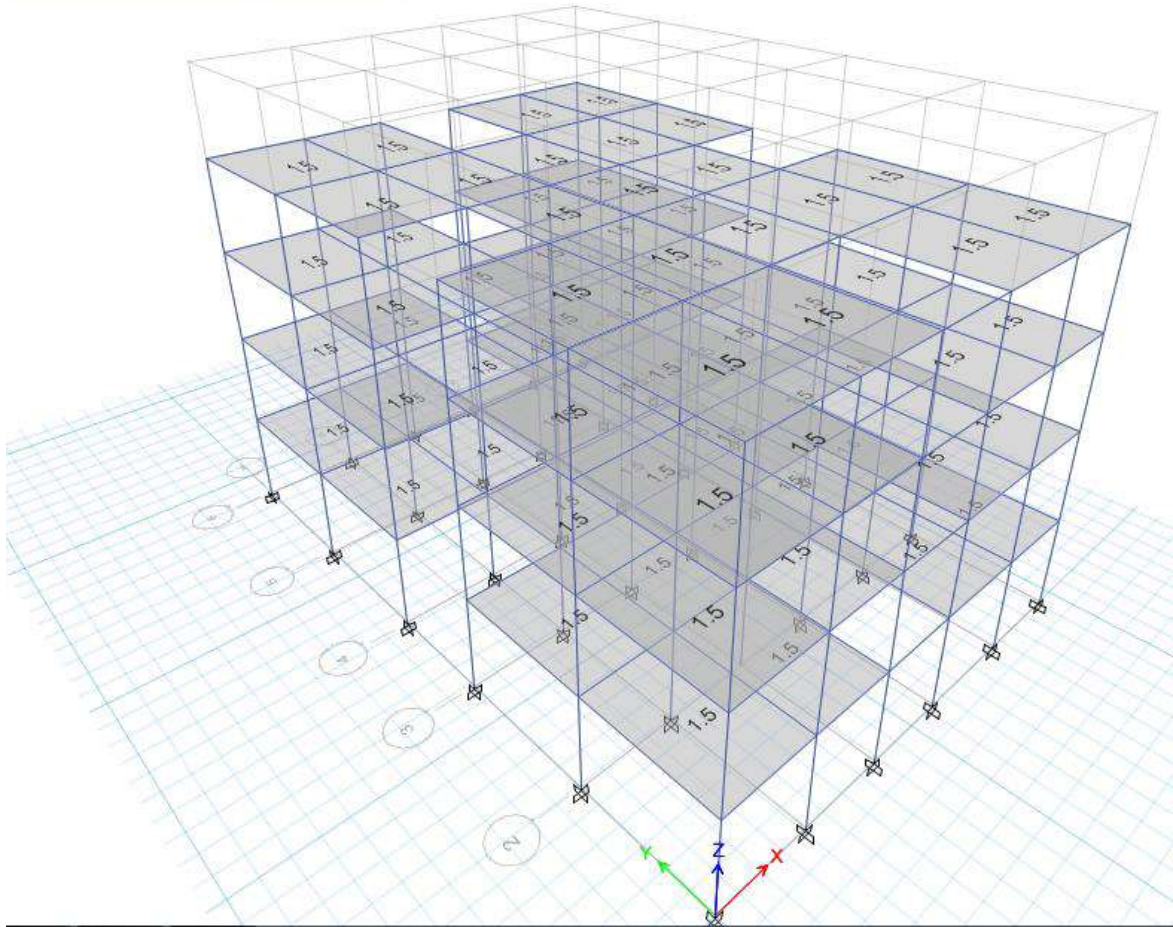
For Residential buildings refer page no. 7

1. Assign the main wall load & partition wall load on the respective beams as per the plan by using **Frame Loads**
2. Assign the Slab loads(Floor finish load & live load) using **Shell Loads**
3. Modify the loads on terrace story.

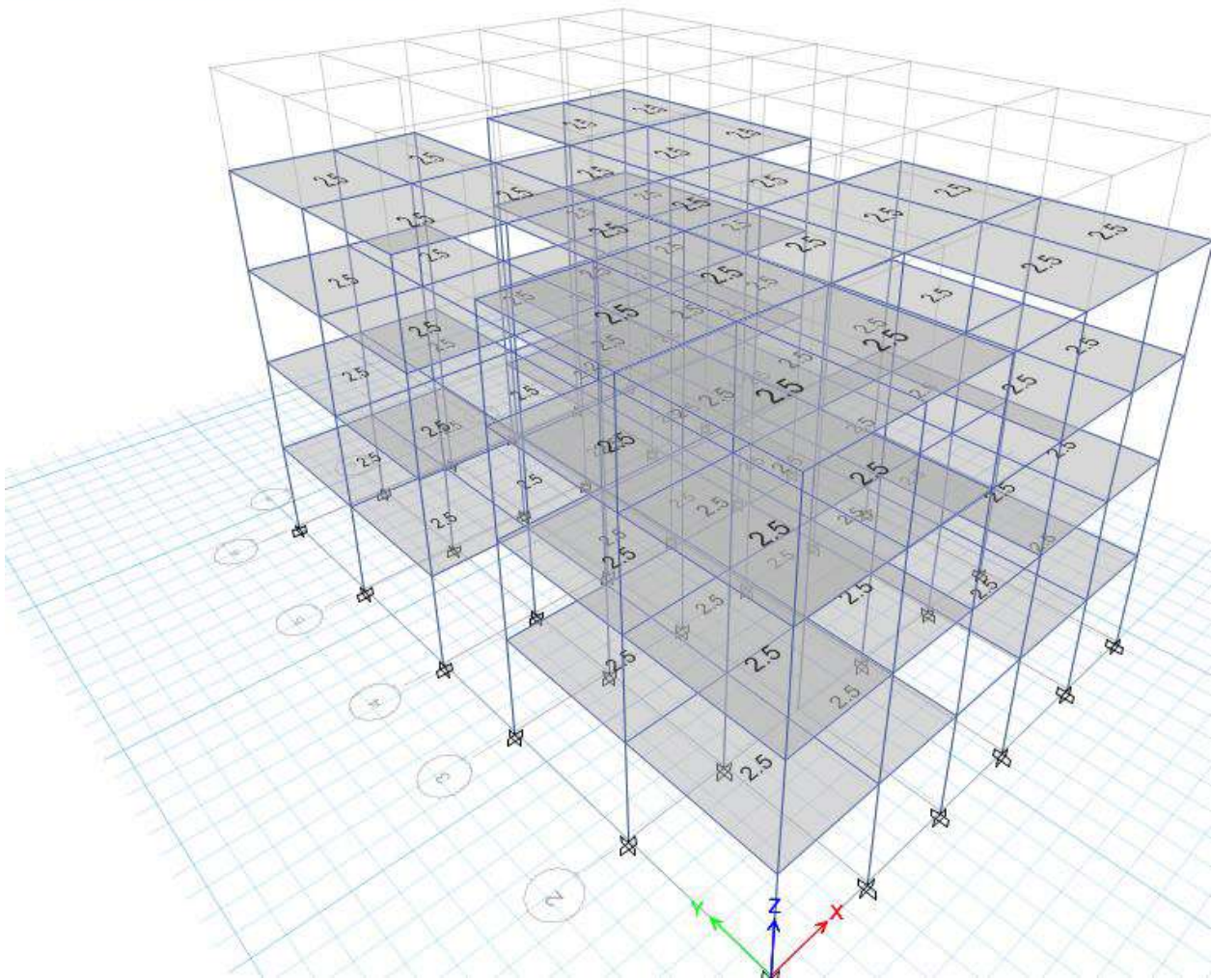


**Fig: Showing wall load assignment on beams**





**Fig: Showing Floor finish load assignment on slabs**



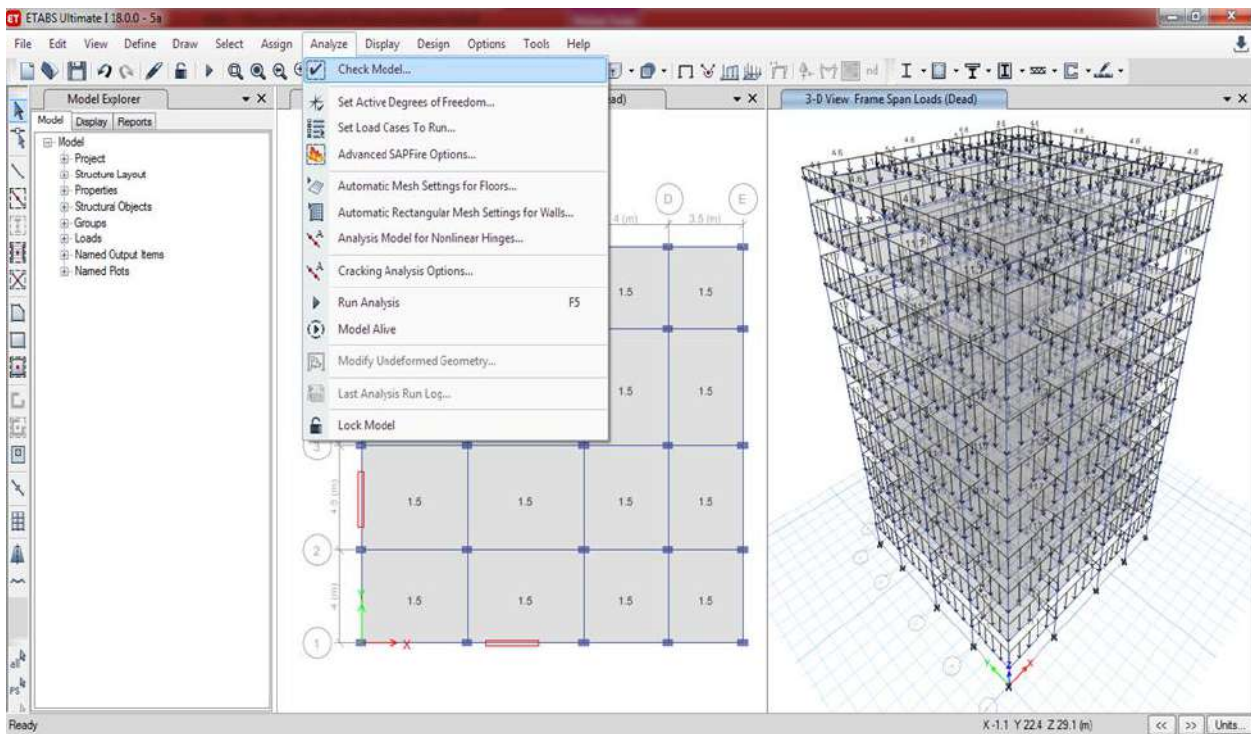
**Fig: Showing Live load assignment on Slabs**

4. To define the load combinations go to **Define Menu> Load Combinations> Add Default Design Combos**, select **Concrete Frame Design** and click **OK**

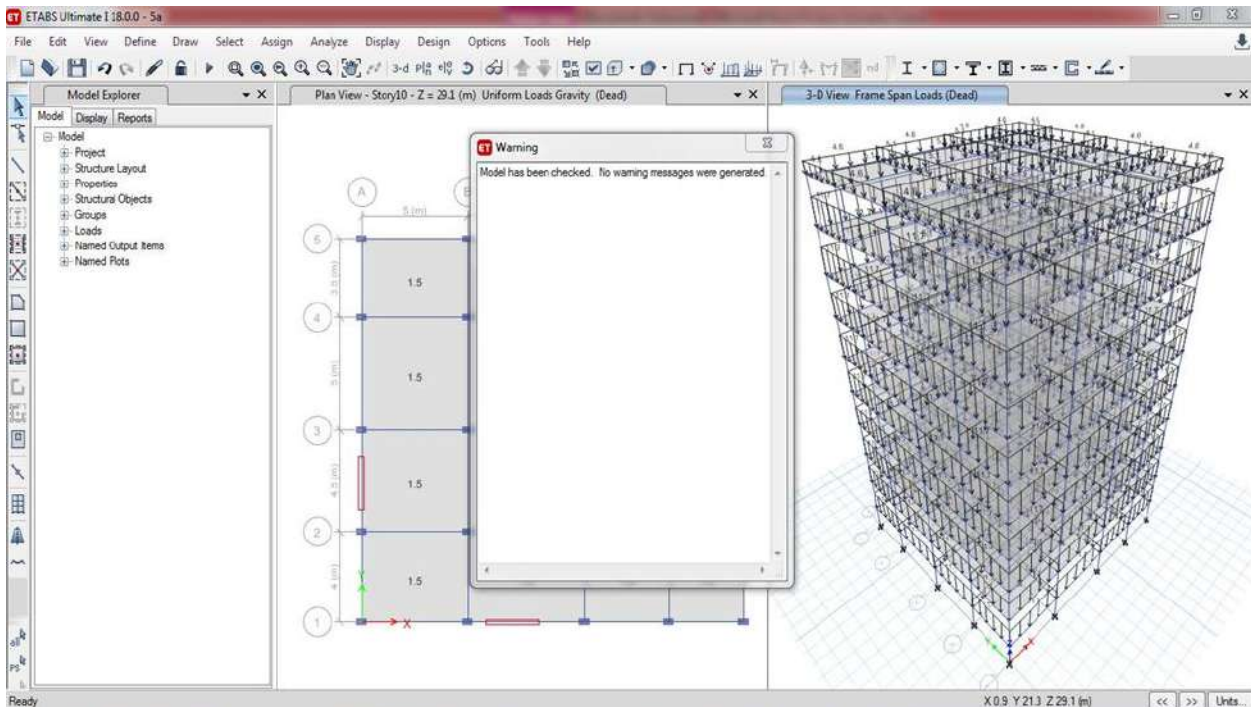
### **Level-6: Model Check**

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



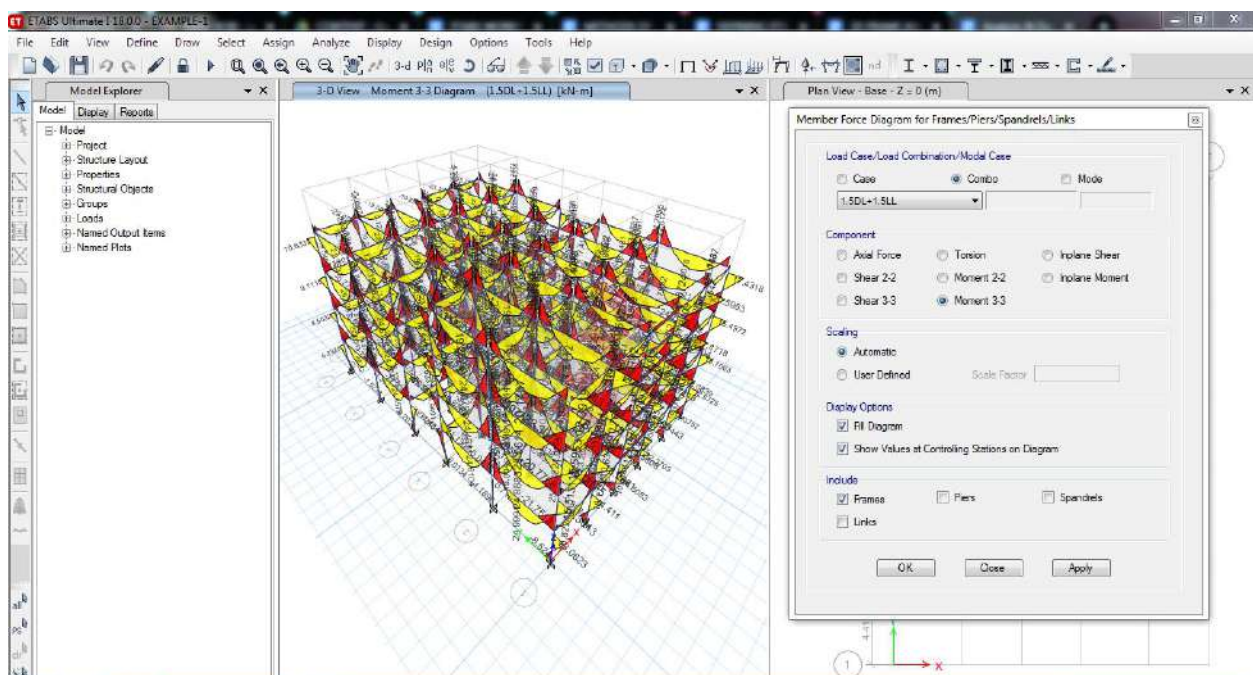
## Level-7: Analysis

In this step you will run the analysis for the generated structure. To run the analysis click the F5 function key or choose **Run Analysis** from **Analyze** menu. As the analysis completes it shows the deflection diagram initially.

To see the animation of deflection click on **Start Animation** button in the Status bar.

## Level-8: Results Interpretation

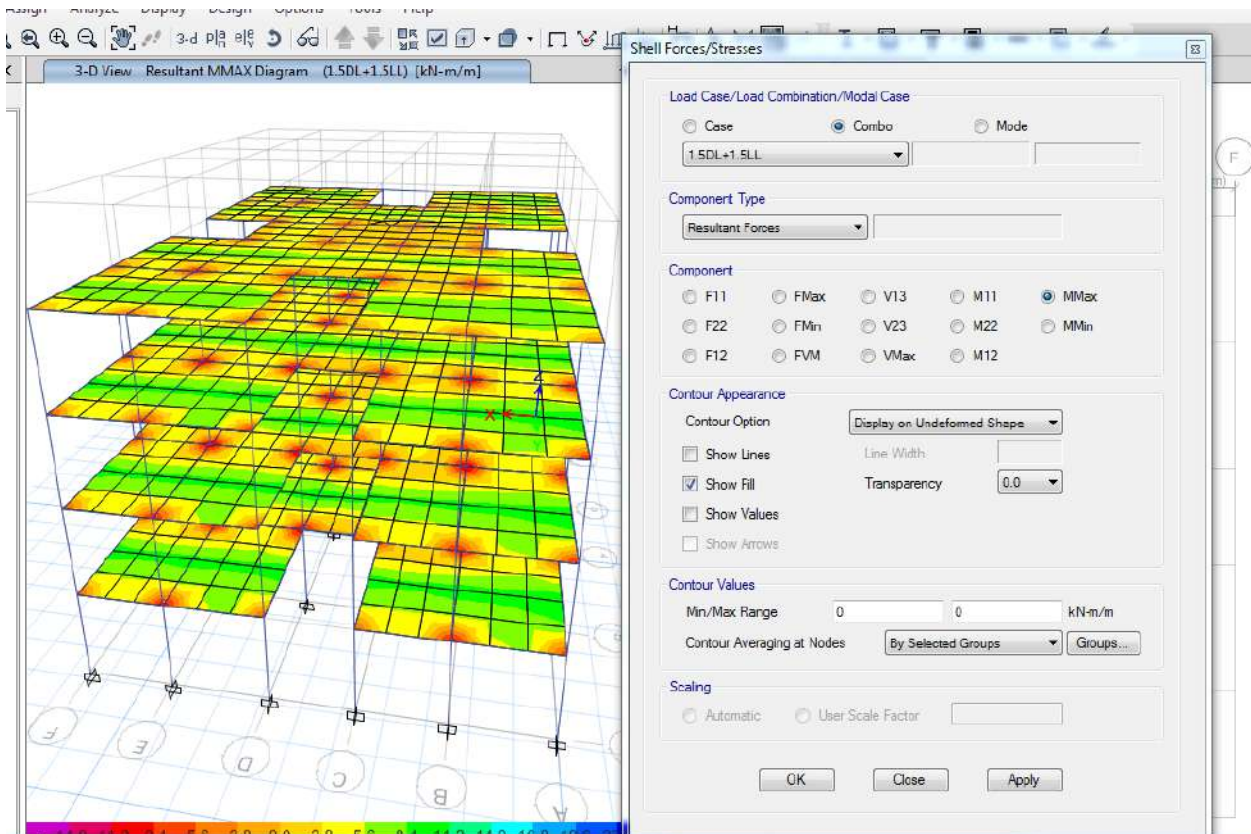
1. After analysis you are accessed to see results such as deflection, bending moment, shear forces and reactions.
2. The results can be graphically represented as shown in fig for each and every structural element individually.
3. To check the results like BMD or SFD click on **Display Frames/Piers/Spandrels/Links** or **F8**, select Load case > select Moment 3-3 or Shear 2-2 respectively and click on **OK**
4. It displays results for all the assigned loads and load combinations individually.



**Fig: BMD**



## 5. To check the results of slabs **Display Shell Stresses/Forces** or **F9**



**Fig: Max moments in Slab**

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

The following fig shows the design results of a structure

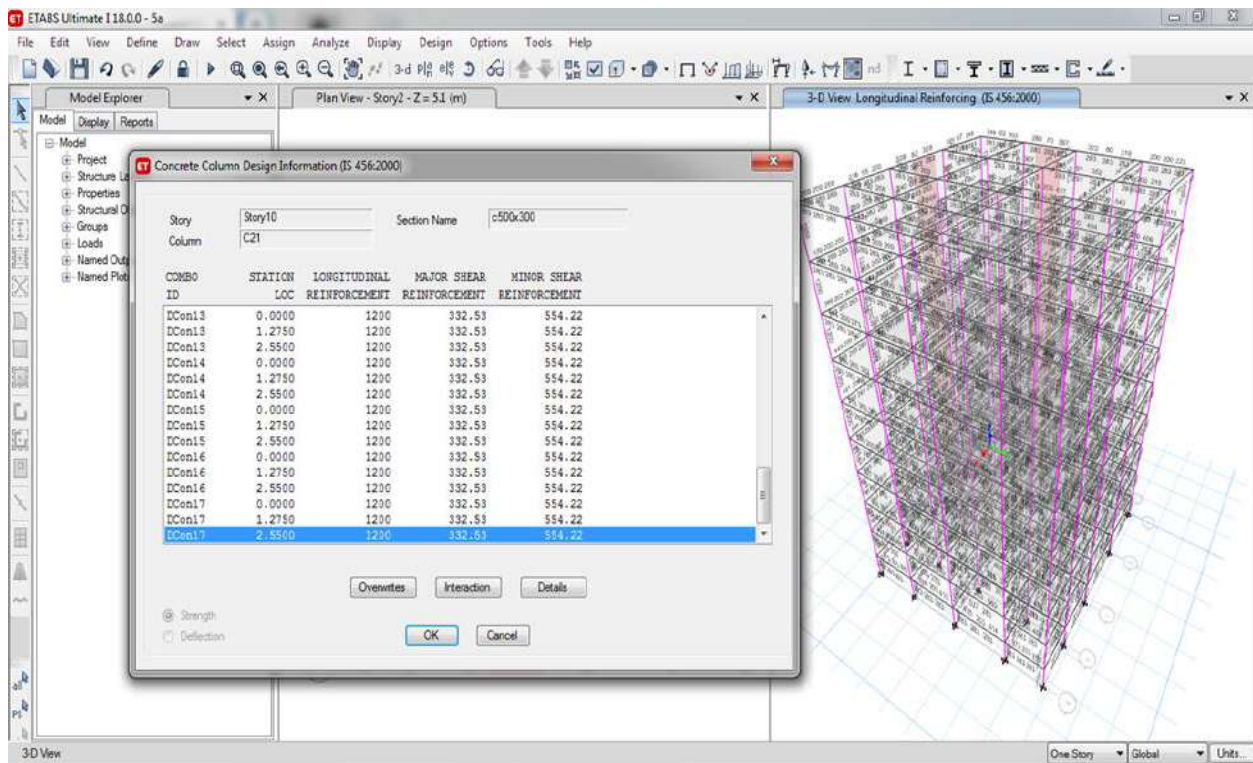


Fig: Design Details

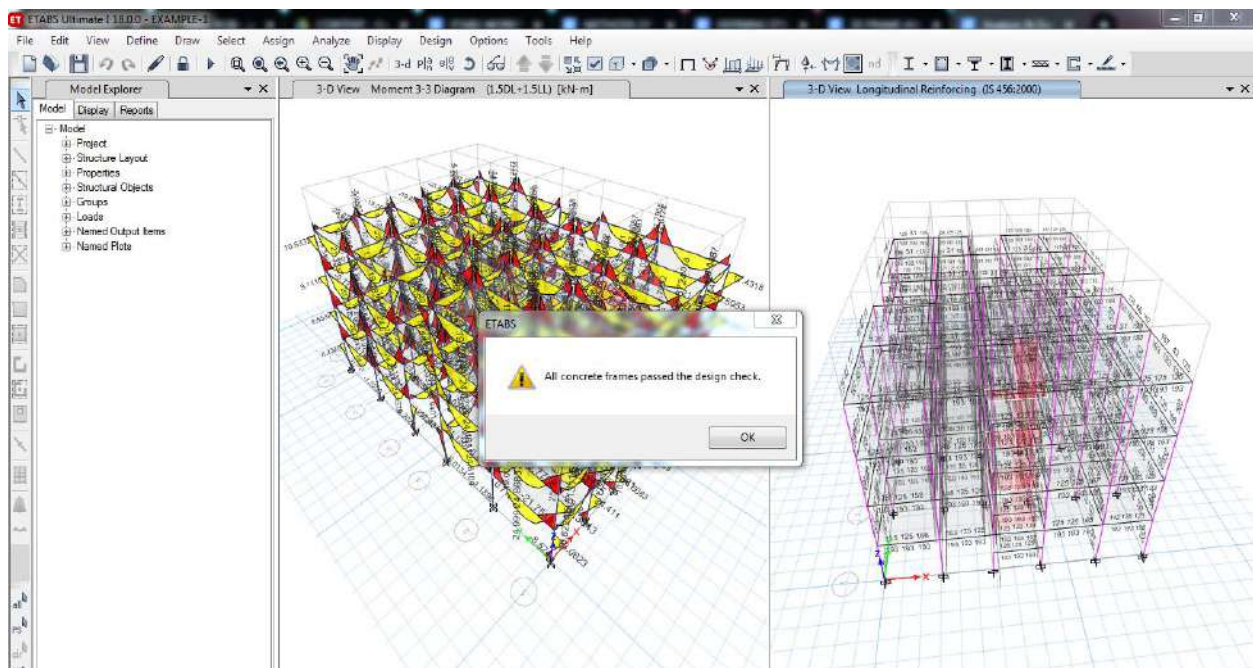


Fig: Design check