Andhra Pradesh State Skill Development Corporation

Skill AP
APSSDC

eSkill AP
Learn Anytime Anywhere
Andhra Pradesh State Skill Development Corporation

Extended Three-Dimensional Analysis of Building System

ETABS

Analysis & Design of Multistoried Building

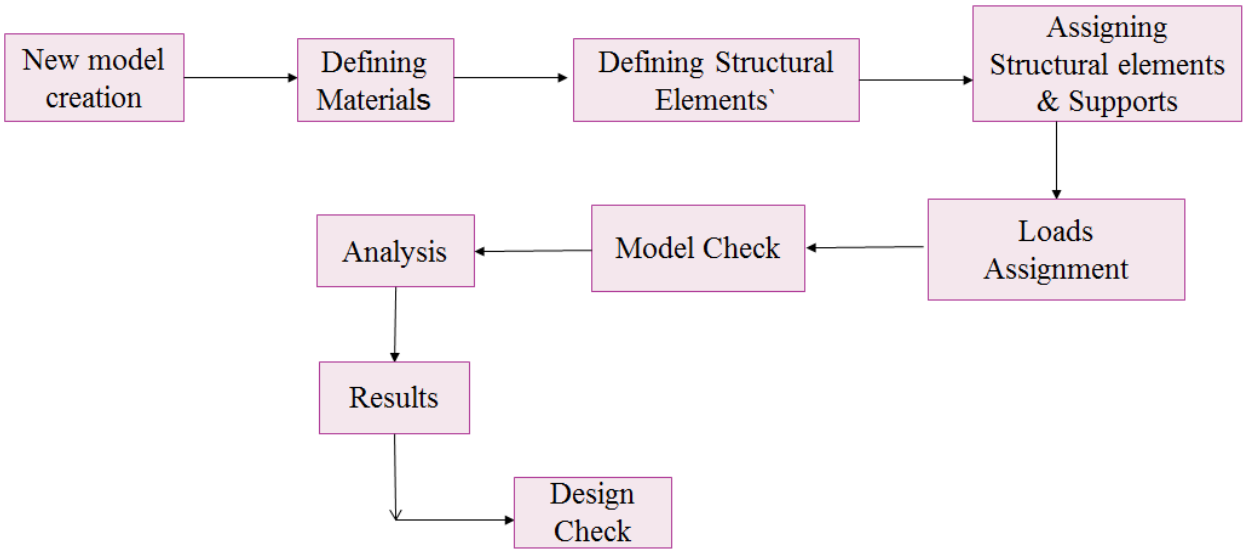
Analysis & Design of Multistoried Building

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

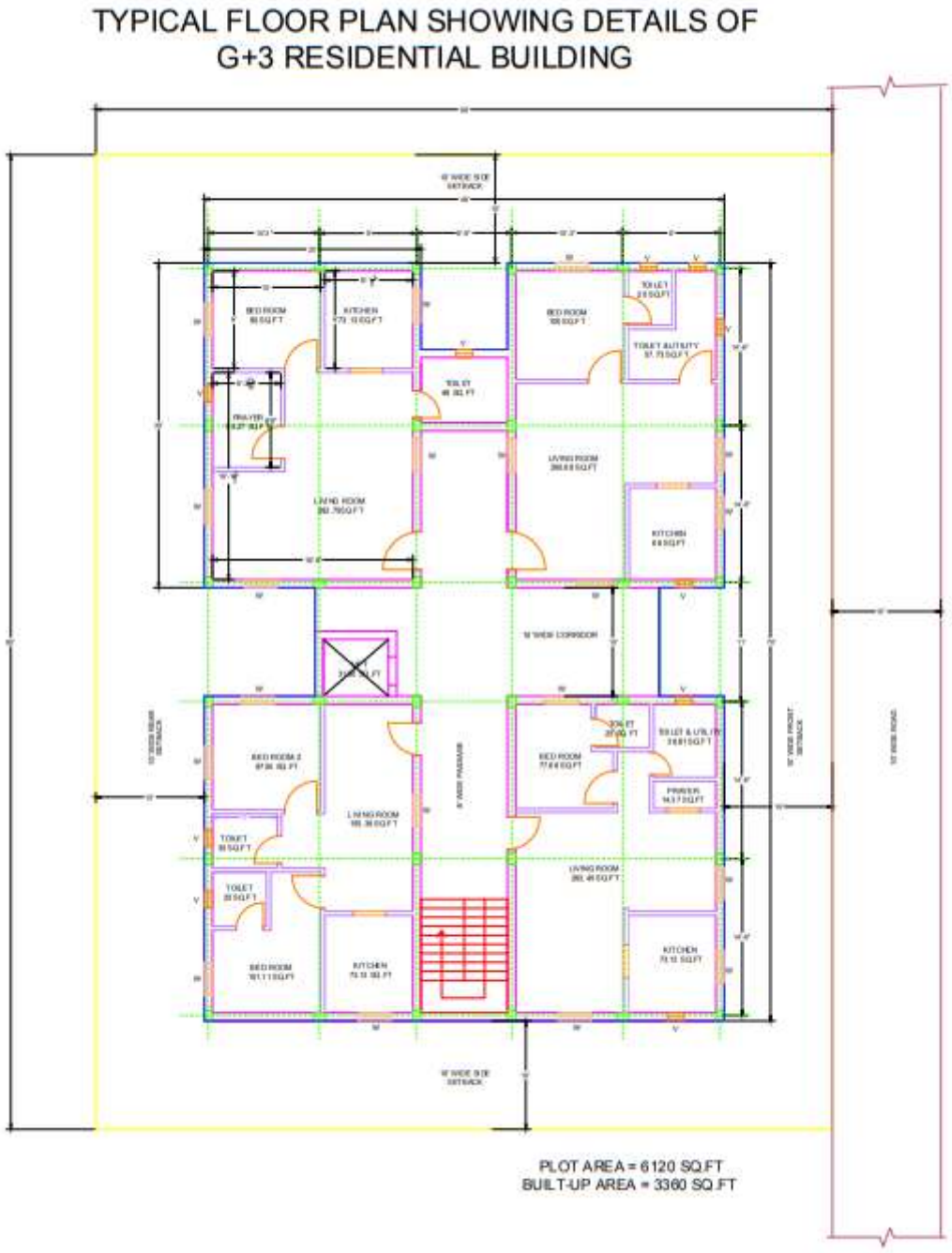
Steel: HYSD500

Section Property

Beam Section: 230mm X 350mm

Column Sections: 230mm X 300mm

Slab: 150mm



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**

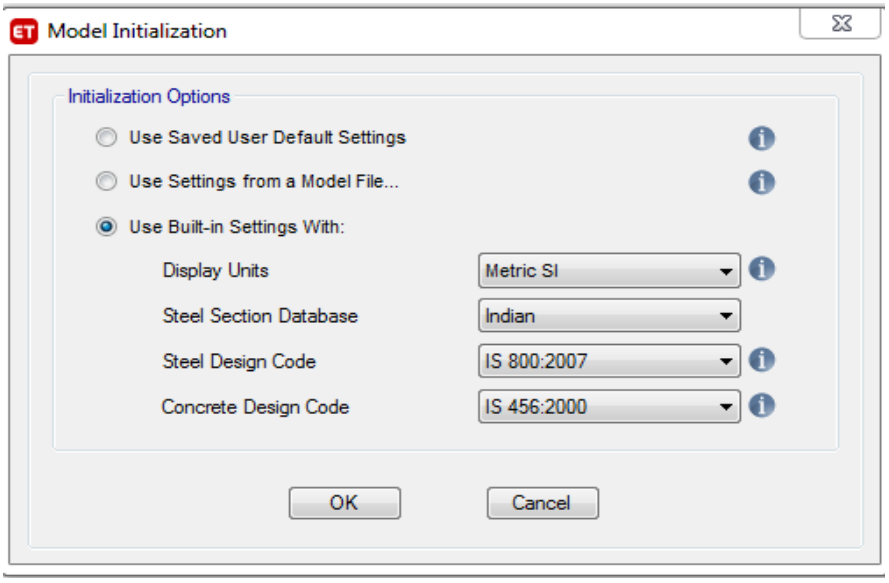


Fig: Model Initialization Form

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.

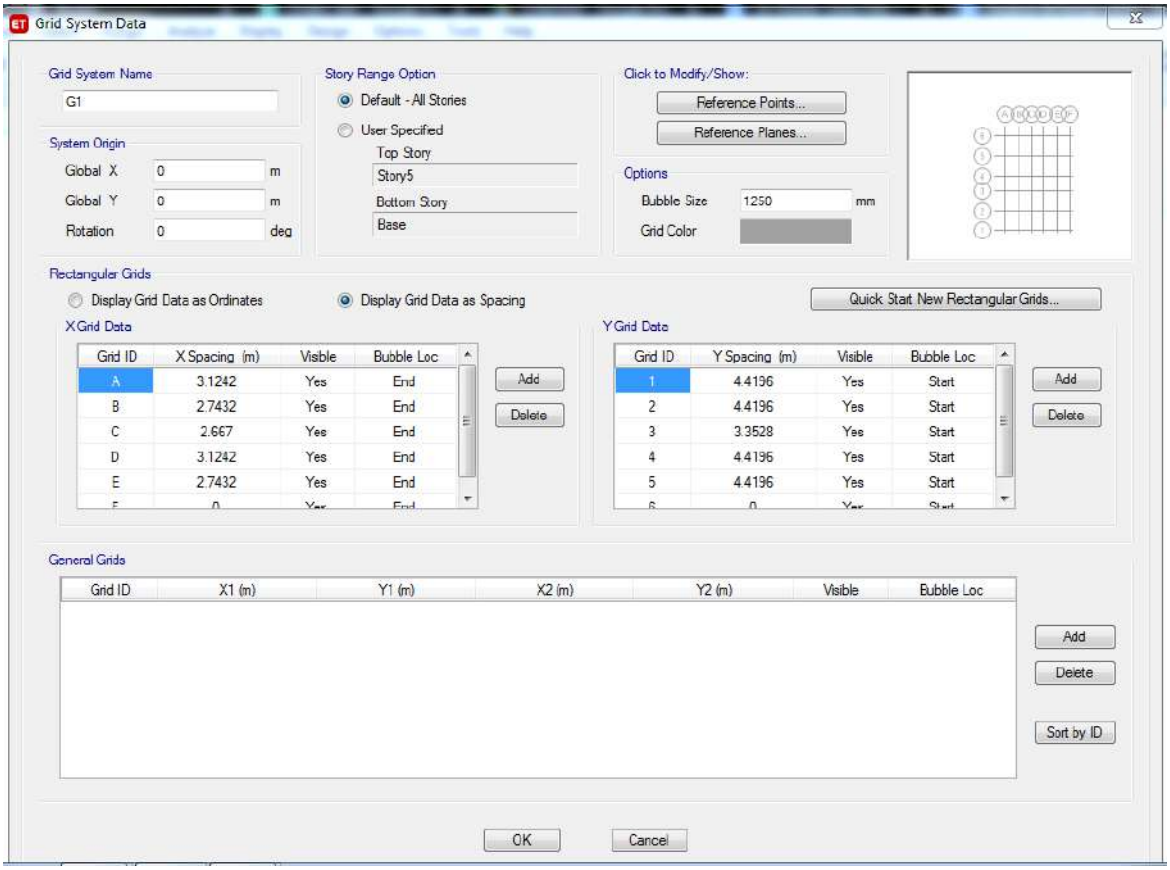


Fig: Grid System Data form

5. 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.
6. Select **Grid only** template from **Add Structural Template.** By keeping remaining settings as default click **Ok.**

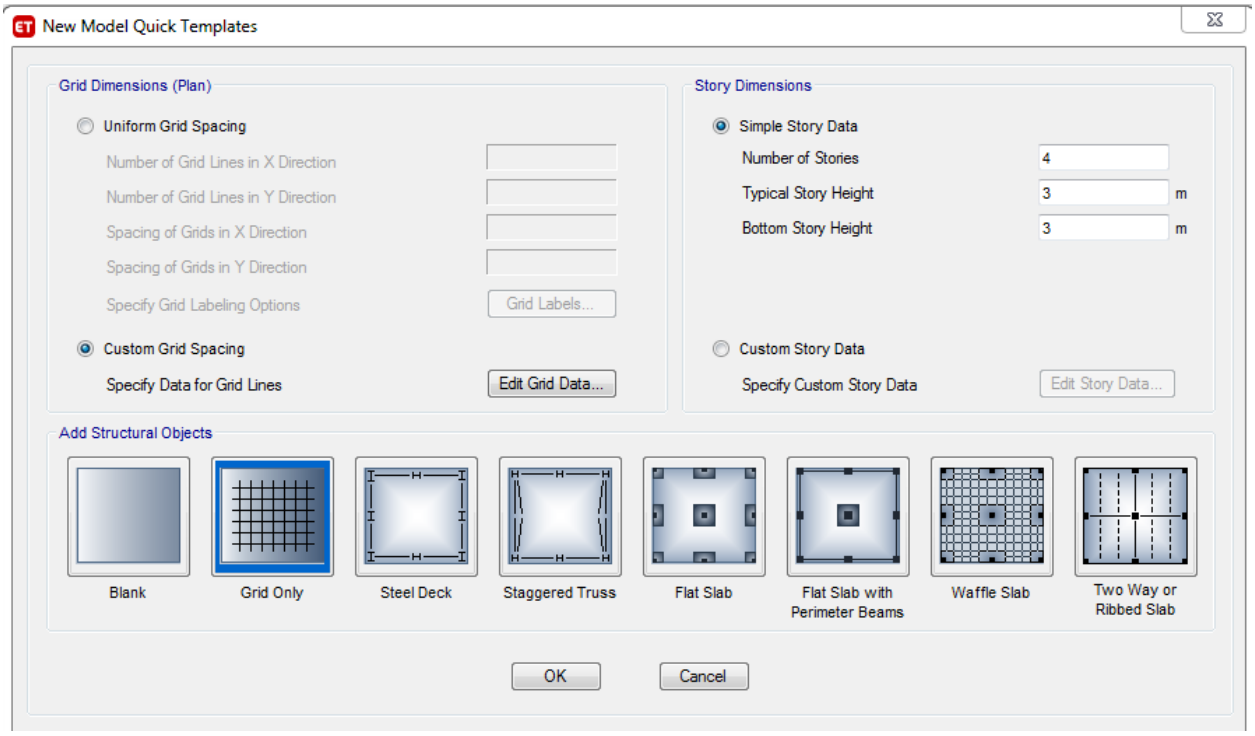
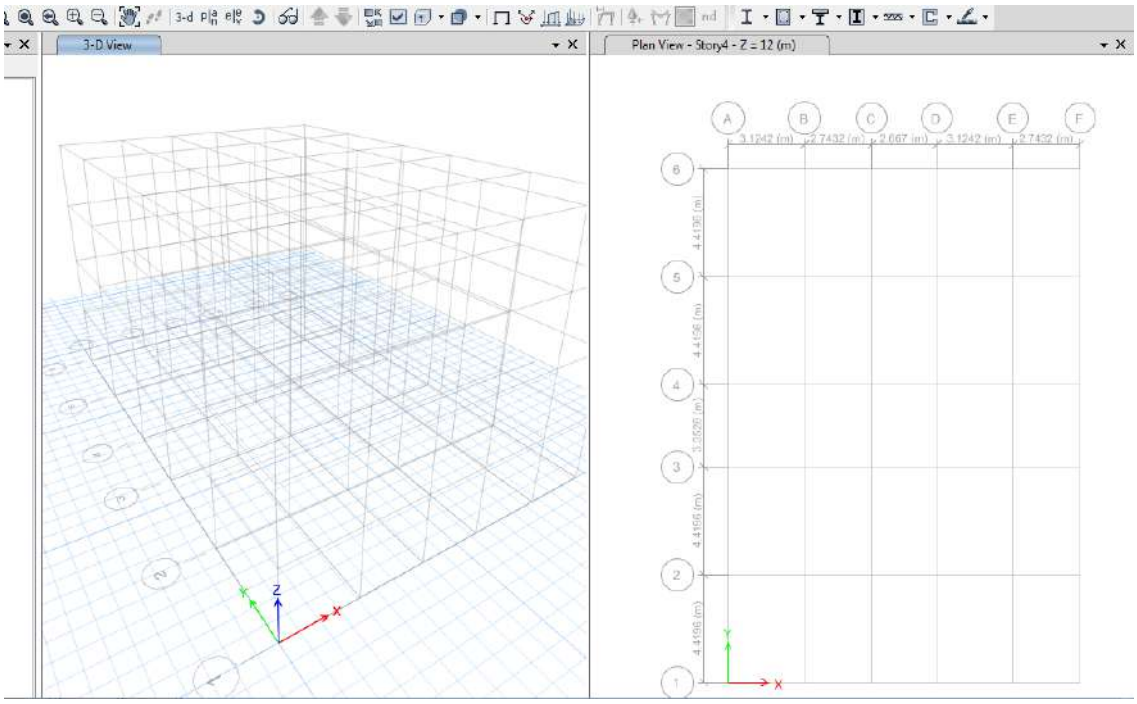


Fig: New Model Quick Template Form

7. 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.

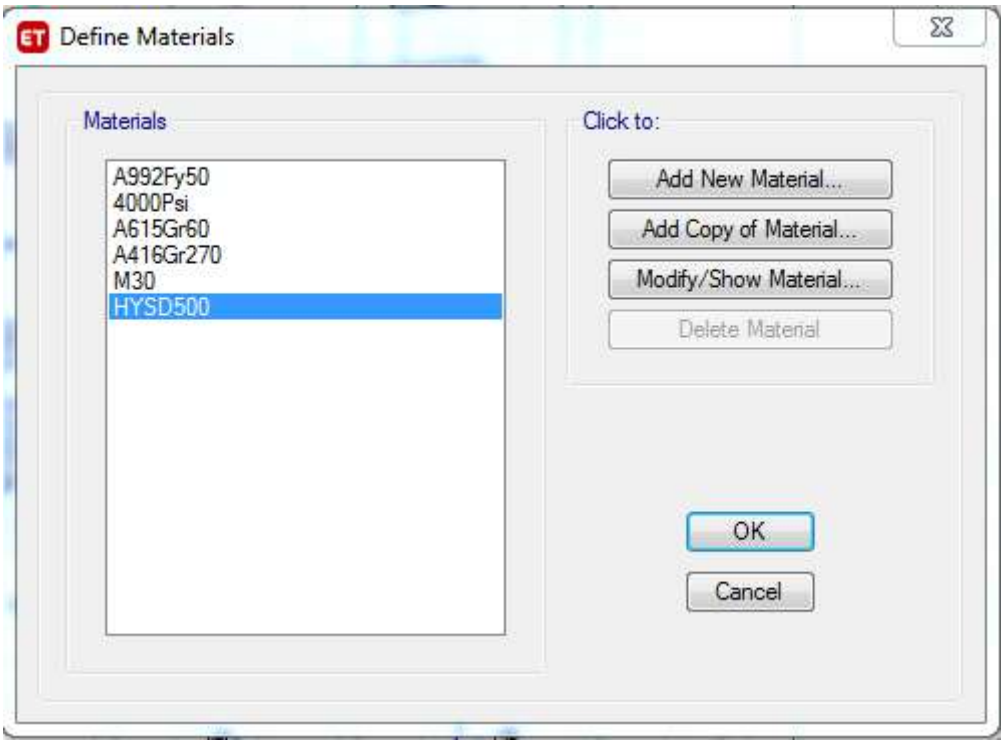


Fig: Define Material Form

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

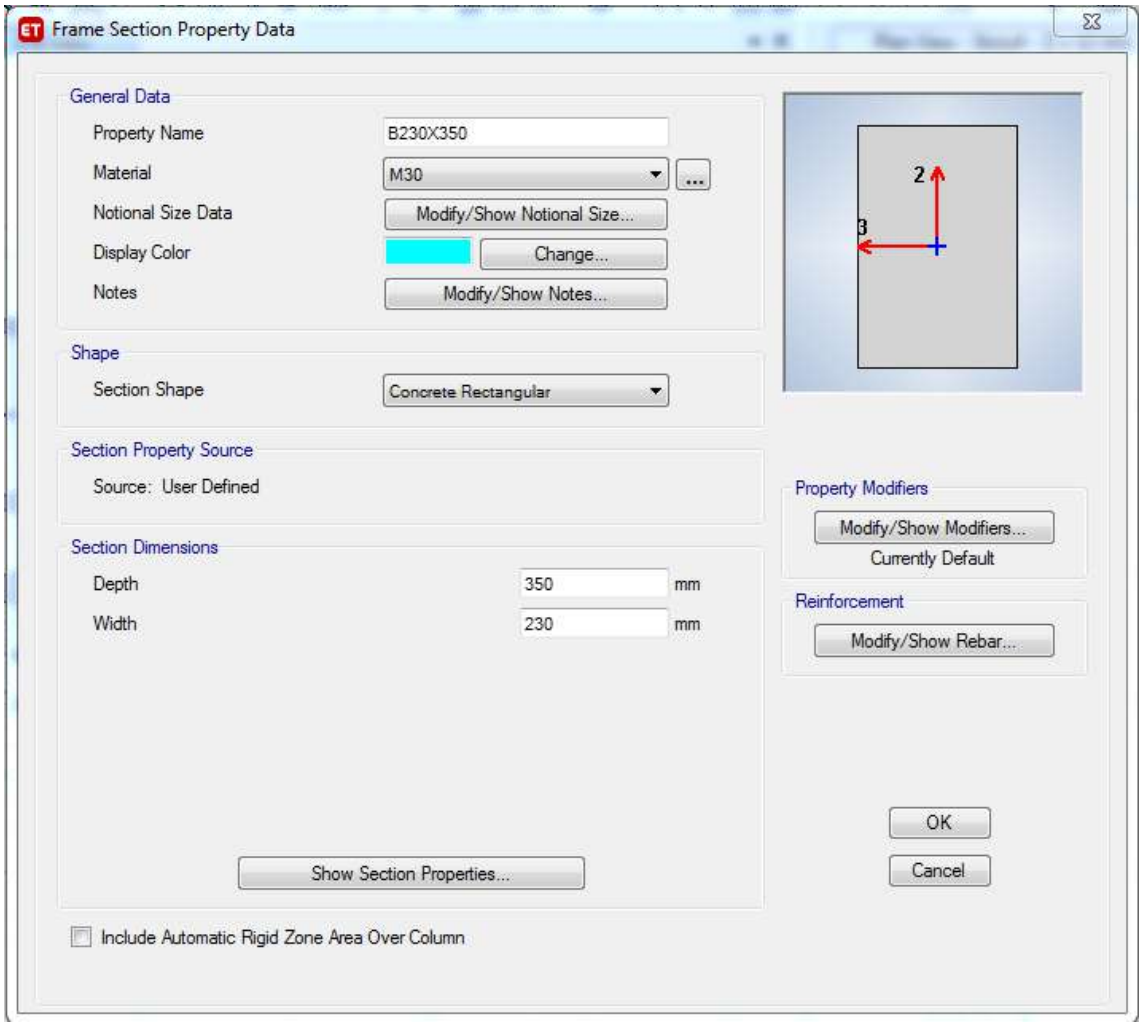


Fig: Frame Section Property Data form (Beam Definition)

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

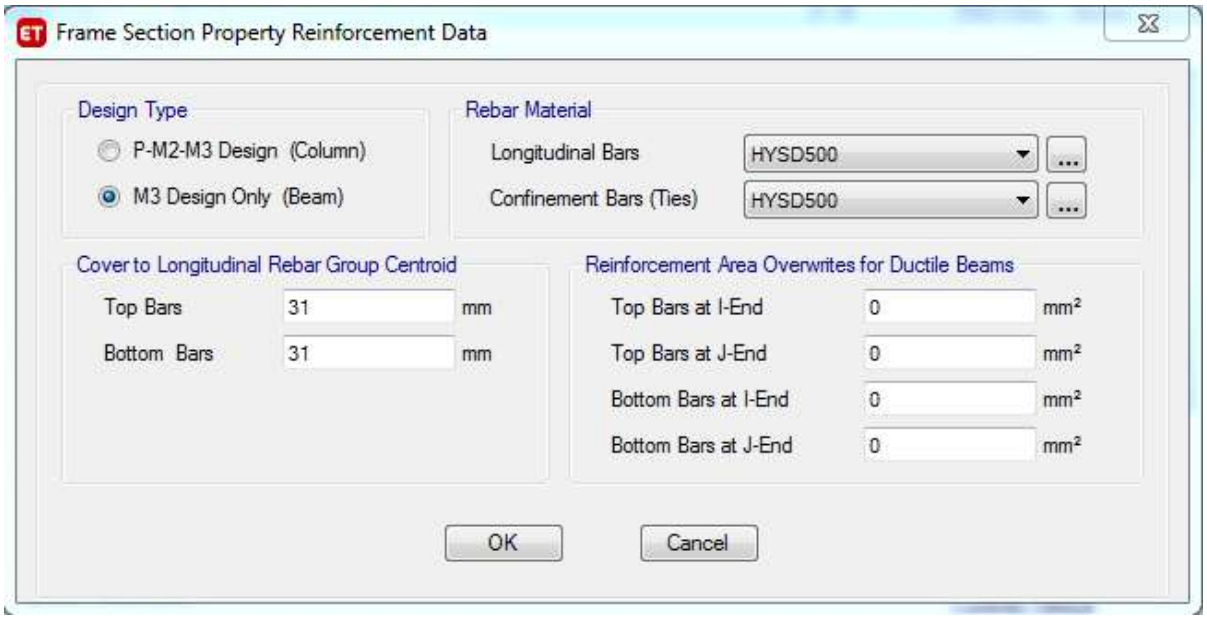


Fig: Frame Section Property Reinforcement Data form (Beam Definition)

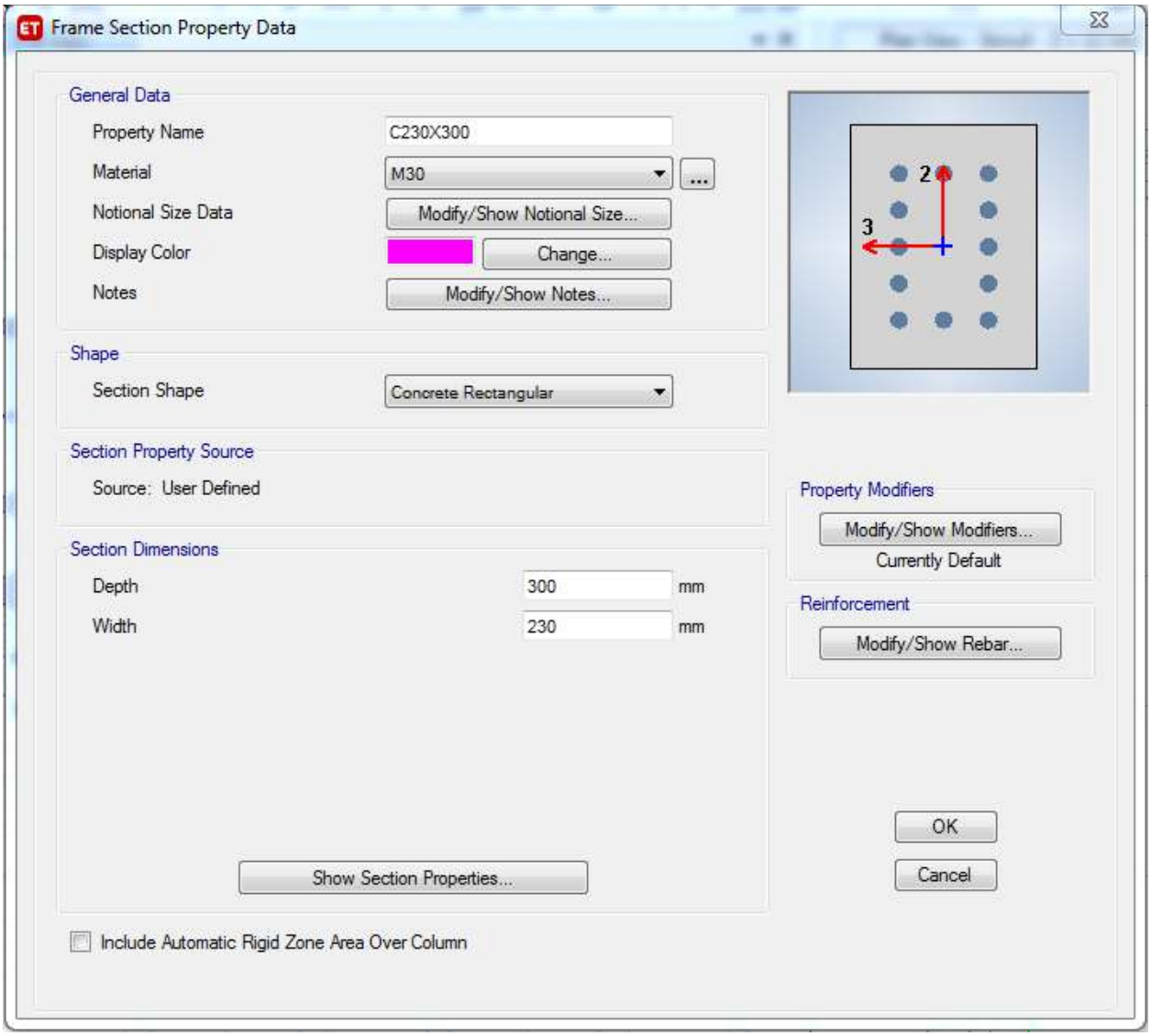


Fig: Frame Section Property Data form (Column Definition)

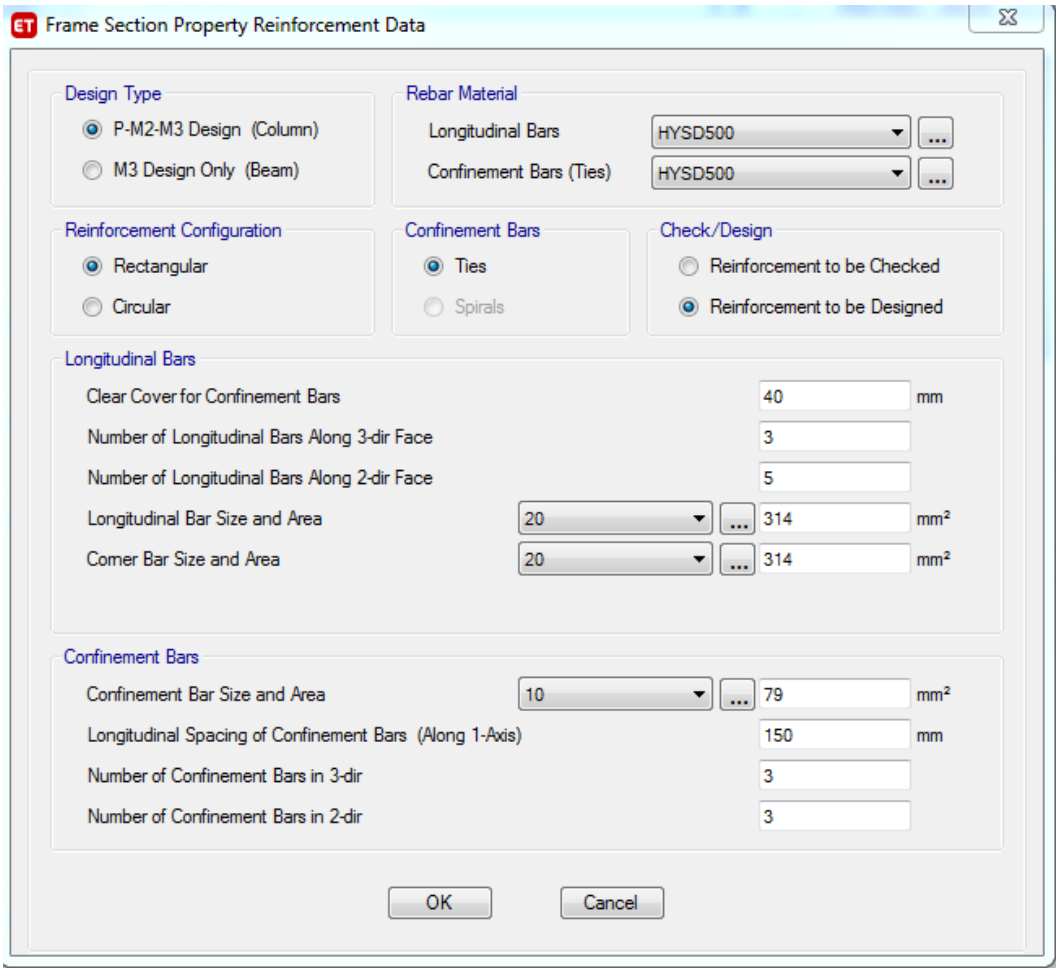


Fig: Frame Section Property Reinforcement Data form (Column Definition)

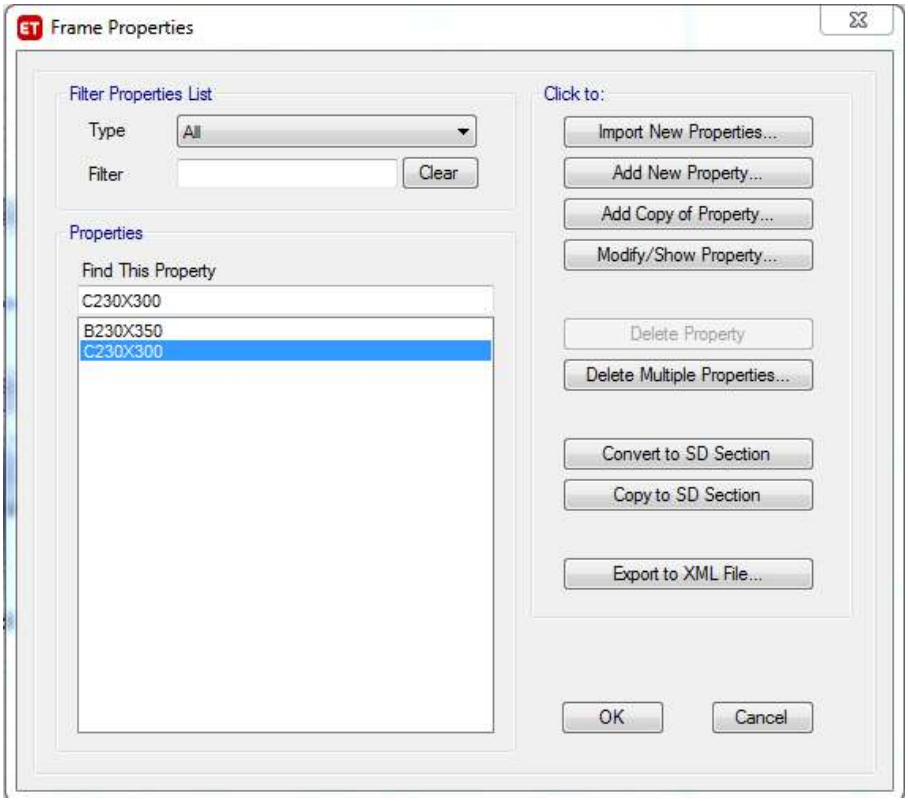


Fig: Frame Properties form

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

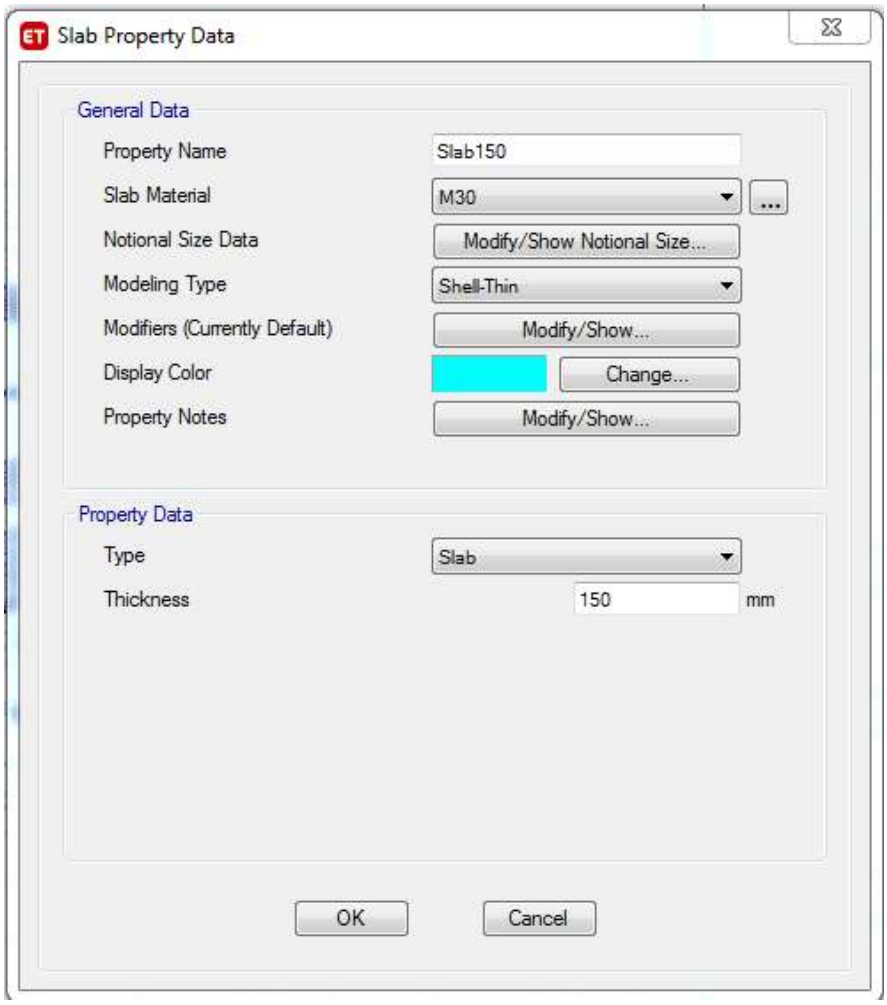


Fig: Slab Property Data form

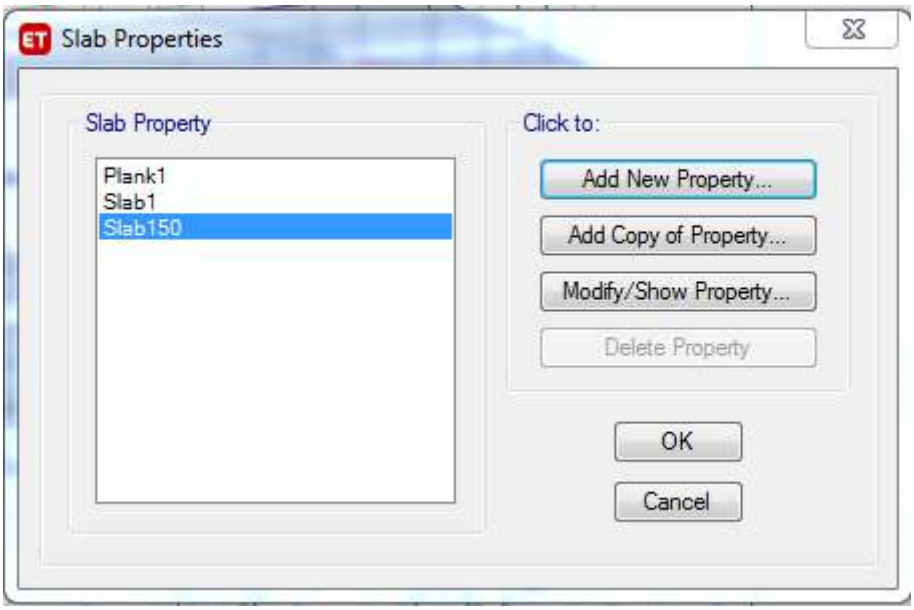


Fig: Slab Properties form

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.

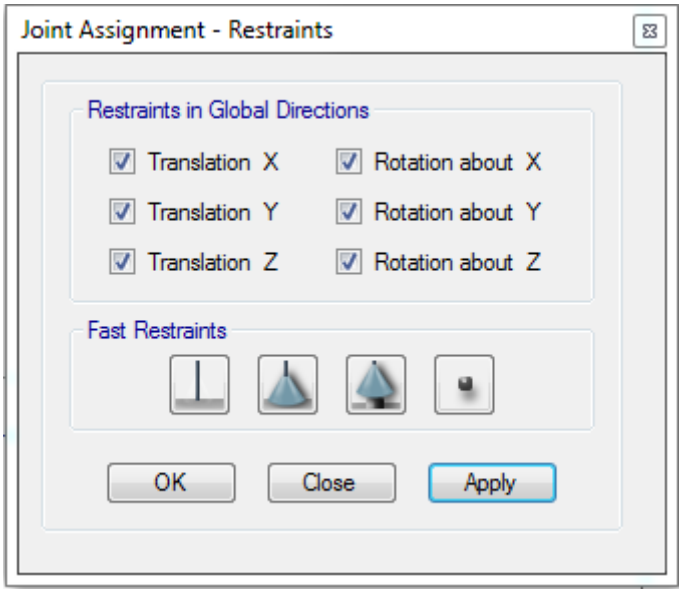


Fig: Joint Assignment- Restraints form

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^3

Main wall load

Thickness of wall = 230 mm

DL = unit weight of brick x thickness of wall x(floor height –beam depth)

=19 x 0.23 x (3 -0.35)

= 11.58 kN/m

Partition wall load

Thickness of wall = 115 mm

= 19 x 0.115 x (3 -0.35)

= 5.79 kN/m

- **Slab load calculations**

Floor finish (Dead Load) = 1.5kN/m2(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.

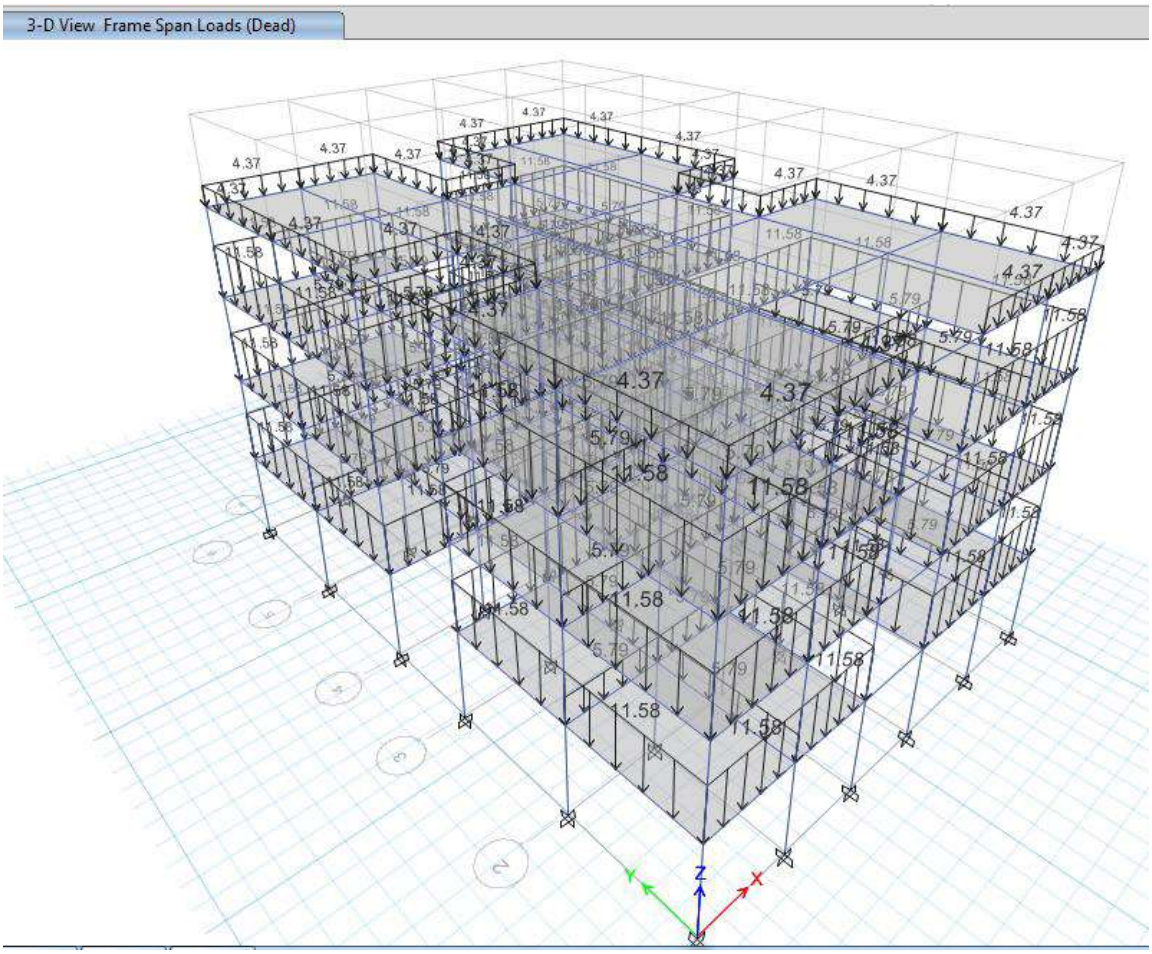


Fig: Showing wall load assignment on beams

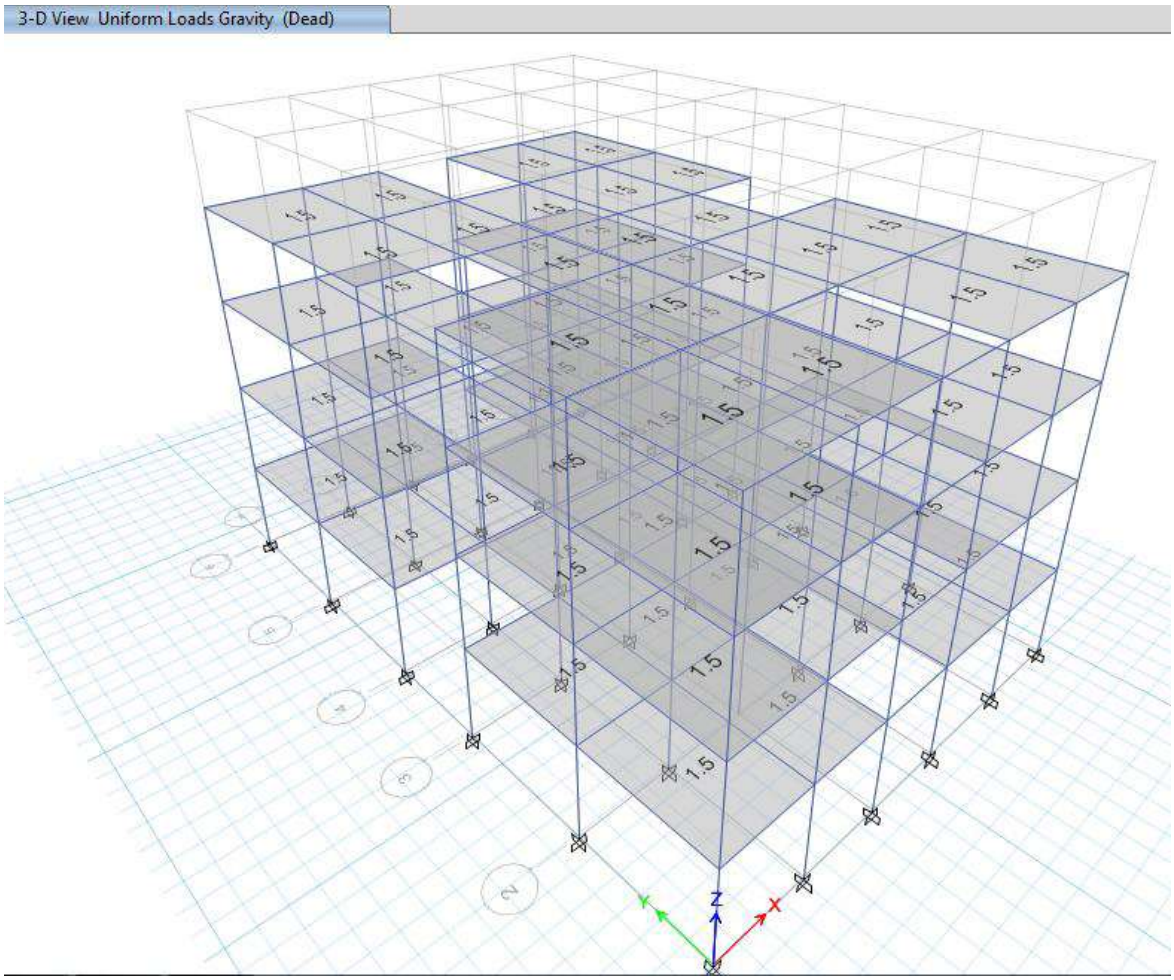


Fig: Showing Floor finish load assignment on slabs

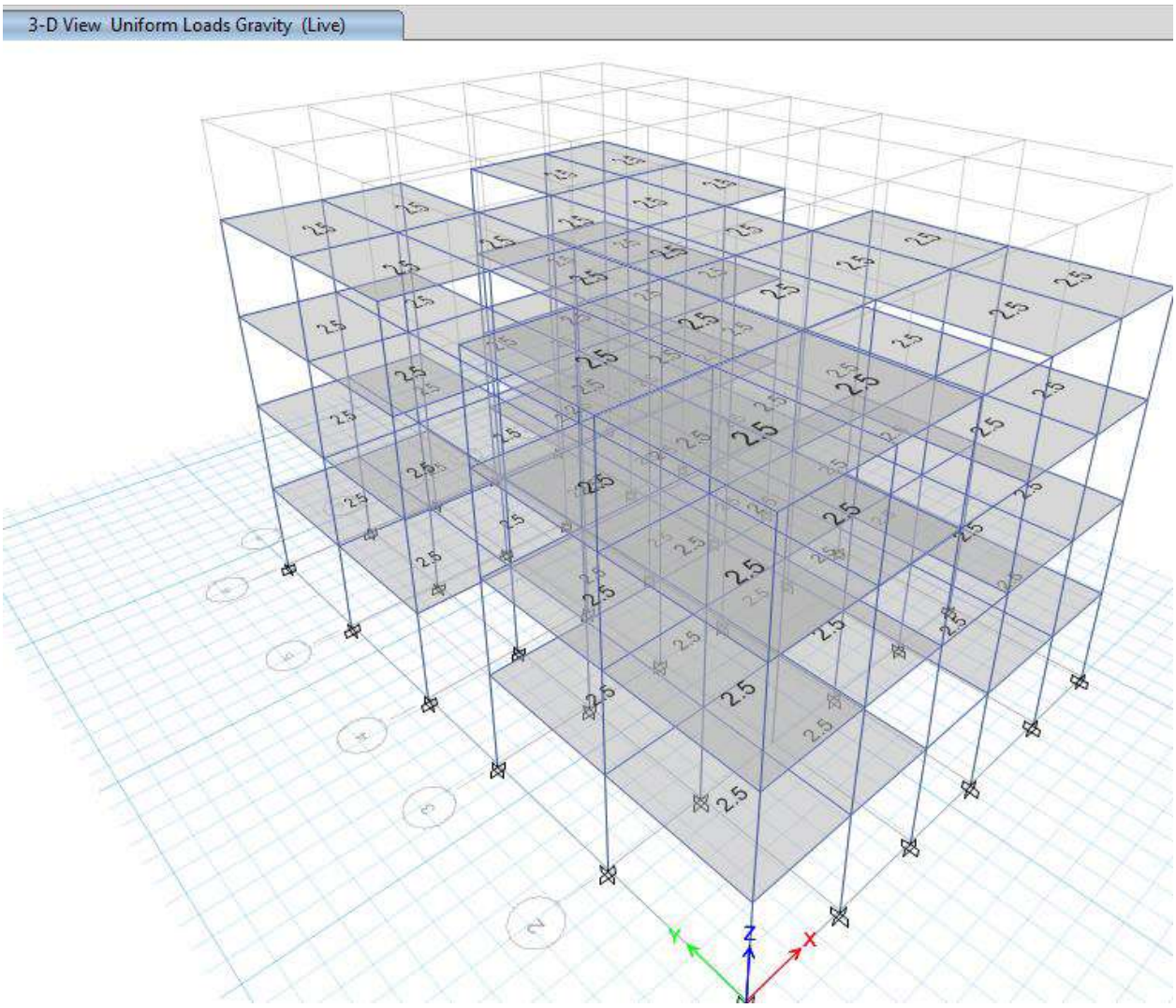
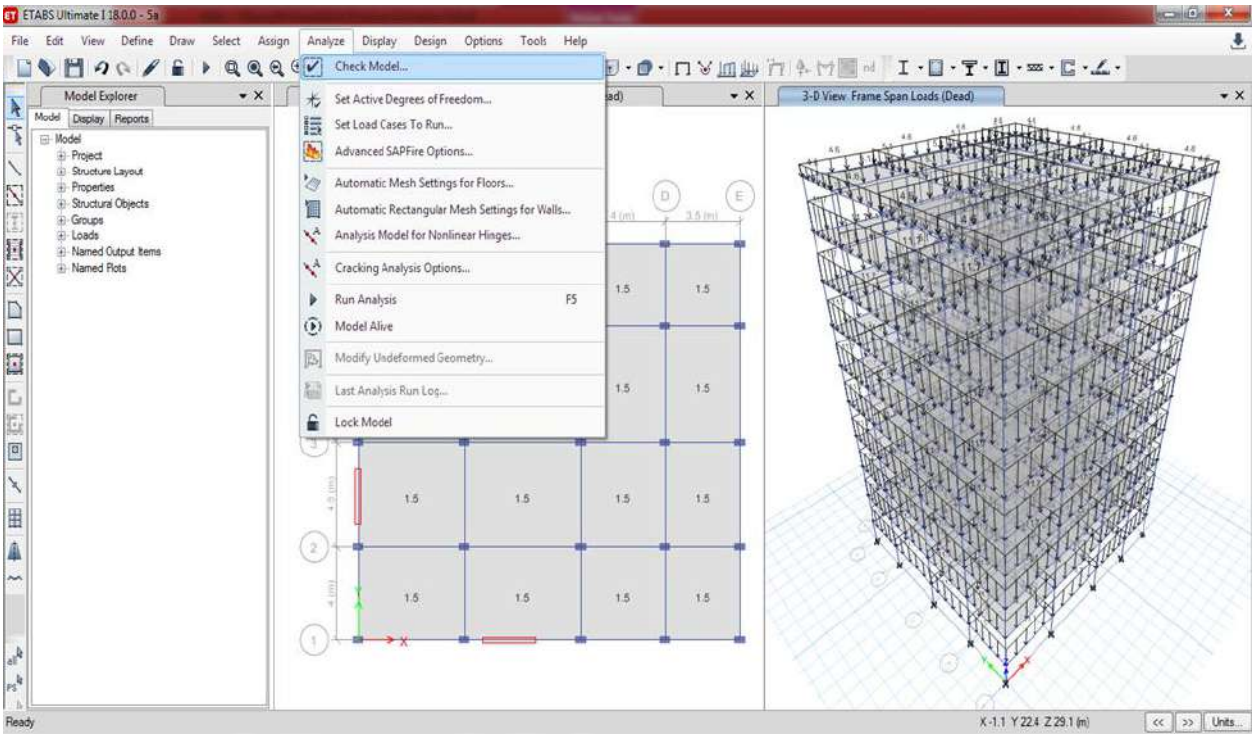


Fig: Showing Live load assignment on Slabs

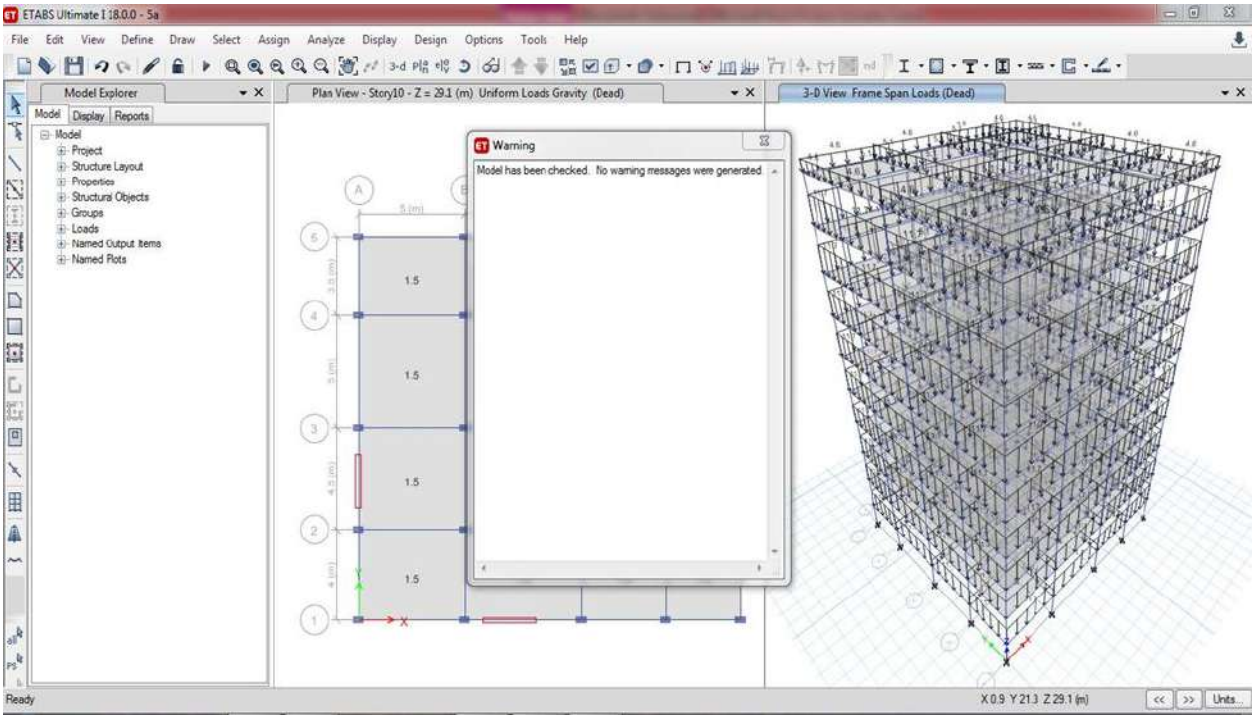
4. To define the load combinations go to **Define Menu> Load Combinations> Add Default Design Combos,** select **Concrete Frame Design** and click **OK**

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.

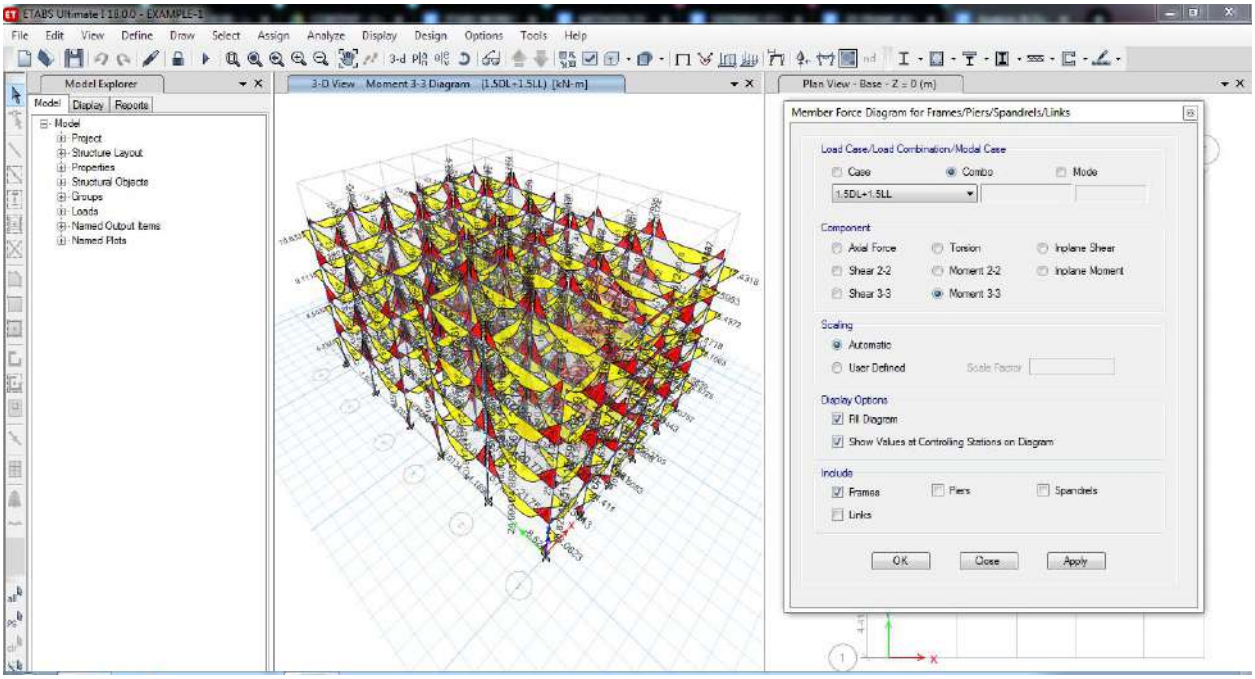


Fig: BMD

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

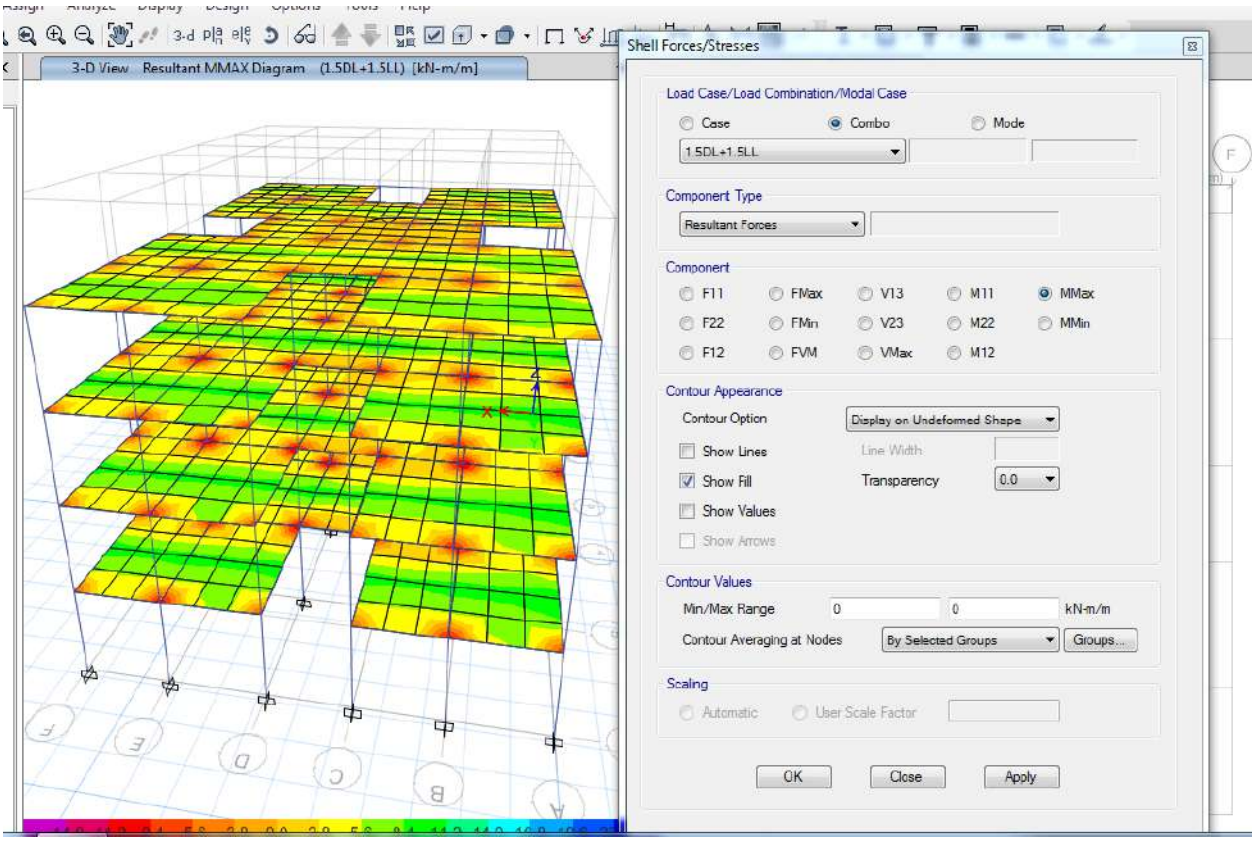


Fig: Max moments in Slab

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

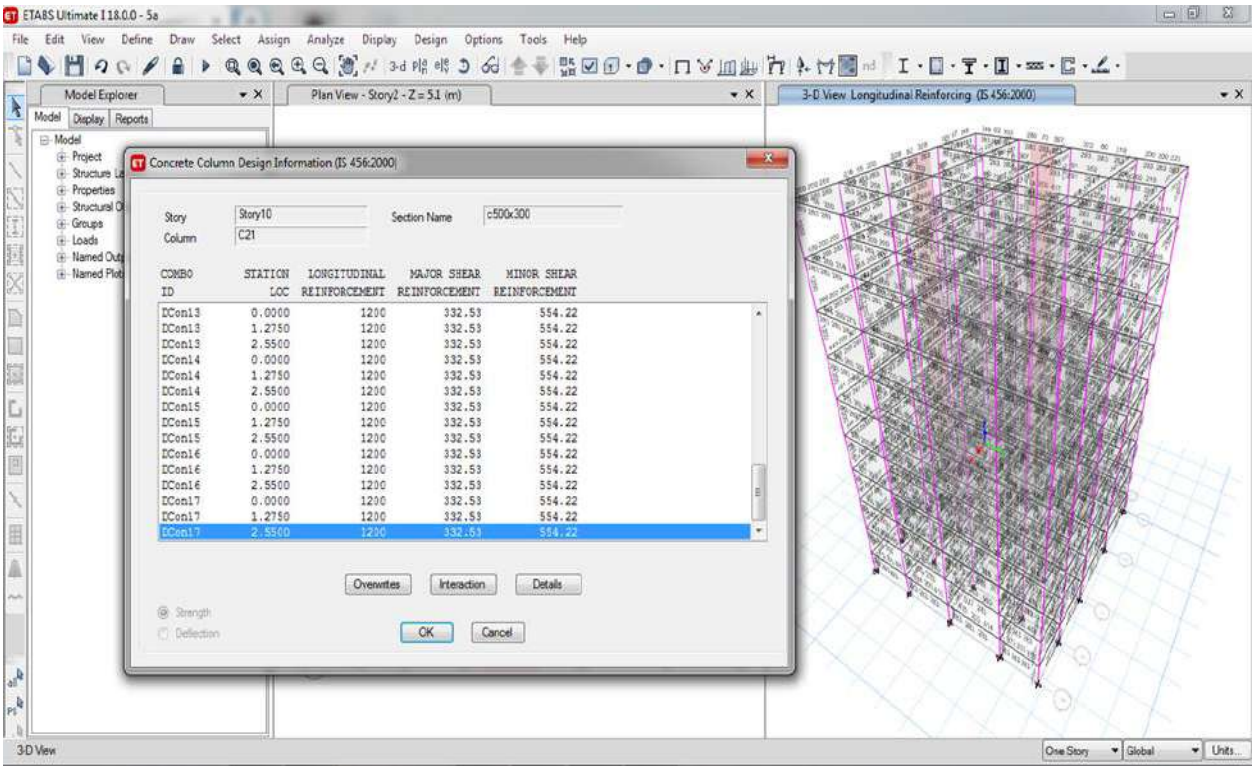


Fig: Design Details

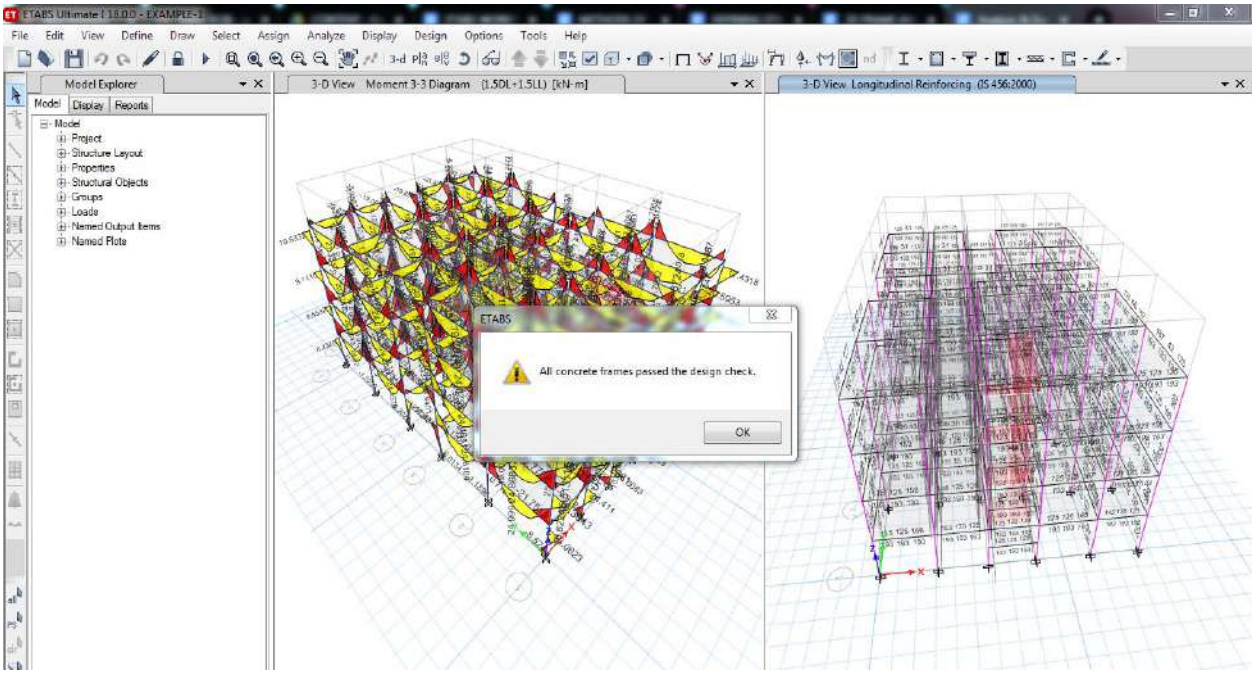


Fig: Design check