Andhra Pradesh State Skill Development Corporation

Extended Three-Dimensional Analysis of Building System

ETABS

2D Frame Analysis - II

2D FRAME ANALYSIS-2

Objective

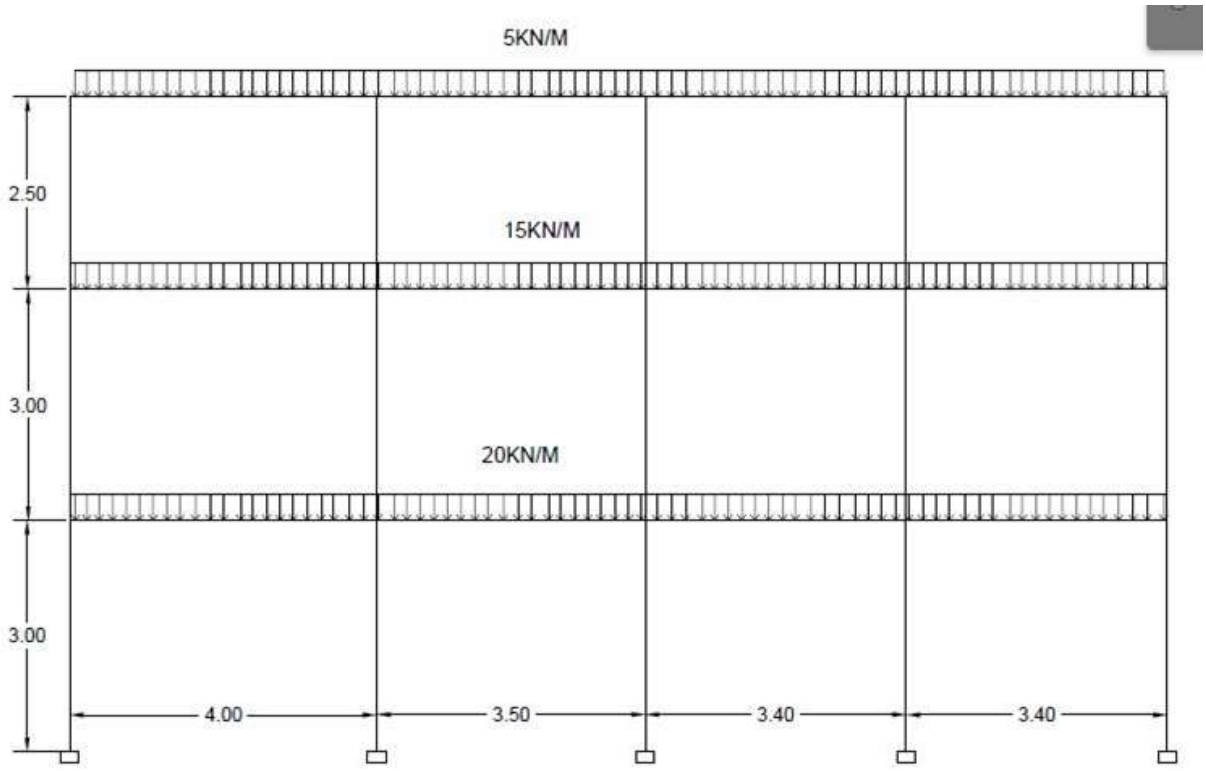
This chapter describes the step by step process of Analysis of 2D Frame with multiple bays in ETABS.

A multiple bay 2D frame contains more than one bay as shown in the following figure
Let's take an example

EXERCISE:

Analyze a 2-D frame as shown in Fig with the following considerations

Material Properties
Concrete: M25
Steel: HYSD415
Section Property
Beam Section: 230mm X 300mm
Column Sections: 300mm X 230mm



Procedure:

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**

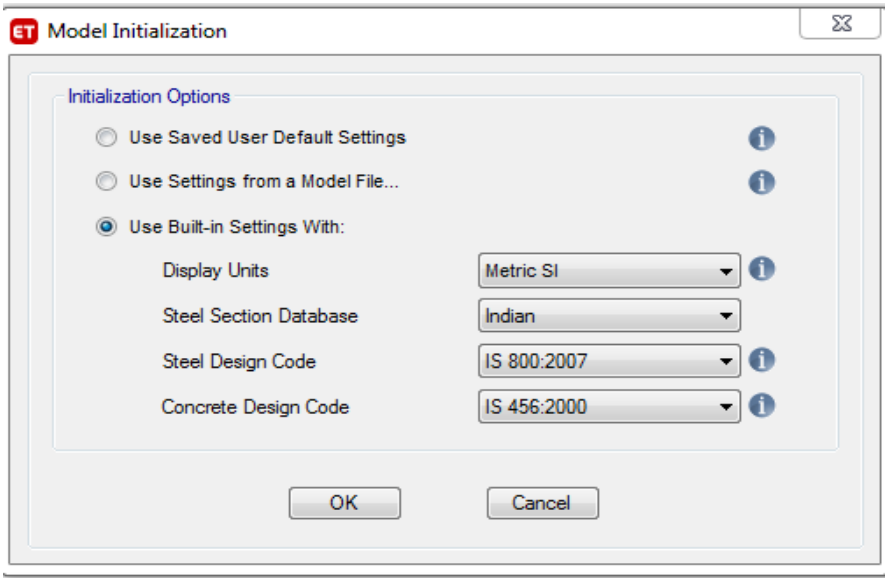


Fig: Model Initialization form

2. Specify the grid dimensions as 5-grids along X-axis, 1- grid along Y-axis and then enter spacing's under **Custom Grid Spacing** by clicking on **Edit Grid Data** in the **New Model Quick Templates** form.
3. To add the grid click on **Add** and to delete the grid click on **Delete** buttons, enter the spacing's in the spacing column.

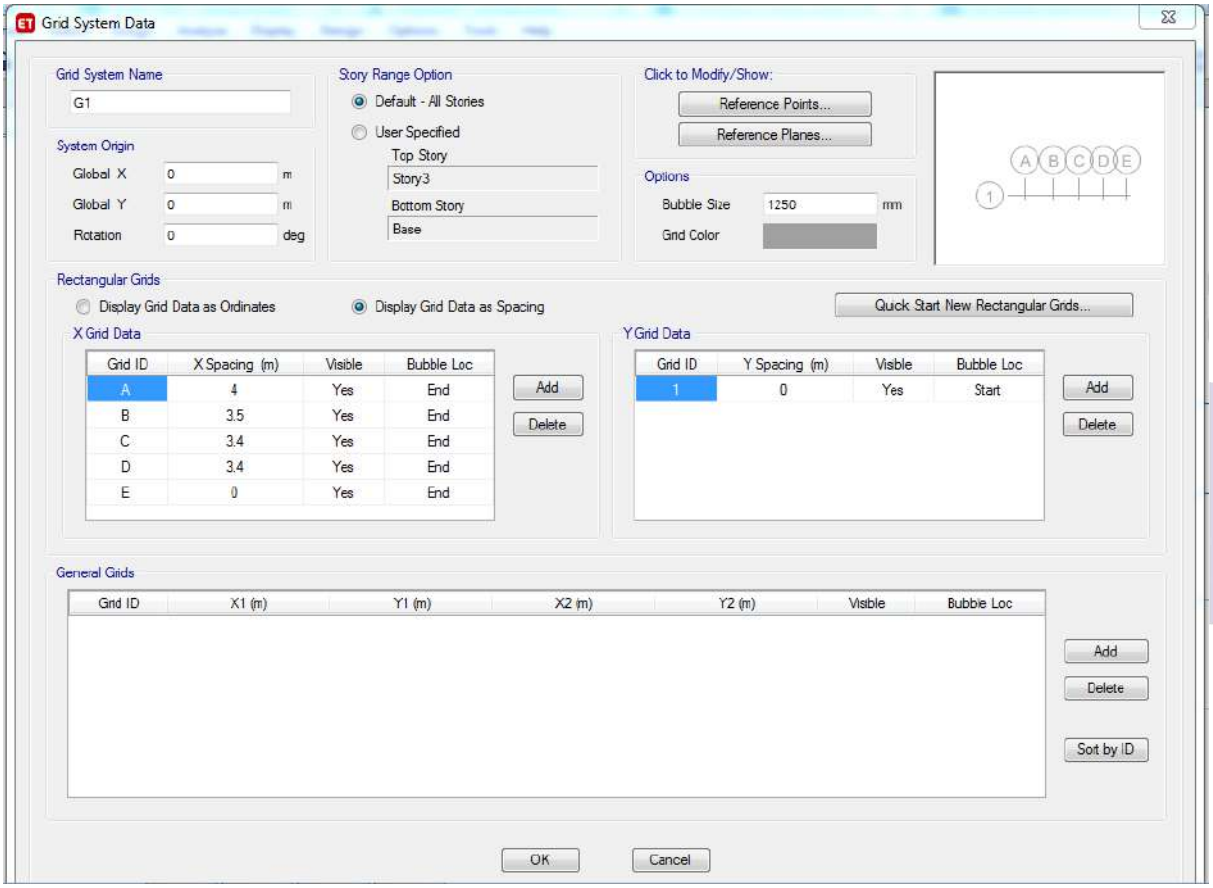


Fig: Grid System Data form

4. Specify the number of stories as 3 under **Simple Story Data** then select **Custom Story Data** and click on **Edit Story Data** to enter the story height.
5. In **Story data** form enter the story heights and then click on **OK**

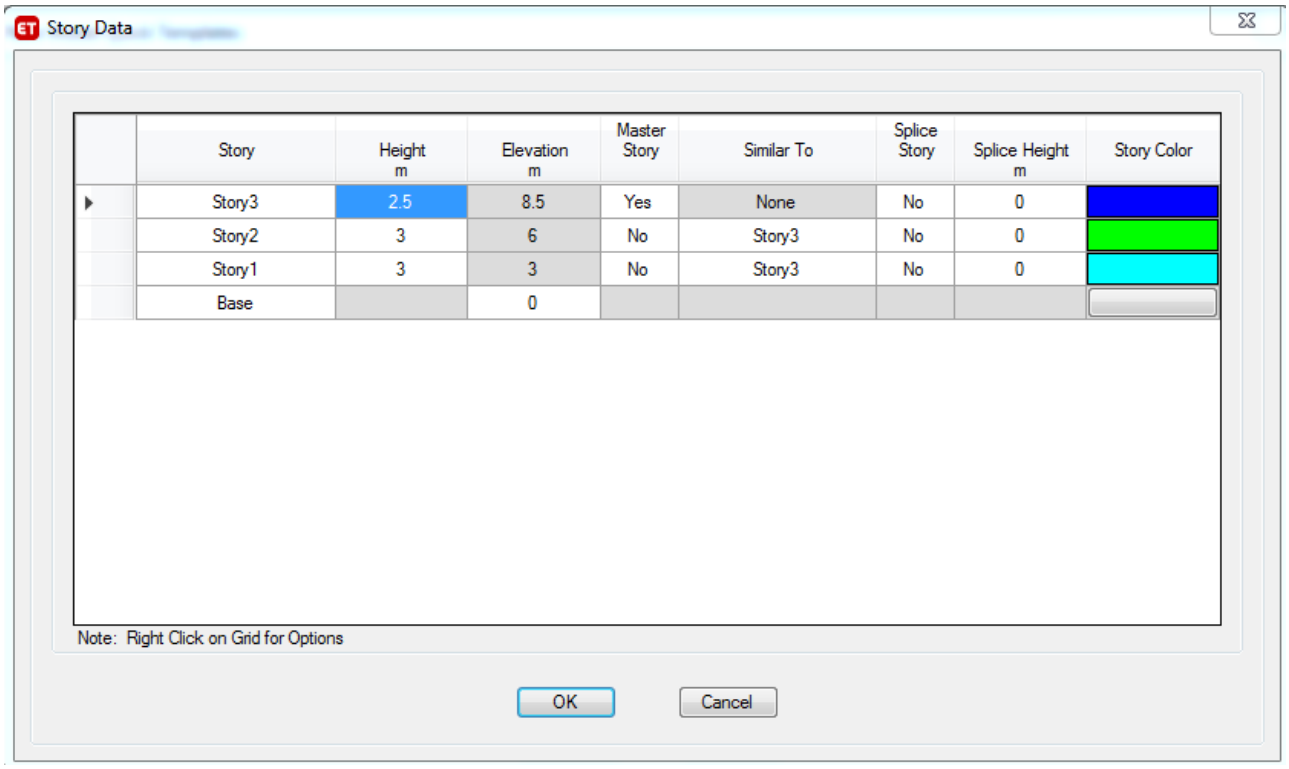


Fig: Story Data Form

6. Select **Grid only** template from **Add Structural Template.**

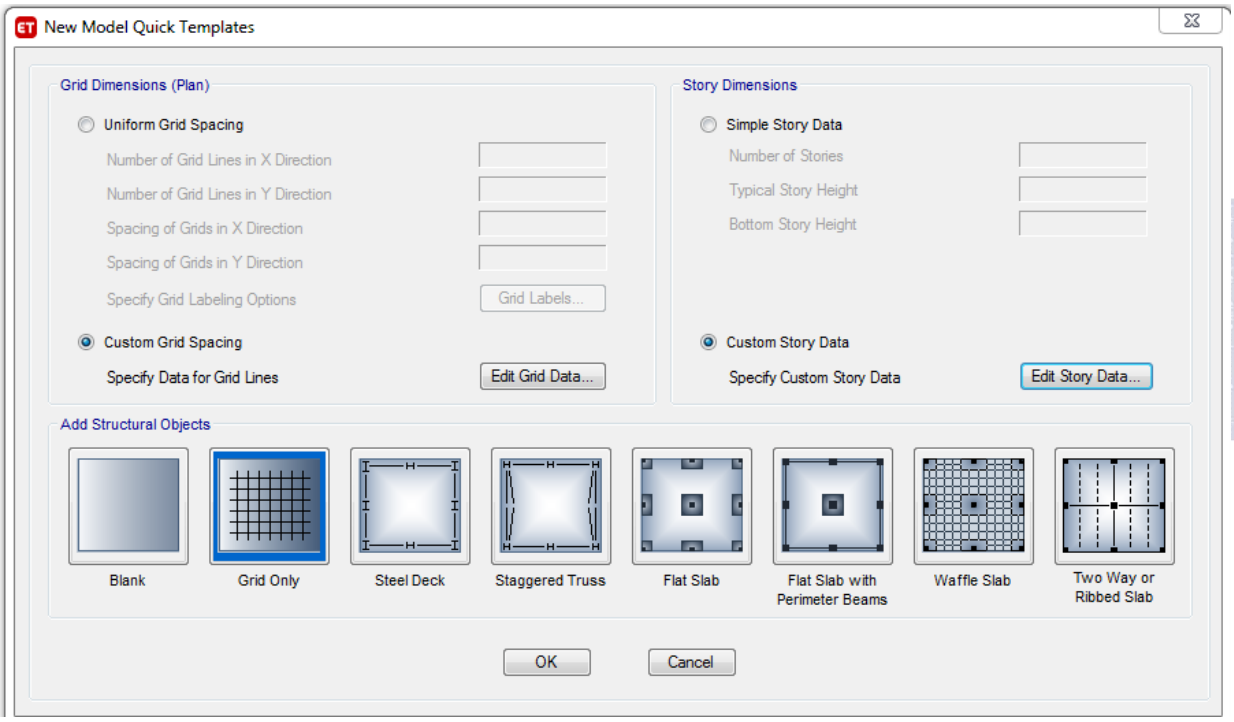


Fig: New Model Quick Templates form

7. By keeping remaining settings as default click **Ok.**
8. Go to **Define menu > Material Properties** , Click on **Add New Material** option and add M25 grade concrete and HYSD415 grade rebar using **Add New Material Property** form.

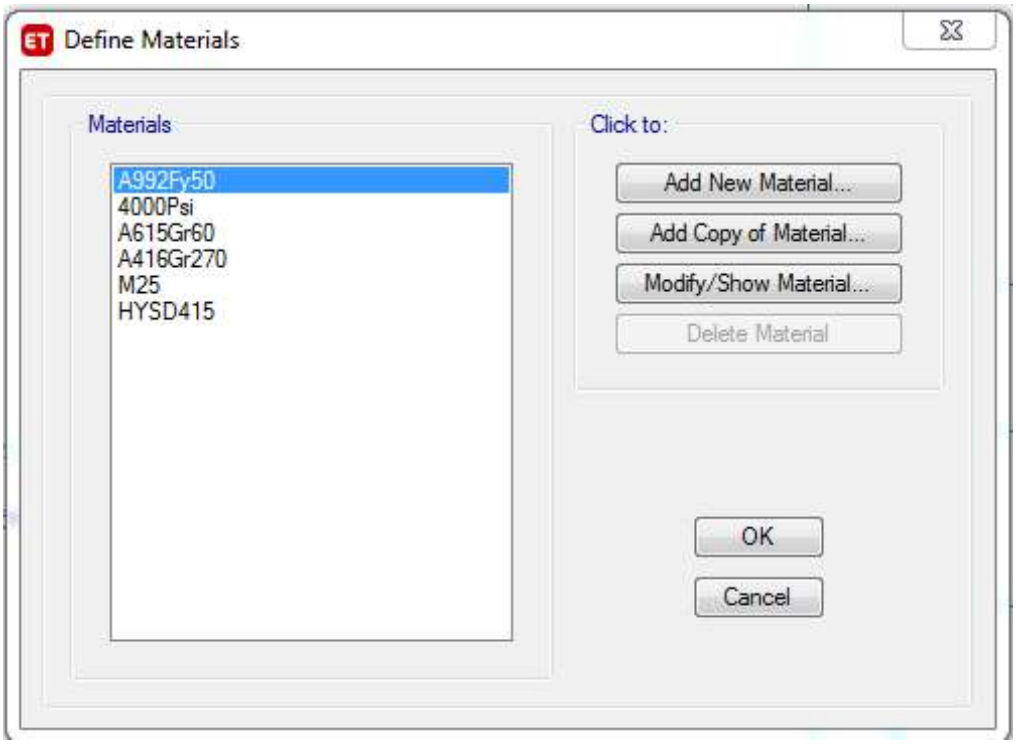


Fig: Define Materials form

9. 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

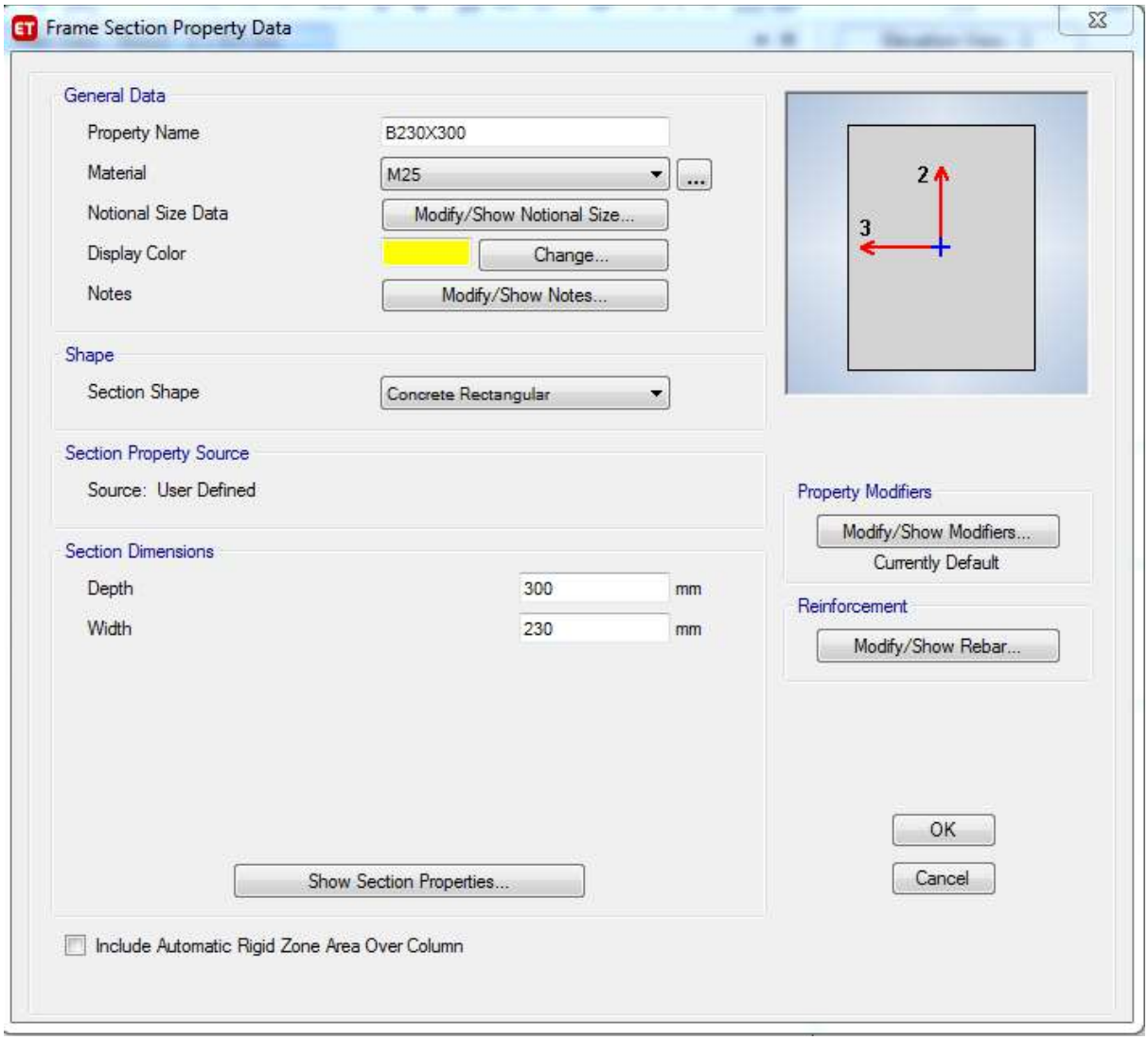


Fig: Frame Section Property Data form (Beam Definition)

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

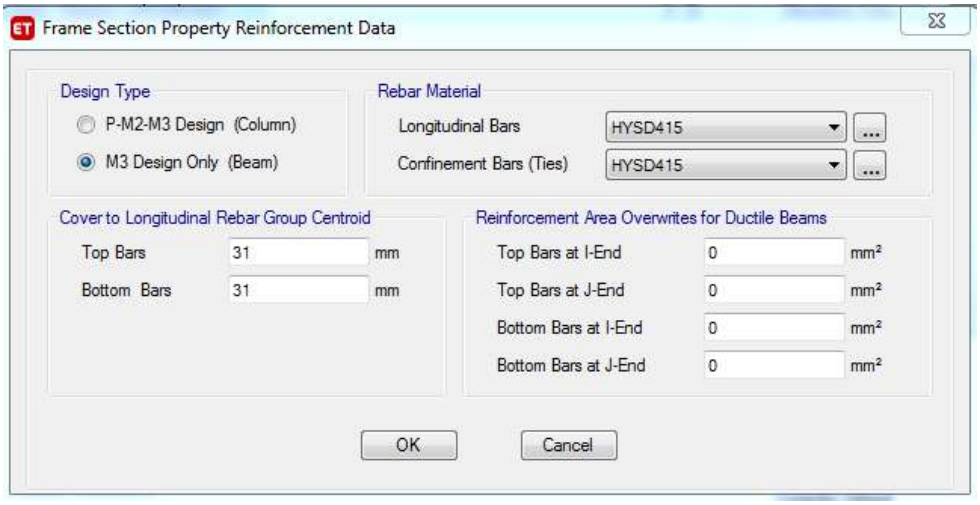


Fig: Frame Section Property Reinforcement Data form (Beam Definition)

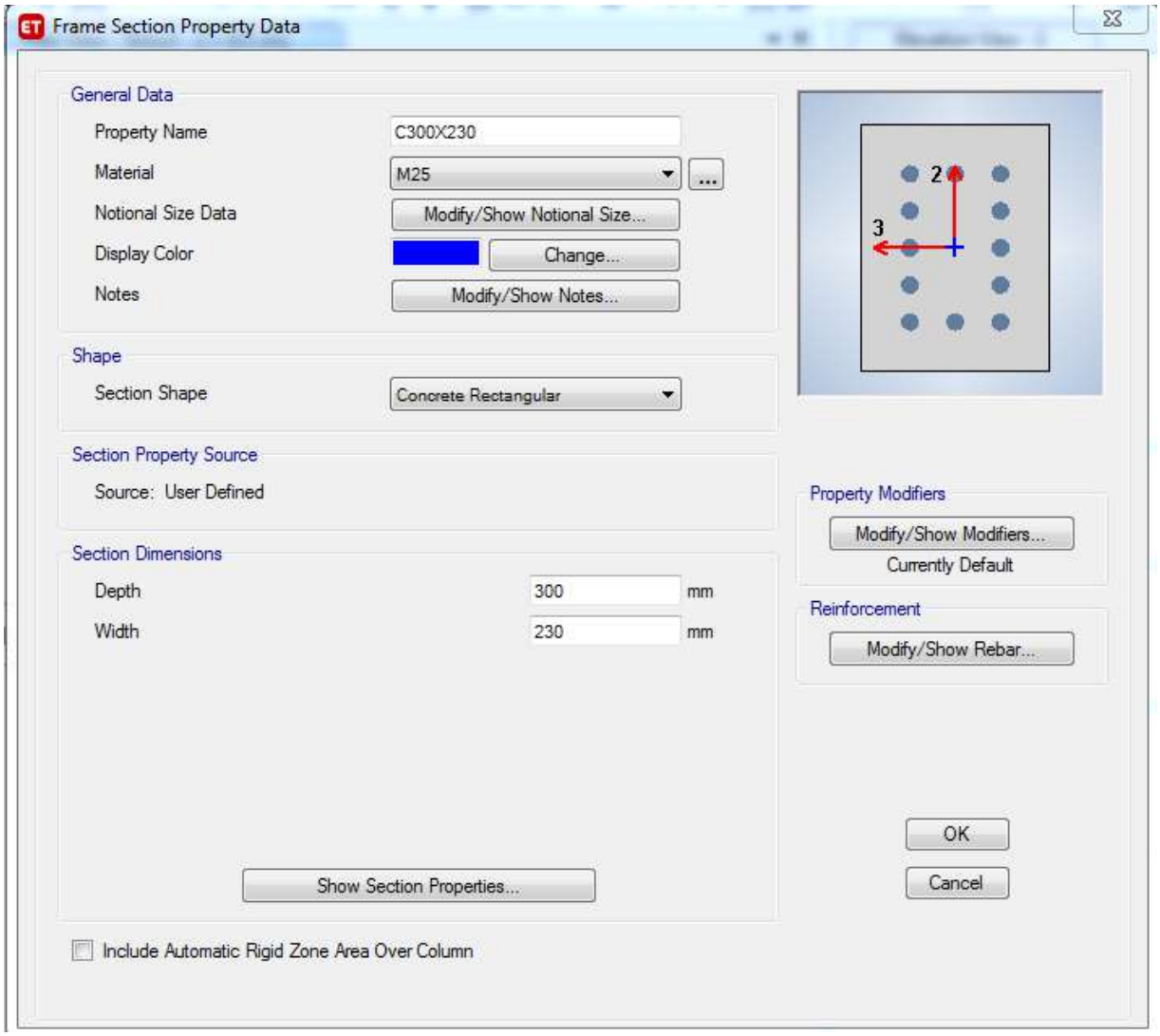


Fig: Frame Section Property Data form (Column Definition)

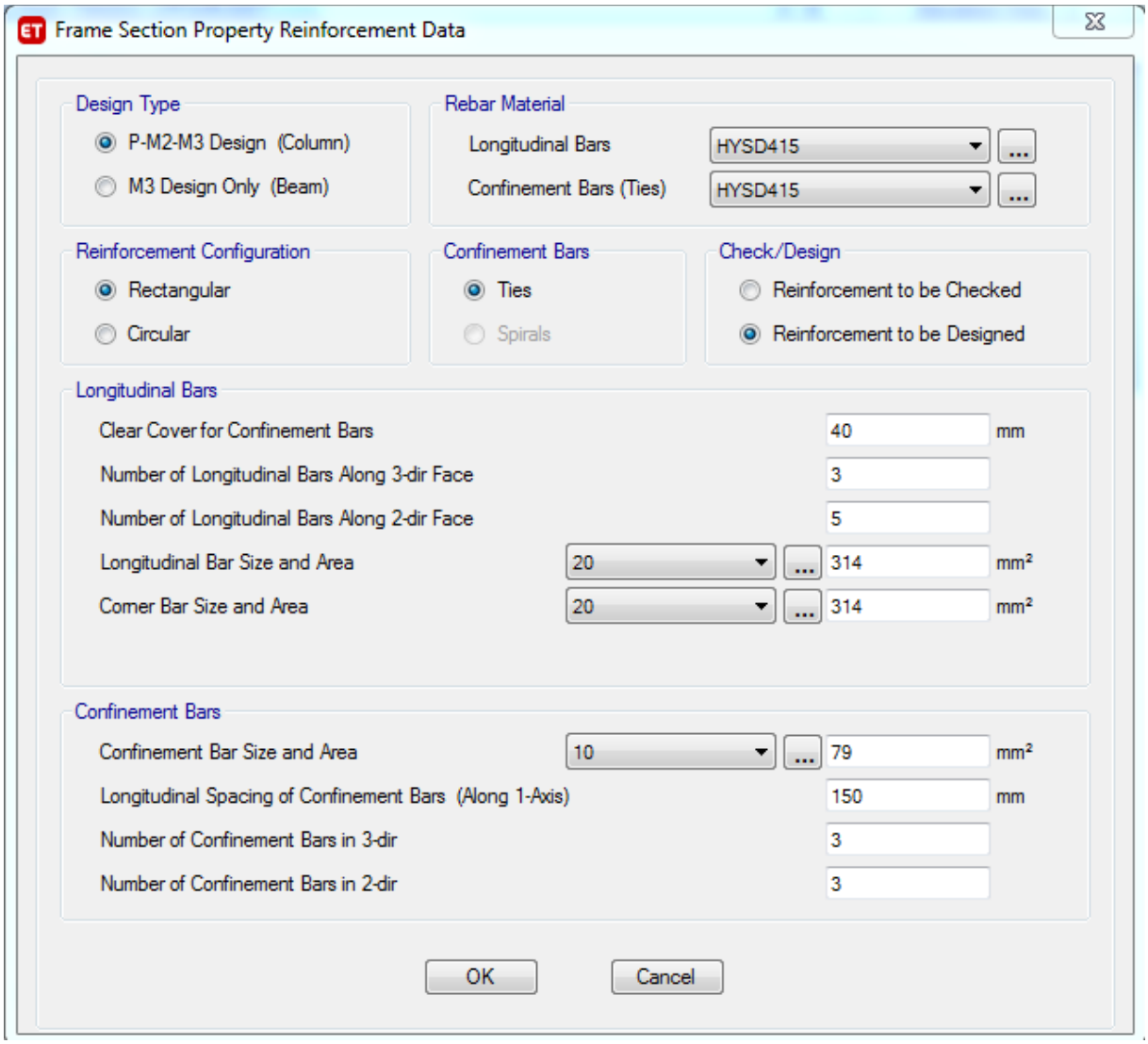
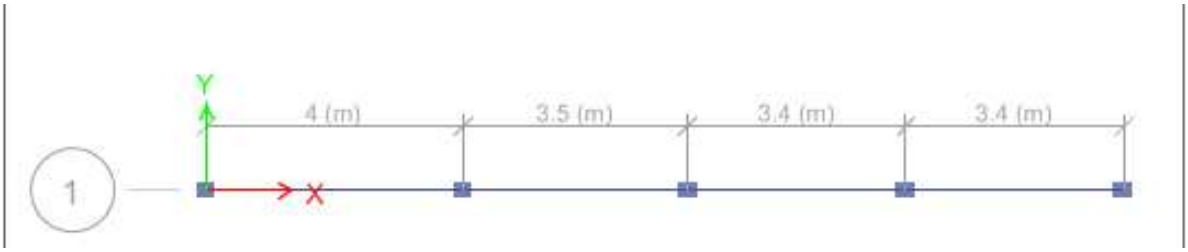
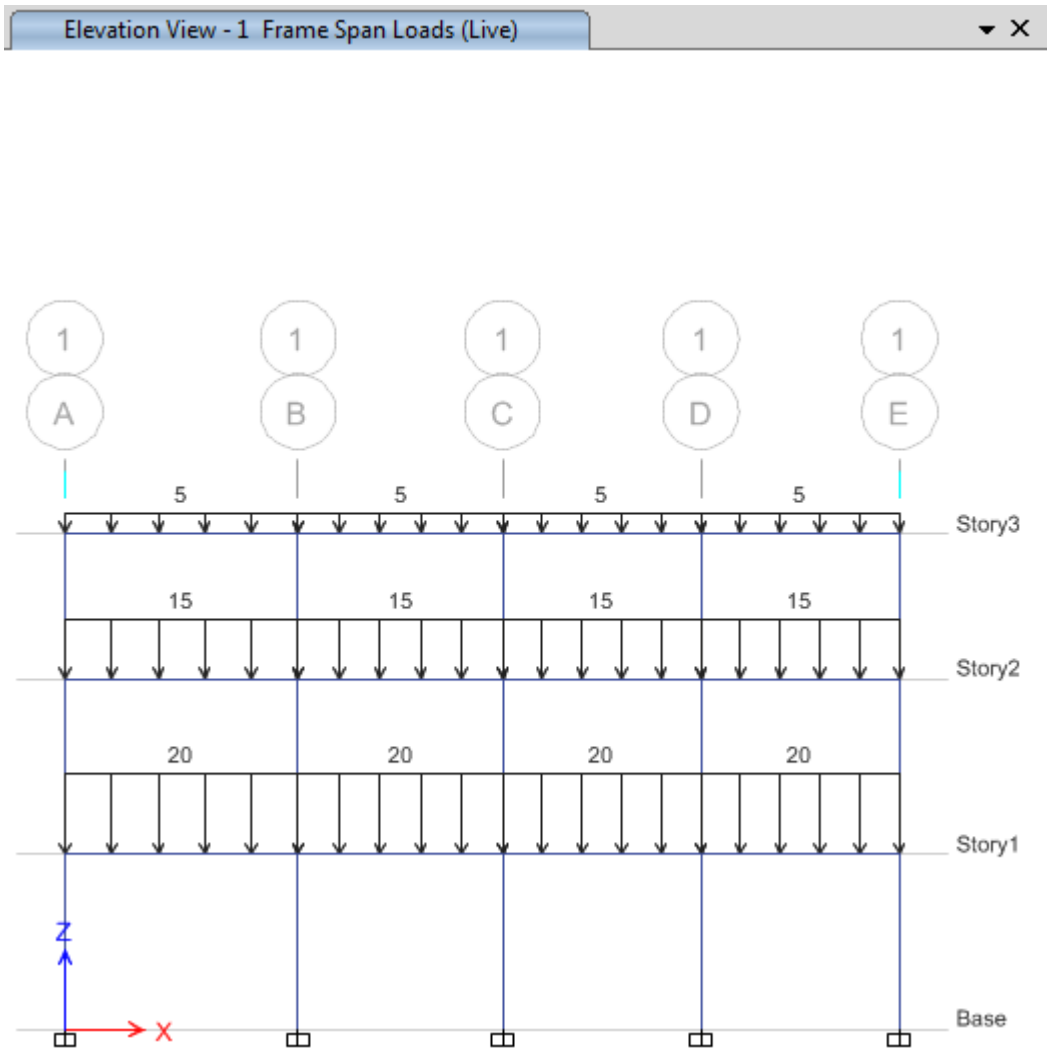


Fig: Frame Section Property Reinforcement Data form (Column Definition)

11. Keep the **Story Settings** as **All Stories** and Assign the beam & 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.



12. Open the plan view of story-1 > change the Story Settings to One Story > Select all beams in Story-1 and assign **UDL** of 20kN/m under Live load pattern using **Frame Loads**, similarly open the plan view of Story-2 > change the Story Settings to One Story >Select all beams in Story-2 and assign **UDL** of 15kN/m using **Frame Loads.** Same process is repeated for the 3rd Story as well.



13. To Assign Supports open grid-1 elevation and select the joints at base using windows selection, go to **Assign > Joints > Restraints**, and assign Fixed support to bottom joints.

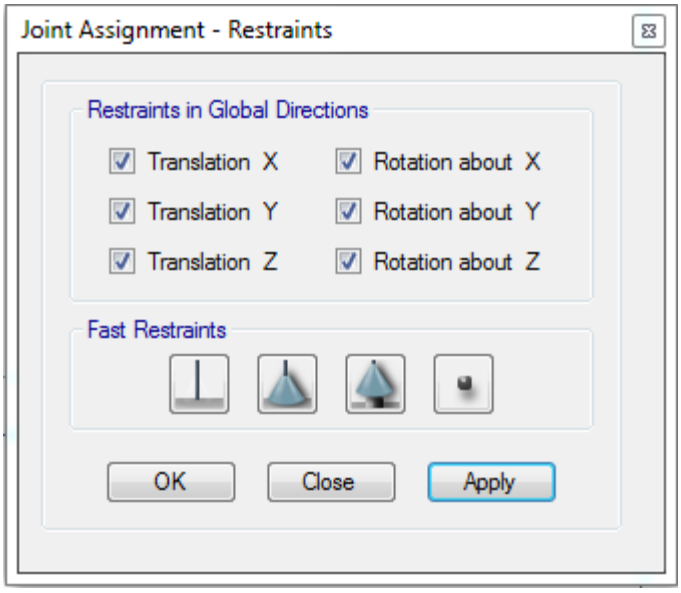


Fig: Joint Assignment-Restraints form

14. 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.**
15. 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.
16. 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**

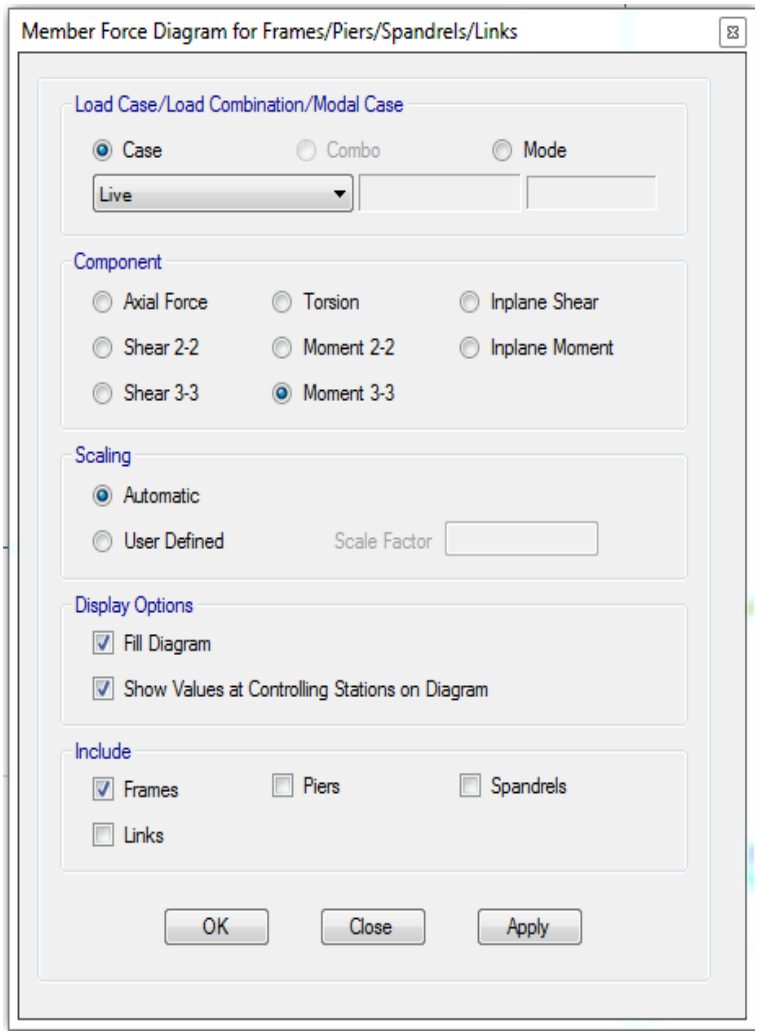


Fig: Member Force Diagram for Frames/Piers/Spandrels /links form

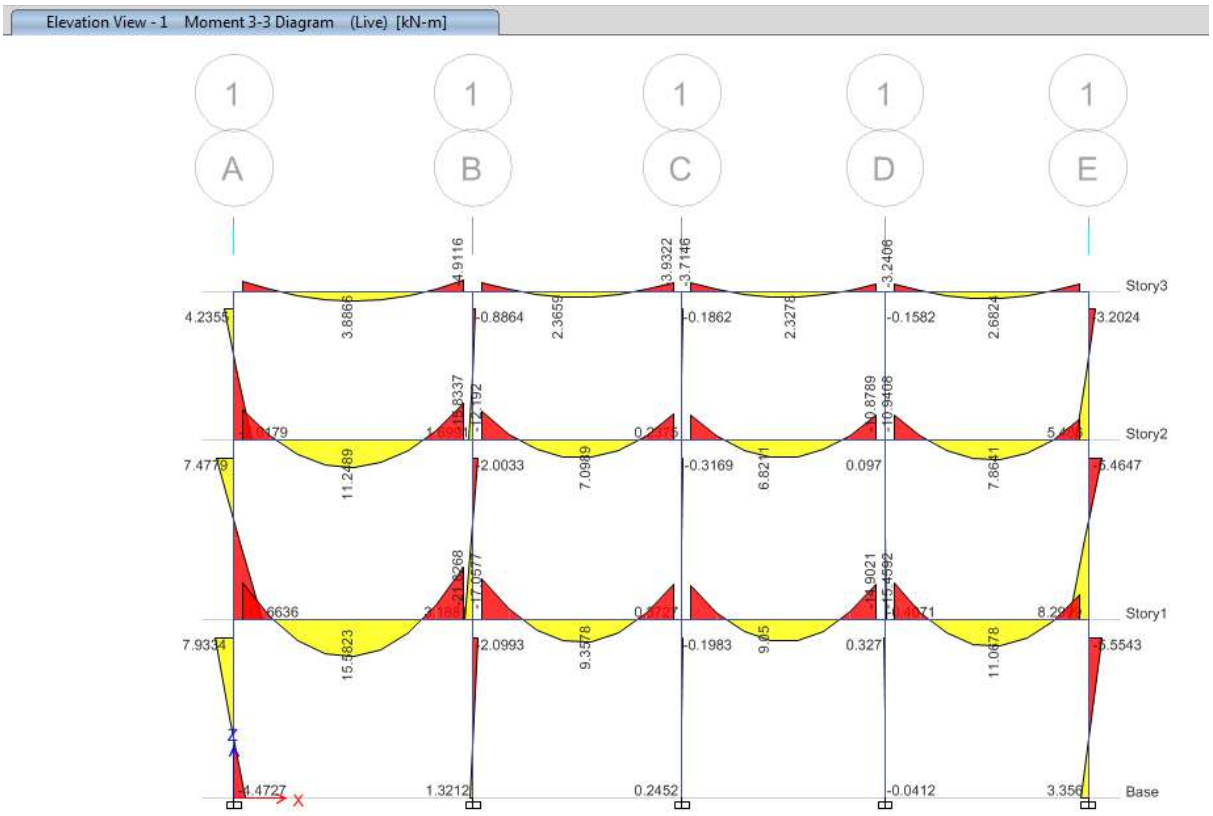


Fig: BMD

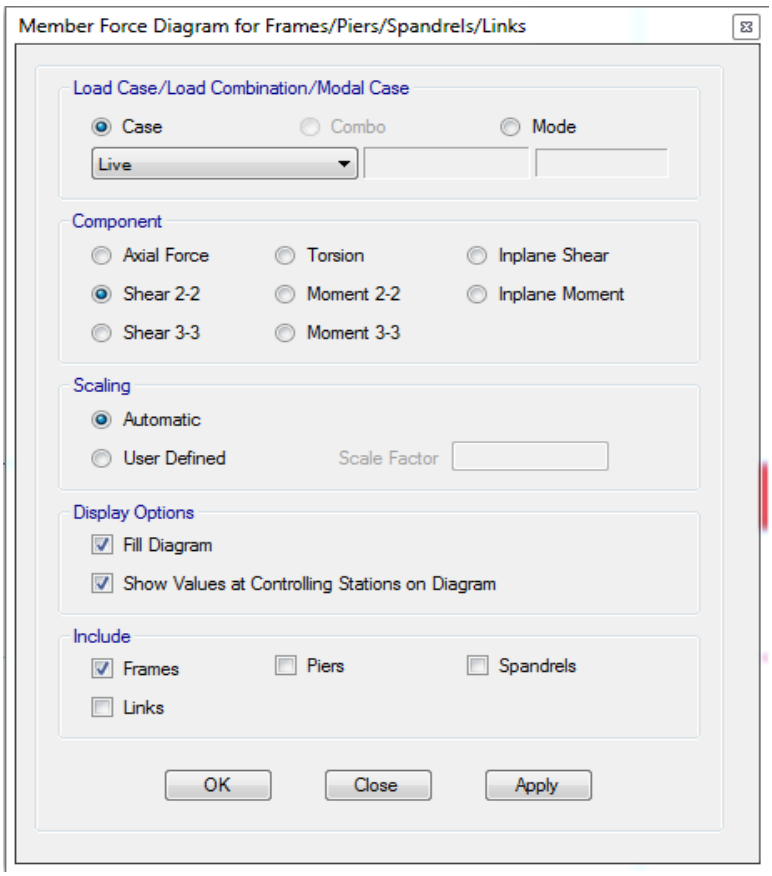


Fig: Member Force Diagram for Frames/Piers/Spandrels /links form

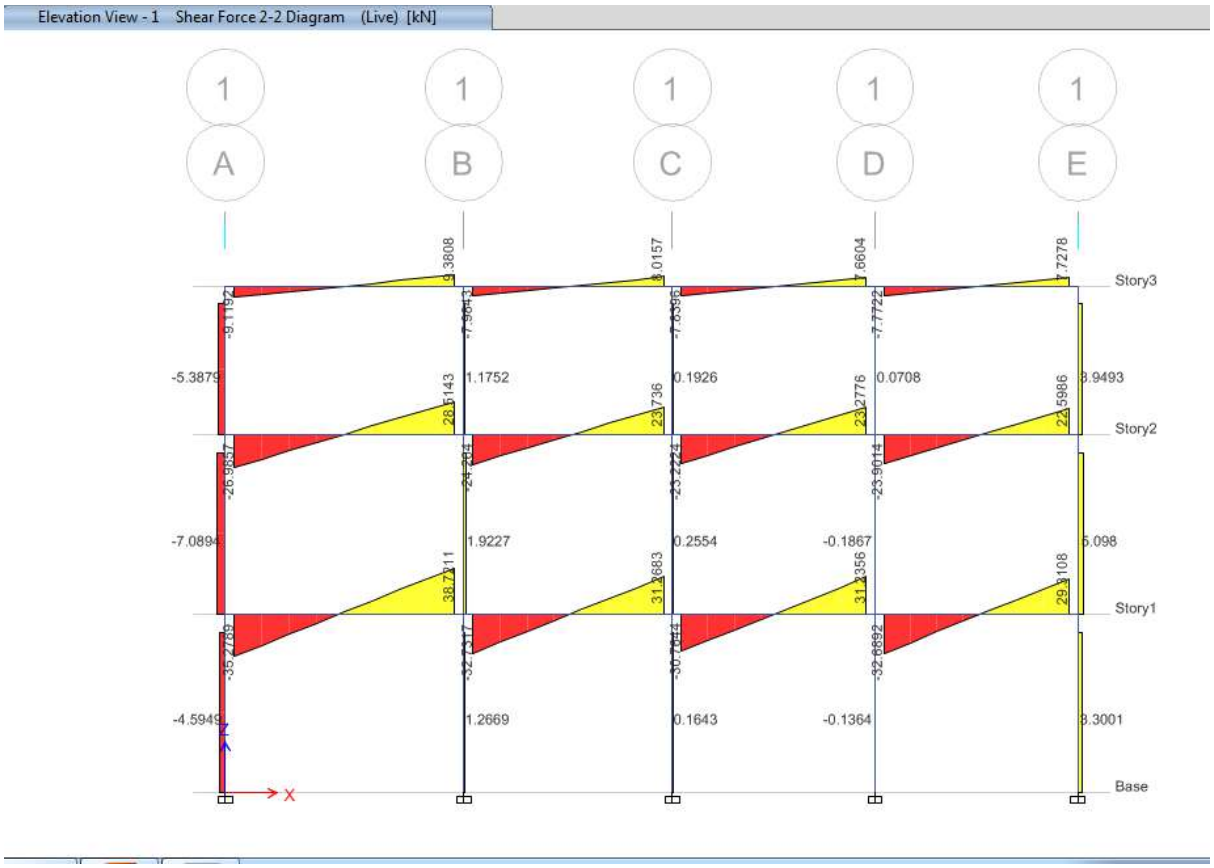


Fig: SFD

17. 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.

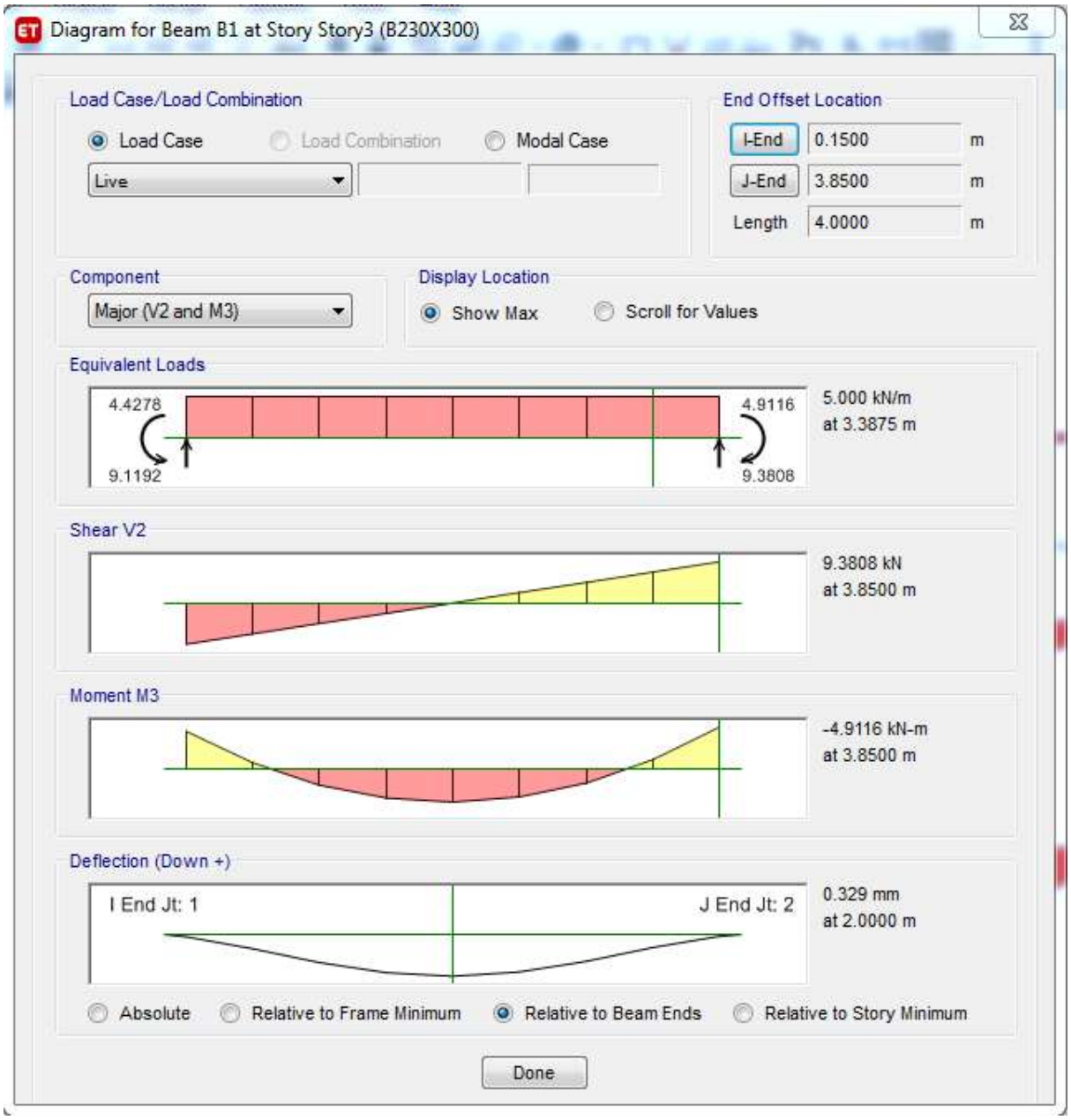


Fig: Diagram for Beam form

Here ends the process of 2D Frame Analysis in ETABS