



# Andhra Pradesh State Skill Development Corporation



## Extended Three-Dimensional Analysis of Building System

# ETABS

## 2D Frame Analysis - I

## 2D FRAME ANALYSIS-1

### Objective

This chapter describes the step by step process of 2D Portal Frame Analysis in ETABS.

**2D FRAME:** A single bay 2D frame contains 2 columns and a single beam in between as shown in the following frame

Let's take an example

### EXERCISE:

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

#### Material Properties

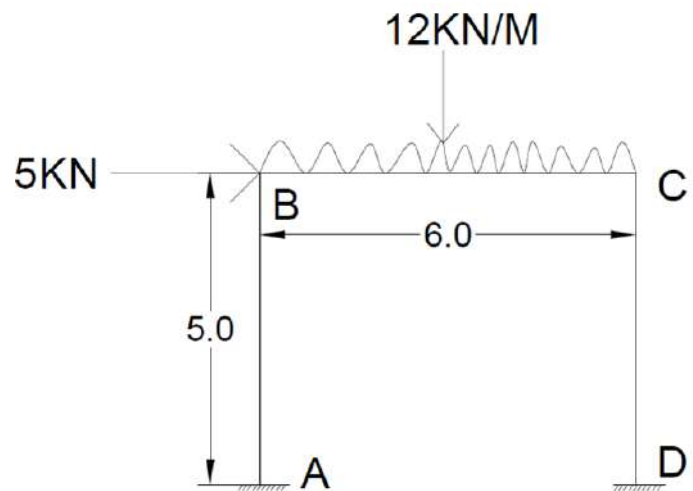
Concrete: M20

Steel: HYSD415

#### Section Property

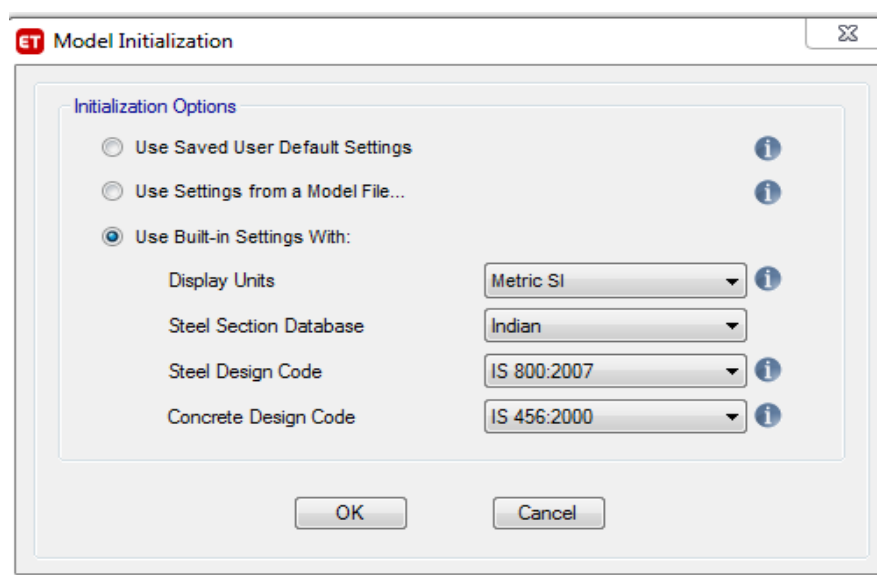
Beam Section: 230mm X 300mm

Column Sections: 230mm 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**

- Specify the grid dimensions as 2-grids along X-axis, 1- grid along Y-axis and spacing along X-axis as 6m under **Uniform Grid Spacing** in the **New Model Quick Templates** form.
- Specify the no.of stories as 1 and Bottom Story height as 5m under **Simple Story Data** in New **Model Quick Templates** form.
- Select **Grid only** template from **Add Structural Template**.

**ET New Model Quick Templates**

**Grid Dimensions (Plan)**

- ☒ **Uniform Grid Spacing**
  - Number of Grid Lines in X Direction: 2
  - Number of Grid Lines in Y Direction: 2
  - Spacing of Grids in X Direction: 6 m
  - Spacing of Grids in Y Direction: 8 m
  - Specify Grid Labeling Options: [Grid Labels...](#)
- ☐ **Custom Grid Spacing**
  - Specify Data for Grid Lines: [Edit Grid Data...](#)

**Story Dimensions**

- ☒ **Simple Story Data**
  - Number of Stories: 1
  - Typical Story Height: 3 m
  - Bottom Story Height: 5 m
- ☐ **Custom Story Data**
  - Specify Custom Story Data: [Edit Story Data...](#)

**Add Structural Objects**

Blank, **Grid Only**, Steel Deck, Staggered Truss, Flat Slab, Flat Slab with Perimeter Beams, Waffle Slab, Two Way or Ribbed Slab

[OK](#) [Cancel](#)

**Fig: New Model Quick Templates form**

- By keeping remaining settings as default click **Ok**.
- Go to **Define menu > Material Properties** , Click on **Add New Material** option and add M20 grade concrete and HYSD415 grade rebar using **Add New Material Property** form.

**ET Define Materials**

**Materials**

- A992Fy50
- 4000Psi
- A615Gr60
- A416Gr270
- M20
- HYSD415

**Click to:**

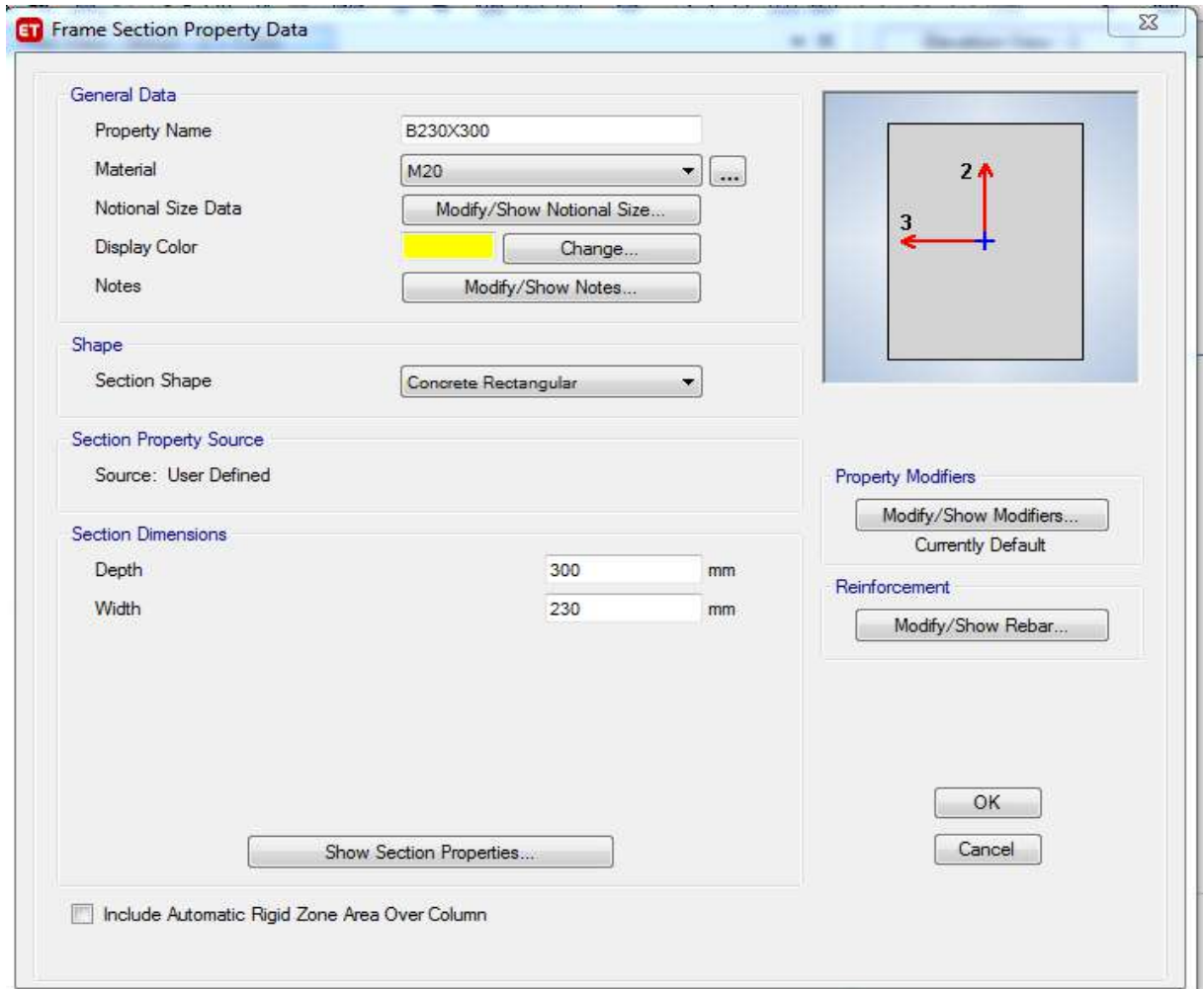
[Add New Material...](#)  
[Add Copy of Material...](#)  
[Modify/Show Material...](#)  
[Delete Material](#)

[OK](#)  
[Cancel](#)

**Fig: Define Materials form**



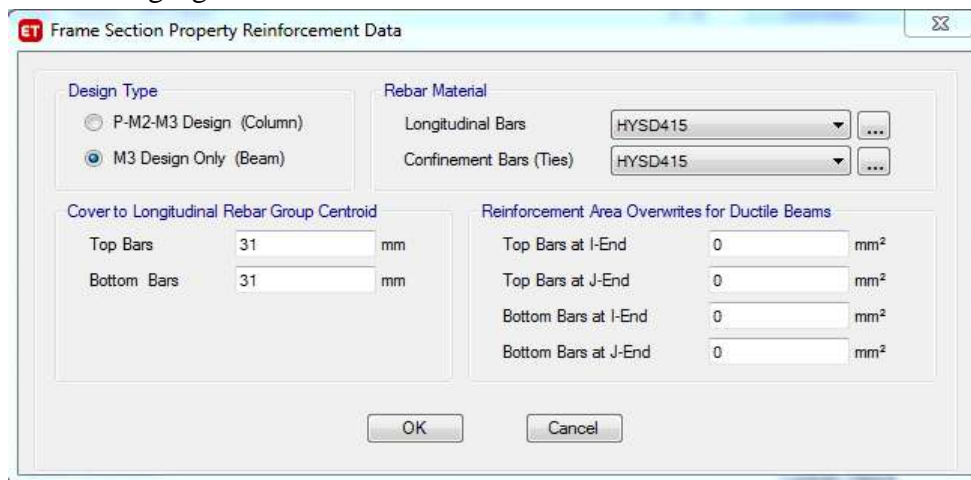
- Go to **Define menu >Section Properties > Frame Sections**, click on **Add New Property** and specify the parameters in **Frame Section Property Data** form as shown in following figures



The screenshot shows the 'Frame Section Property Data' dialog box. It contains several sections: 'General Data' with fields for Property Name (B230X300), Material (M20), Notional Size Data (Modify/Show Notional Size...), Display Color (yellow), and Notes (Modify/Show Notes...); 'Shape' with a dropdown for Section Shape (Concrete Rectangular); 'Section Property Source' with a dropdown for Source (User Defined); 'Section Dimensions' with fields for Depth (300 mm) and Width (230 mm); 'Property Modifiers' with a button for Modify/Show Modifiers... (Currently Default); 'Reinforcement' with a button for Modify/Show Rebar...; and a checkbox for 'Include Automatic Rigid Zone Area Over Column'. A 3D coordinate system diagram is also visible on the right.

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

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



The screenshot shows the 'Frame Section Property Reinforcement Data' dialog box. It contains several sections: 'Design Type' with radio buttons for P-M2-M3 Design (Column) and M3 Design Only (Beam); 'Rebar Material' with dropdowns for Longitudinal Bars (HYSD415) and Confinement Bars (Ties) (HYSD415); 'Cover to Longitudinal Rebar Group Centroid' with fields for Top Bars (31 mm) and Bottom Bars (31 mm); and 'Reinforcement Area Overwrites for Ductile Beams' with fields for Top Bars at I-End, Top Bars at J-End, Bottom Bars at I-End, and Bottom Bars at J-End, all set to 0 mm². There are OK and Cancel buttons at the bottom.

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

**Frame Section Property Data**

**General Data**

Property Name: C230X230

Material: M20

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: 230 mm

Width: 230 mm

**Property Modifiers**

Modify/Show Modifiers...  
Currently Default

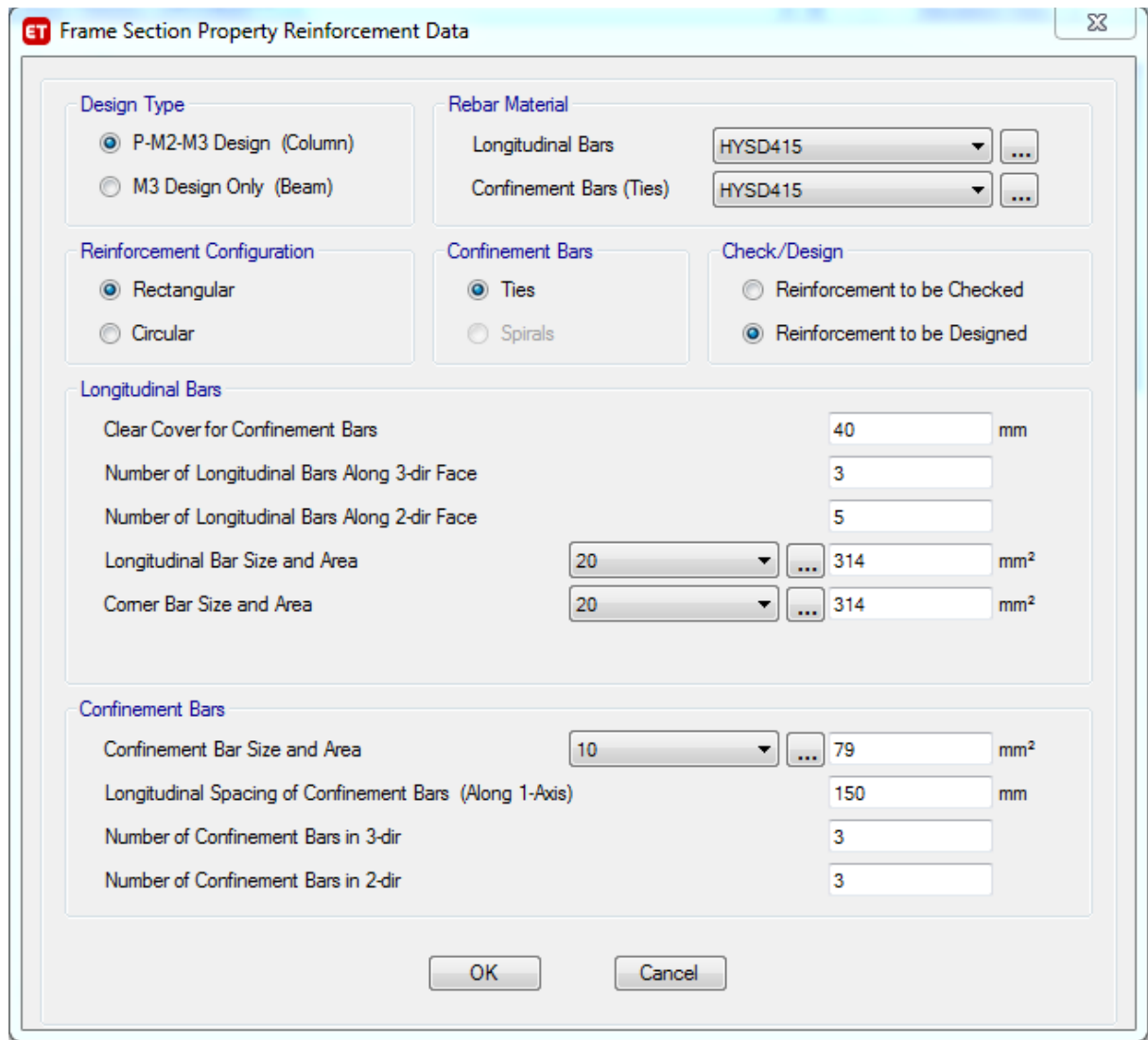
**Reinforcement**

Modify/Show Rebar...

☐ Include Automatic Rigid Zone Area Over Column

Show Section Properties... OK Cancel

**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: HYSD415

Confinement Bars (Ties): HYSD415

**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>

Corner 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

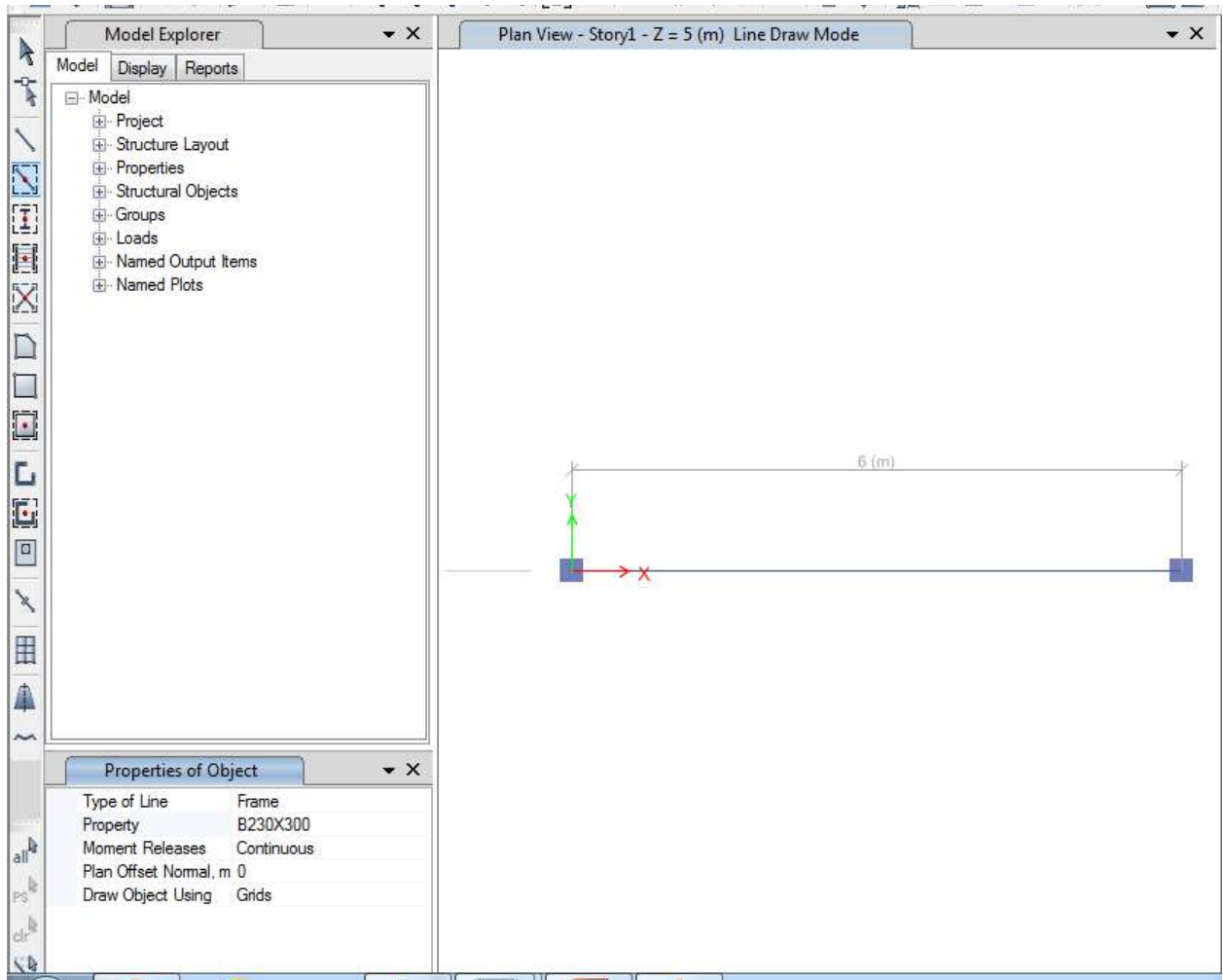
Number of Confinement Bars in 3-dir: 3

Number of Confinement Bars in 2-dir: 3

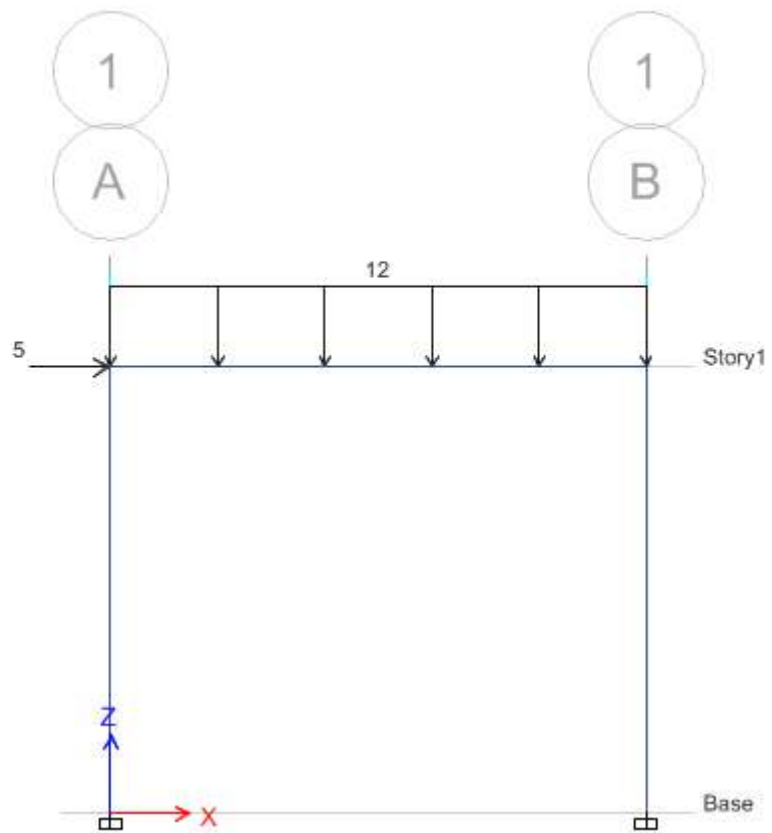
OK Cancel

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

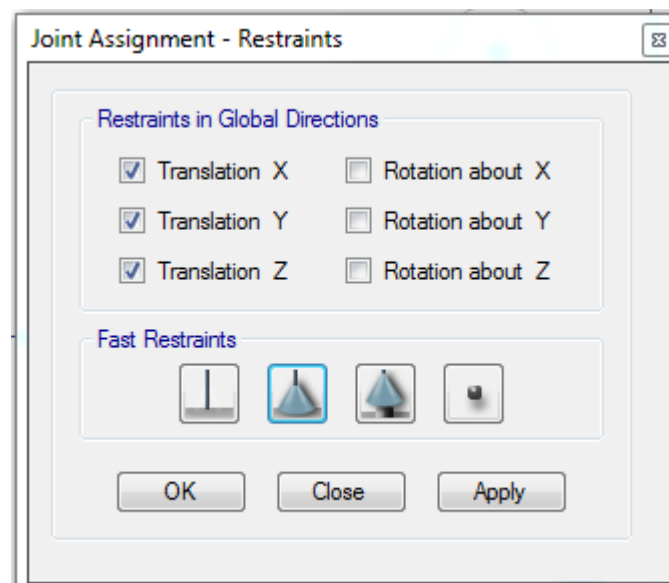
- Assign the beam & columns to grid by using either **Draw Beam/Column/brace (Plan, Elev, 3D)** or **Quick Draw beam Beams/columns (Plan, Elev, 3D)** tool from **Draw** menu.



10. Select the beam and assign **UDL** of 12kN/m using **Frame Loads**, select the Grid A1 intersection joint and assign **Force** of 5kN using **Joint loads** in **Assign** menu



11. To Assign Supports select the joints one by one, go to Assign > Joints > Restraints, and assign Fixed support to bottom joints.

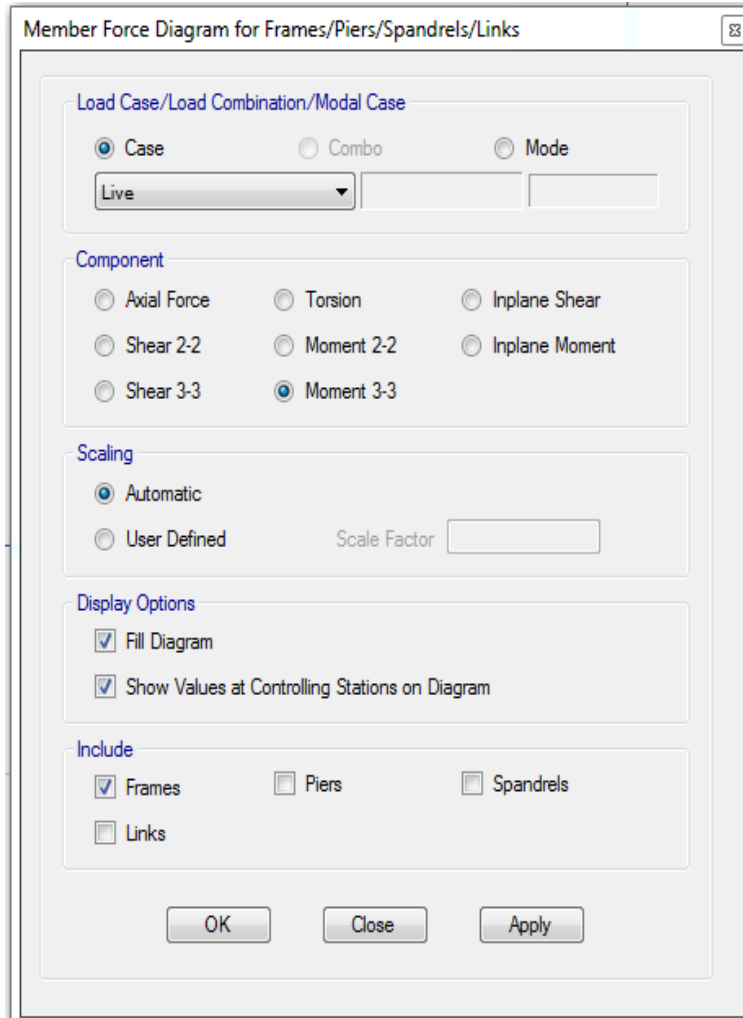


**Fig: Joint Assignment-Restraints form**

12. 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**.



13. 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.
14. To check the results like BMD or SF 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**

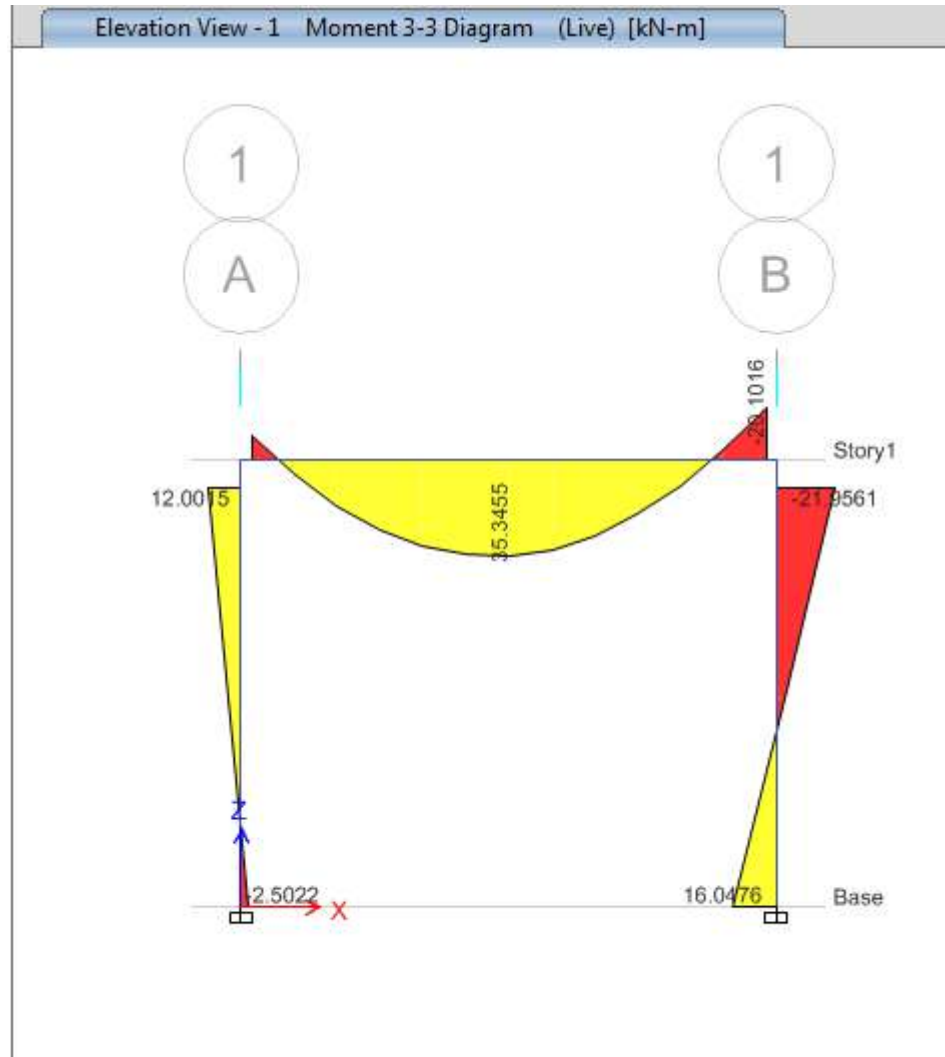


Fig: BMD

Member Force Diagram for Frames/Piers/Spandrels/Links

Load Case/Load Combination/Modal Case

☒ Case ☐ Combo ☐ Mode

Live

Component

☐ Axial Force ☐ Torsion ☐ Inplane Shear

☒ Shear 2-2 ☐ Moment 2-2 ☐ Inplane Moment

☐ Shear 3-3 ☐ Moment 3-3

Scaling

☒ Automatic ☐ User Defined Scale Factor

Display Options

☒ Fill Diagram ☒ Show Values at Controlling Stations on Diagram

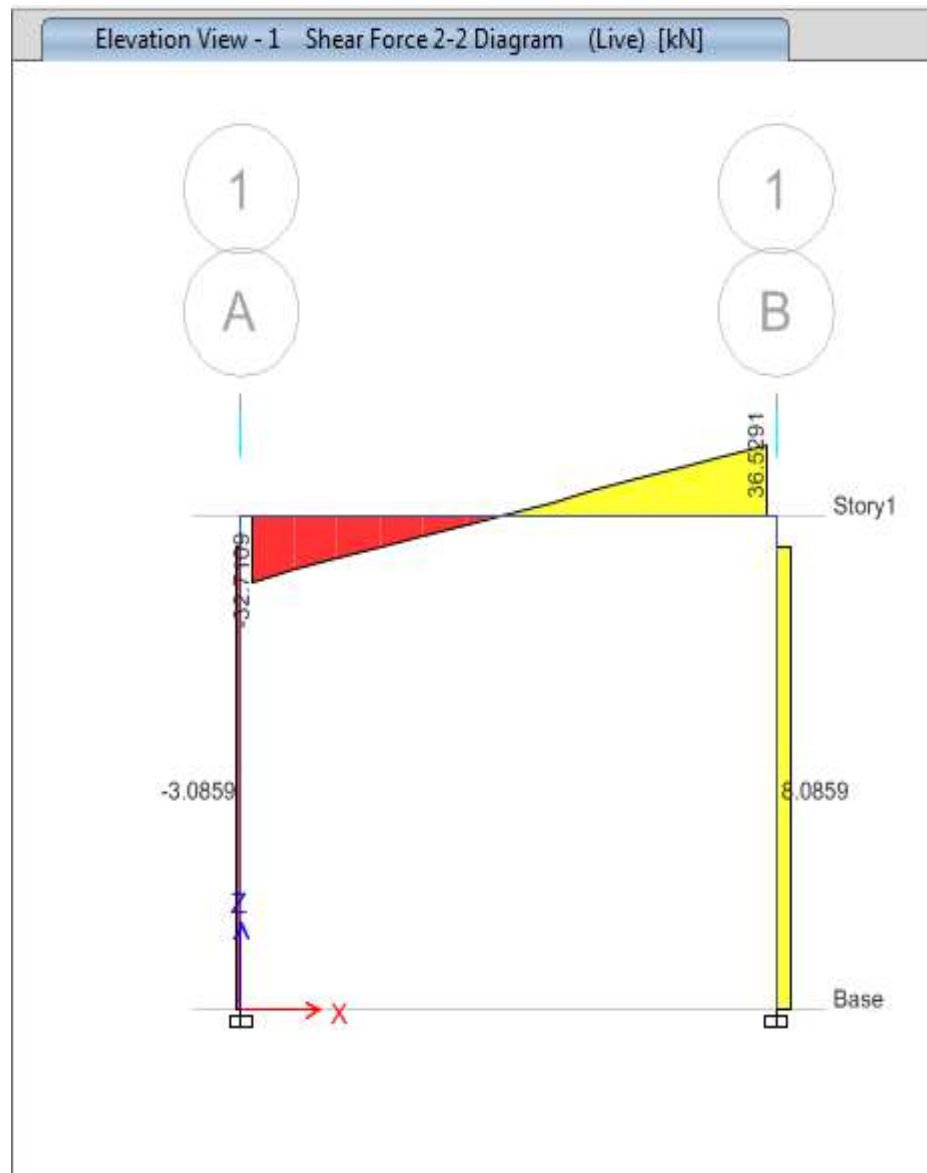
Include

☒ Frames ☐ Piers ☐ Spandrels

☐ Links

OK Close Apply

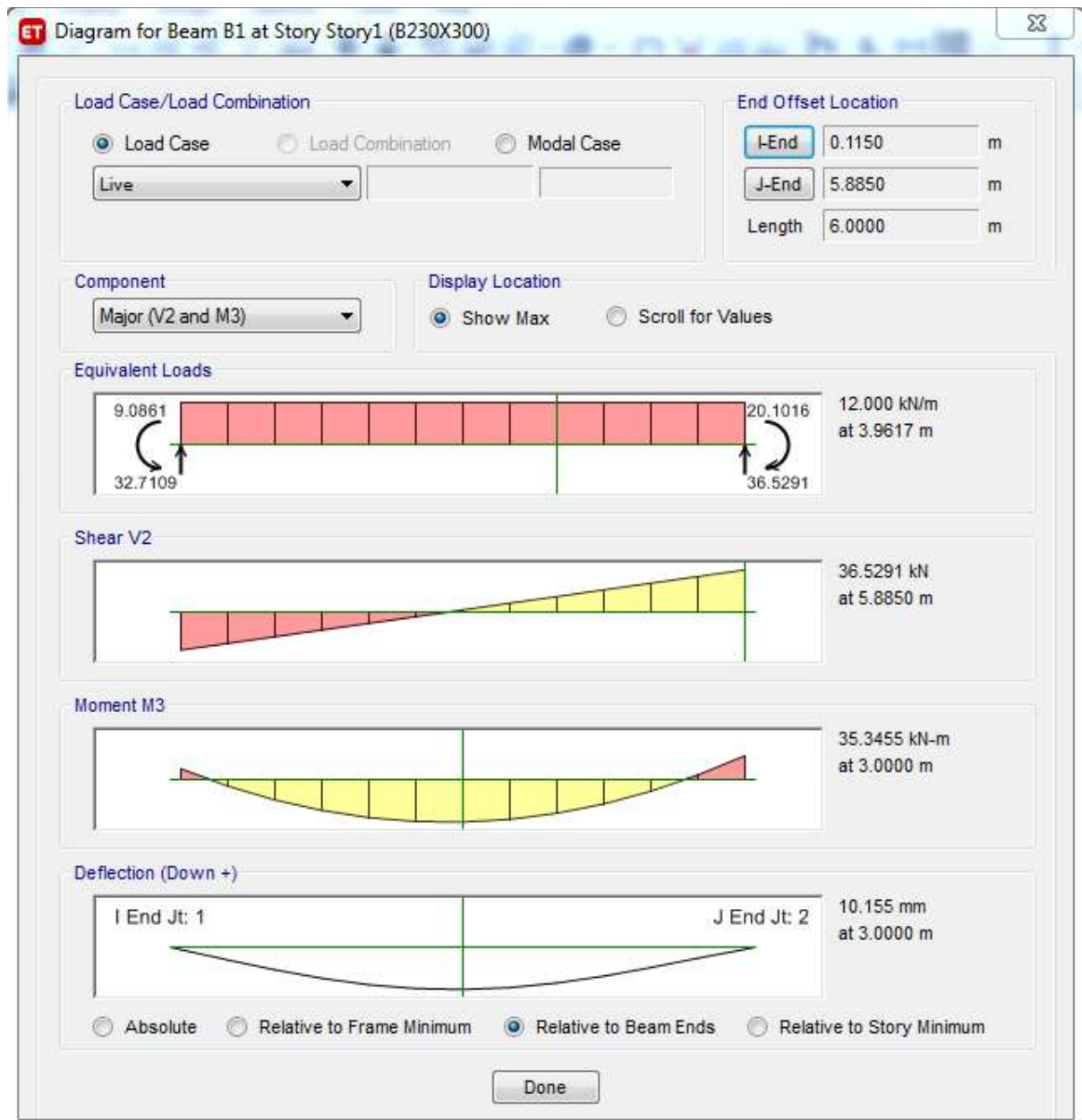
**Fig: Member Force Diagram for Frames/Piers/Spandrels /links form**



**Fig: SFD**

- To see the results like BMD, SFD & Deflection for individual beams, select the beam 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 Portal Frame Analysis in ETABS