



**Andhra Pradesh State Skill
Development Corporation**



Extended Three-Dimensional Analysis of Building System

ETABS

Simply Supported Beam Analysis

SIMPLY SUPPORTED BEAM ANALYSIS

Objective

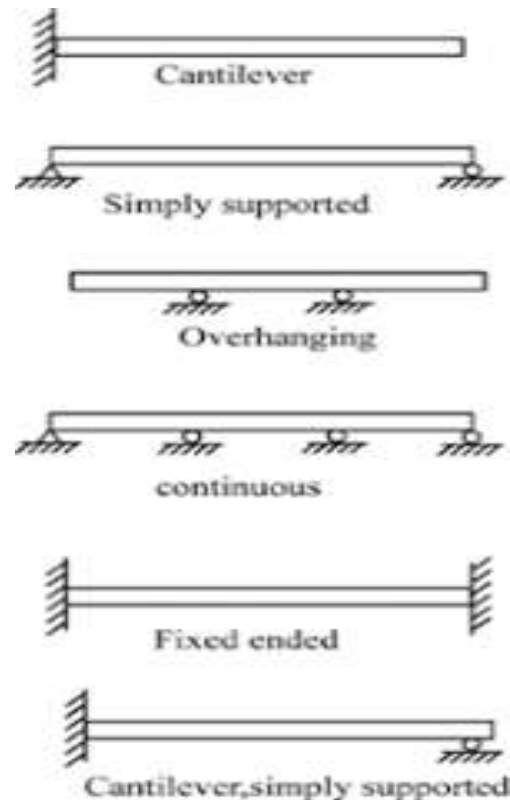
This chapter describes the step by step process of Simply Supported Beam Analysis in ETABS.

BEAM: Beam is a horizontal member which bends along the Z-axis (in ETABS) when the load is applied transversely. As it experiences both the tension and compression forces when it is loaded hence it is called as flexural member.

Types of Beams

1. Classification of beams based on support conditions

- Cantilever Beam : In this type of beams one end is fixed and the other end is free
- Simply Supported Beam: In this type of beams one end is hinged and the other end is constrained with roller support
- Overhanging beam: In this type of beams both the ends are overhanging (free) and contain intermediate supports.
- Continuous Beam: this type of beams contain intermediate supports along with end supports as shown in figure
- Fixed Beam: This type of beams contain fixed supports on both the ends
- Cantilever, Simply Supported Beam: This type of beams contains fixed support at one end and roller support at the other end.



2. Based on Spans

- Single Span
- Multiple span

Let's take an example of SSB

EXERCISE:

Considerations:

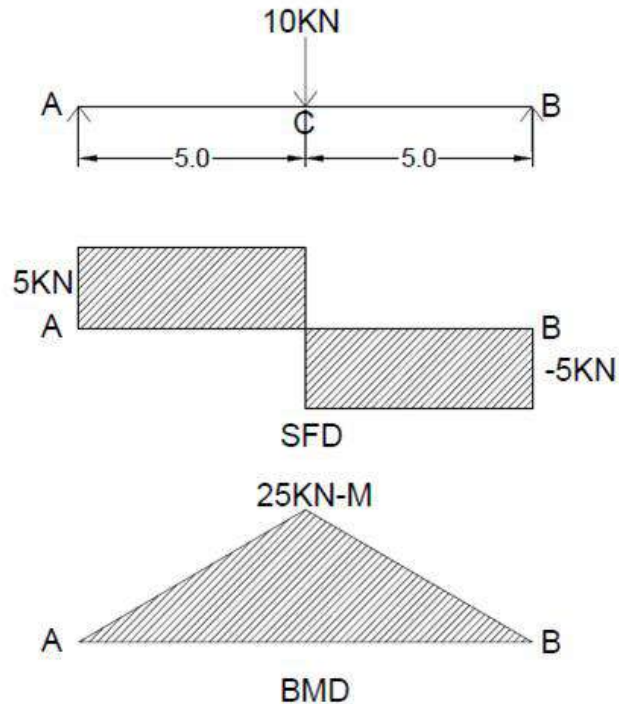
Material Properties

Concrete: M25

Steel: HYSD415

Section Property

Beam Section: 350x300 MM.Sq



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**

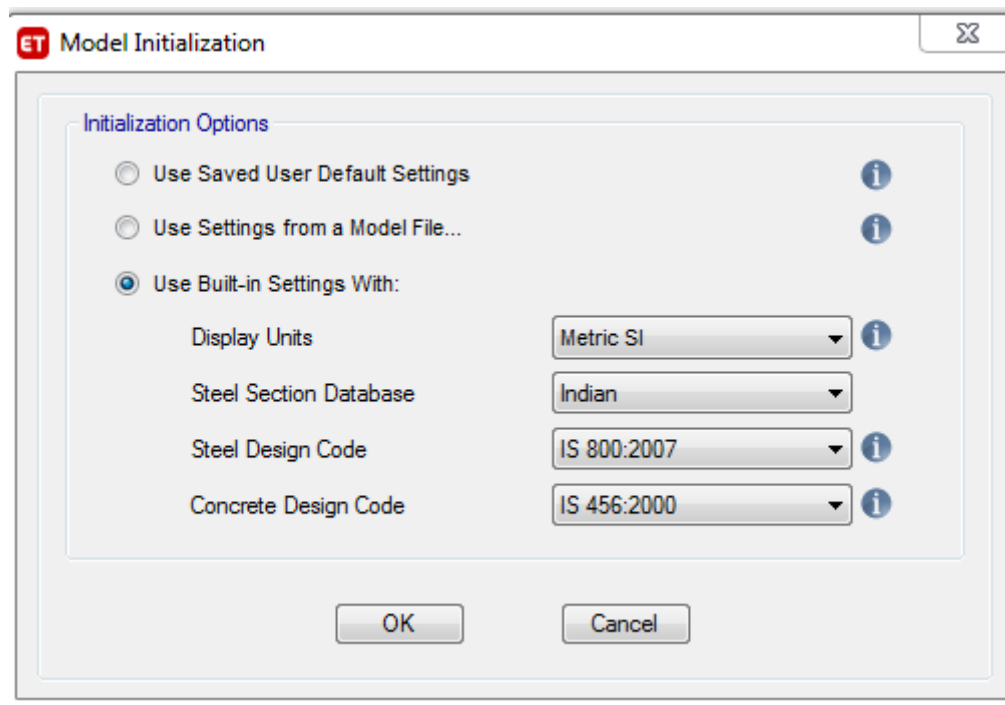


Fig: Model Initialization form

2. Specify the grid dimensions as 2-grids along X-axis, 1- grid along Y-axis and spacing along X-axis as 10m under **Uniform Grid Spacing** in the **New Model Quick Templates** form.

- Specify the no. of stories as 1 under **Story Data** in New **Model Quick Templates** form.
- Select **Grid only** template from **Add Structural Template**.

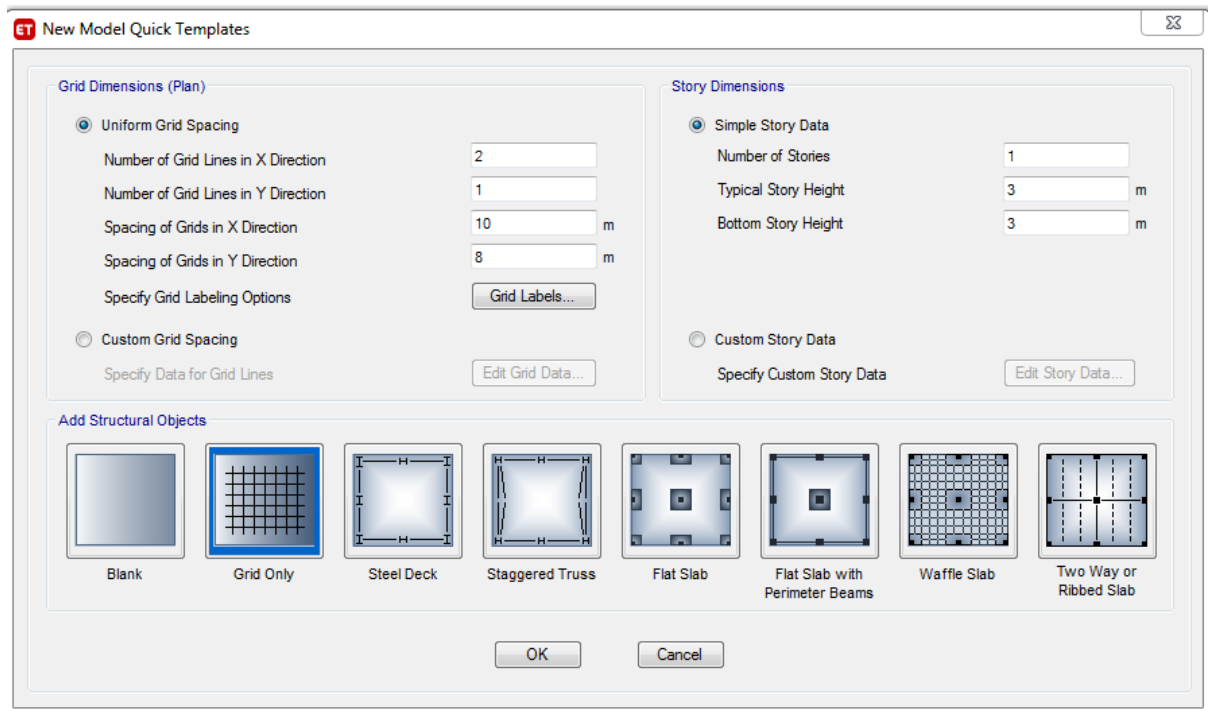


Fig: New Model Quick Templates form

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

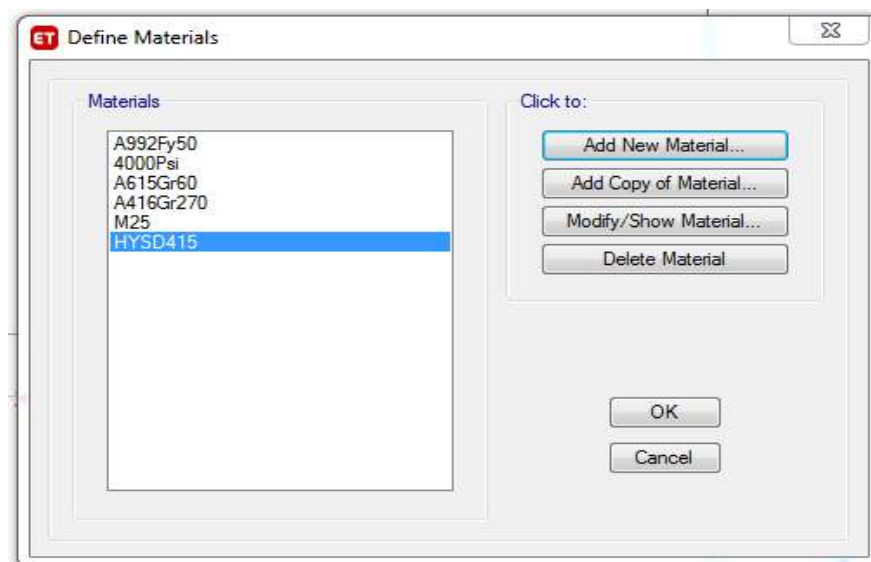
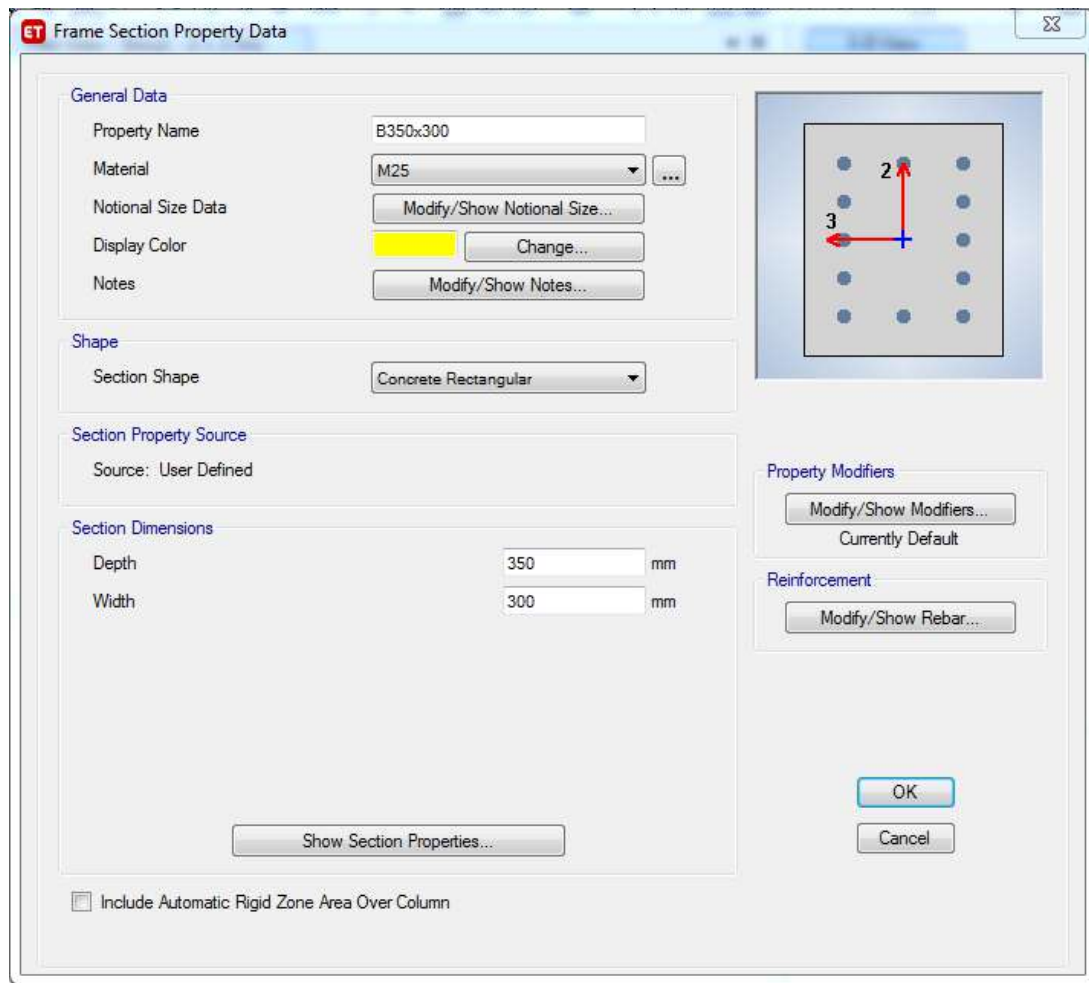


Fig: Define Materials form

- 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



ET Frame Section Property Data

General Data

Property Name: B350x300

Material: M25

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: 300 mm

Property Modifiers

Modify/Show Modifiers...
Currently Default

Reinforcement

Modify/Show Rebar...

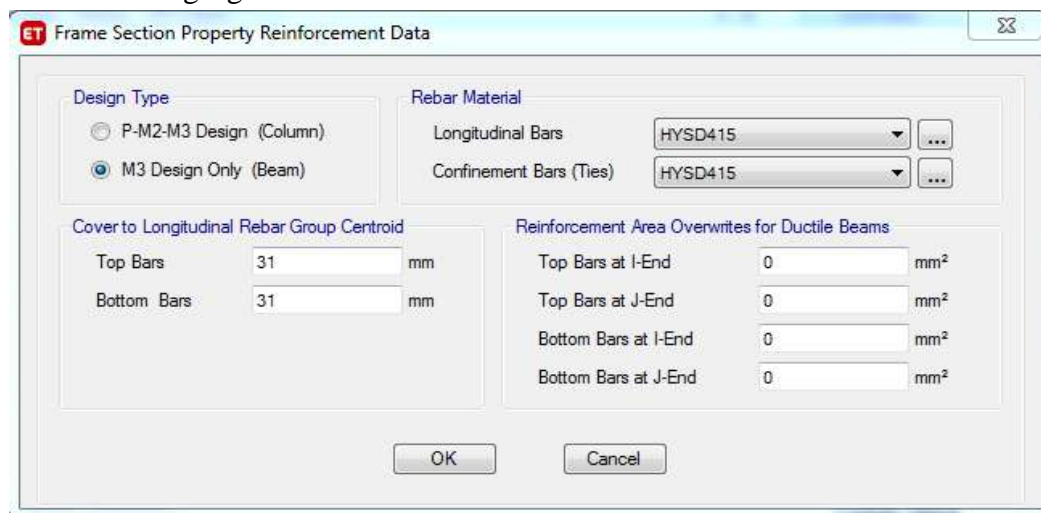
OK Cancel

☐ Include Automatic Rigid Zone Area Over Column

Show Section Properties...

Fig: Frame Property Data form

8. Click on **Modify/Show Rebar** to Specify the design type, rebar materials & cover 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: HYSD415

Confinement Bars (Ties): HYSD415

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 mm²

Top Bars at J-End: 0 mm²

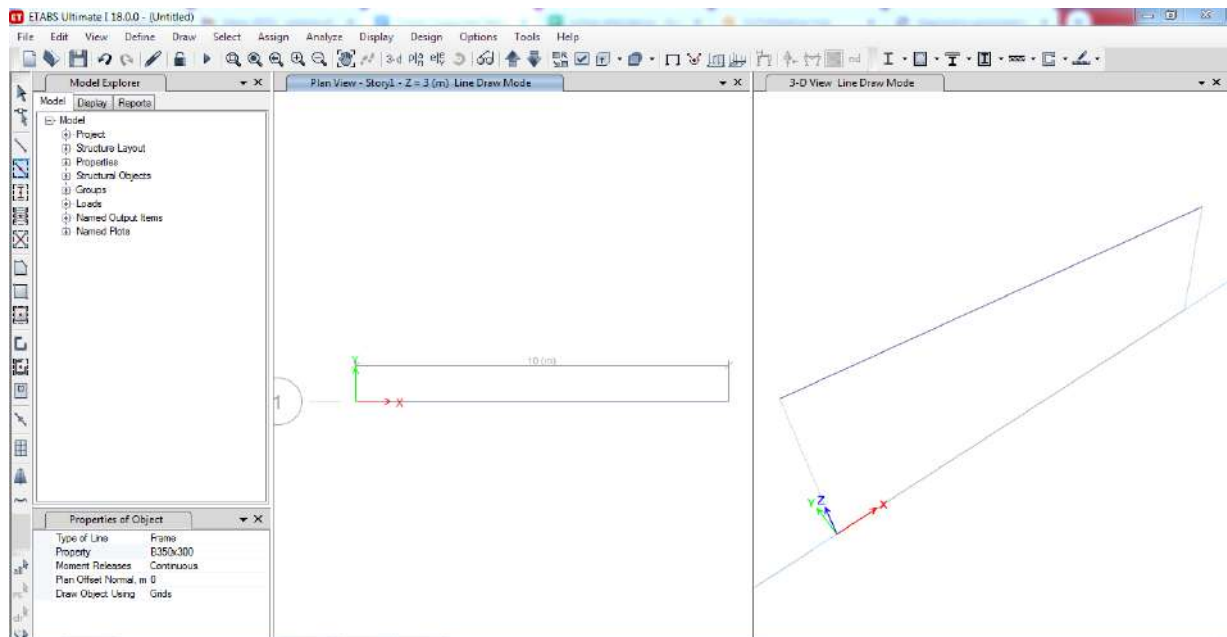
Bottom Bars at I-End: 0 mm²

Bottom Bars at J-End: 0 mm²

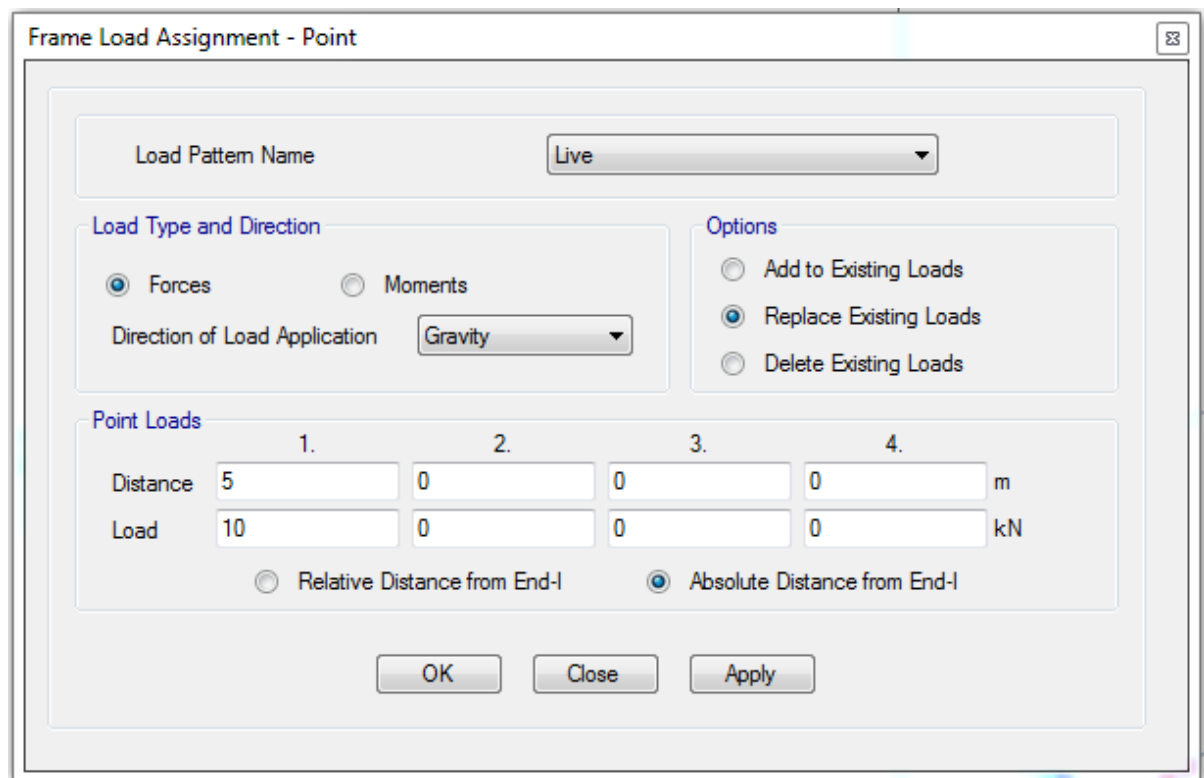
OK Cancel

Fig: Frame Section Property Reinforcement Data form

9. Assign the beam 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 frame and assign **Point load** of 10kN using **Frame Loads** in **Assign** menu



The screenshot shows the 'Frame Load Assignment - Point' dialog box. The 'Load Pattern Name' is set to 'Live'. Under 'Load Type and Direction', 'Forces' is selected, and 'Direction of Load Application' is set to 'Gravity'. Under 'Options', 'Replace Existing Loads' is selected. The 'Point Loads' section shows a table with 4 columns for distance and load. The first column has a distance of 5 m and a load of 10 kN. The other three columns have distances of 0 m and loads of 0 kN. The 'Absolute Distance from End-I' option is selected.

	1.	2.	3.	4.	
Distance	5	0	0	0	m
Load	10	0	0	0	kN

☐ Relative Distance from End-I ☒ Absolute Distance from End-I

OK Close Apply

Fig: Frame Load Assignment-Point form

11. To Assign Supports select the joints one by one, go to **Assign > Joints > Restraints**, and assign hinge support to one joint and roller support to another joint.

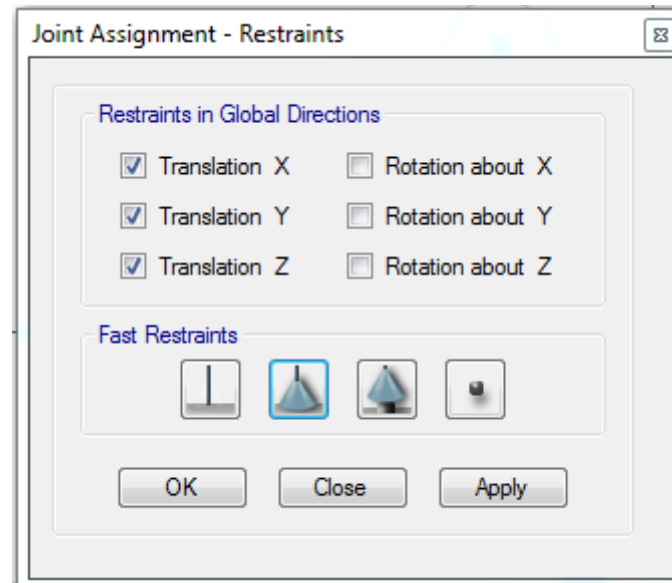


Fig: Joint Assignment-Restraints form



Fig: Elevational View

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**

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

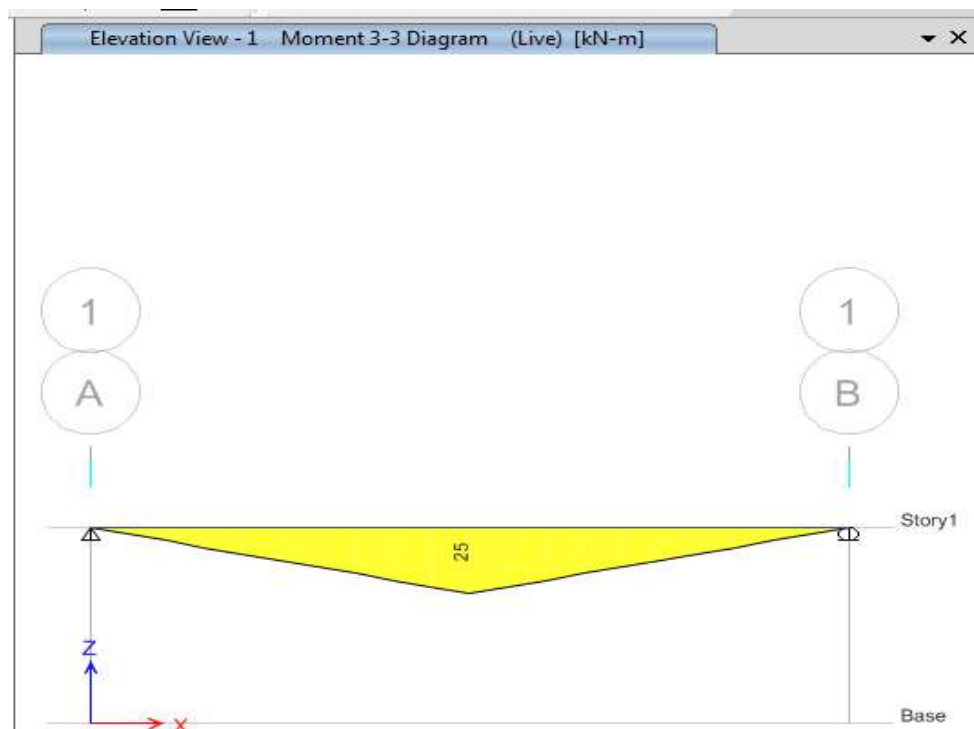


Fig: BMD

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

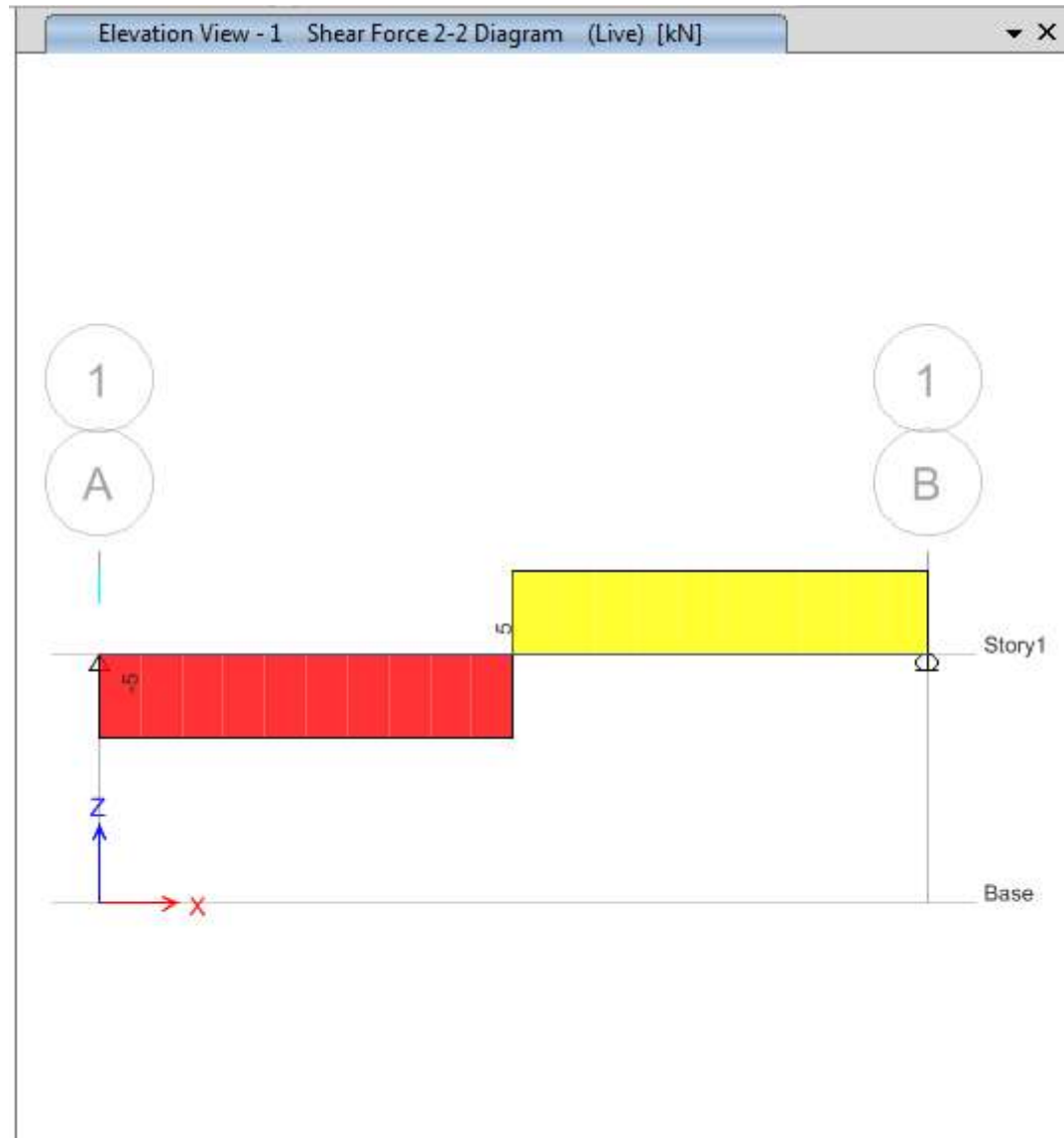


Fig: SFD

- 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.

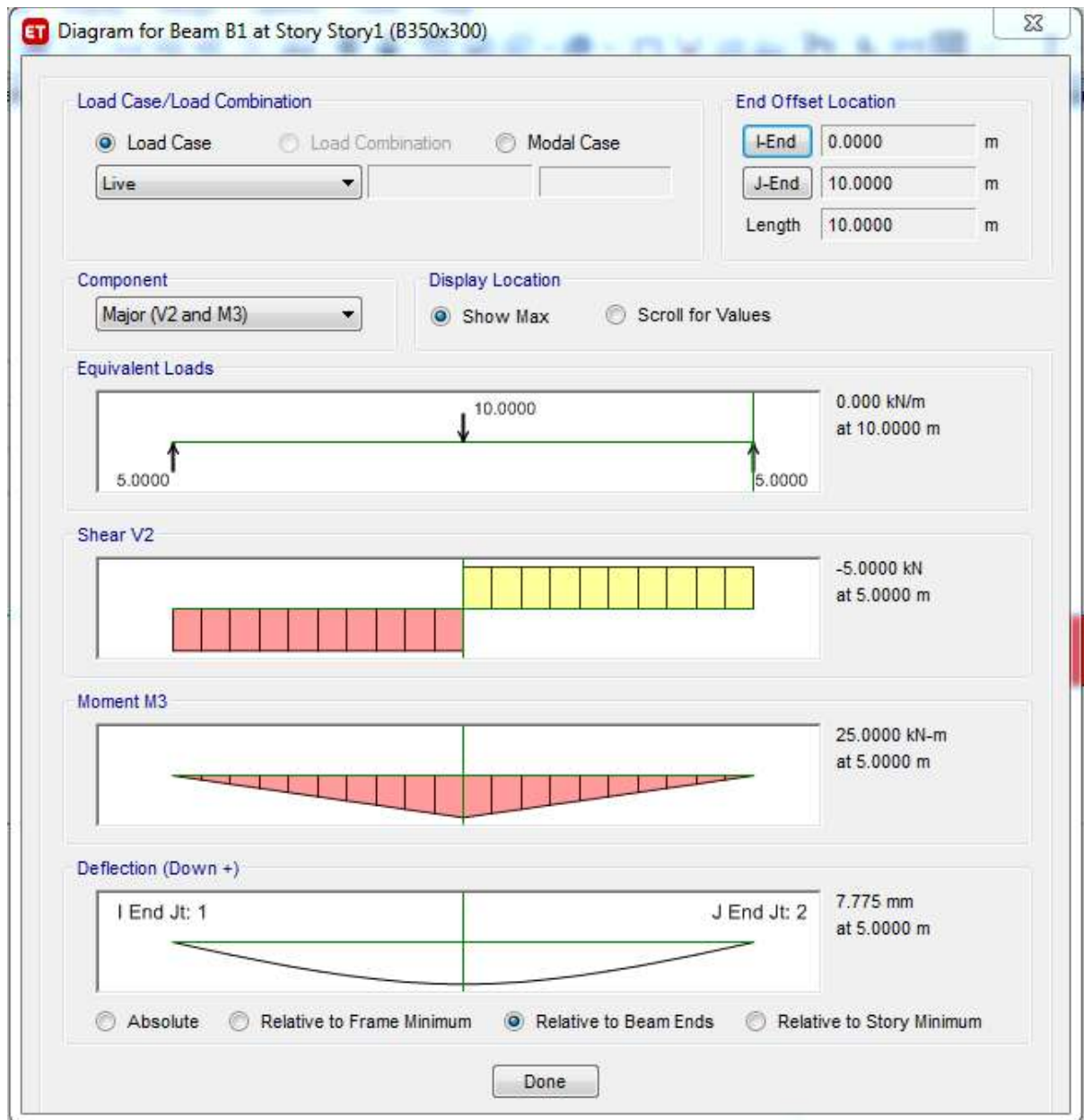


Fig: Diagram for Beam form

Here ends the process of SSB analysis in ETABS