Andhra Pradesh State Skill Development Corporation

e Skill AP
Learn Anytime Anywhere

Andhra Pradesh State Skill Development Corporation

courses
software
documentation
tracking
LMS
system
education
e-learning
management

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 f 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.
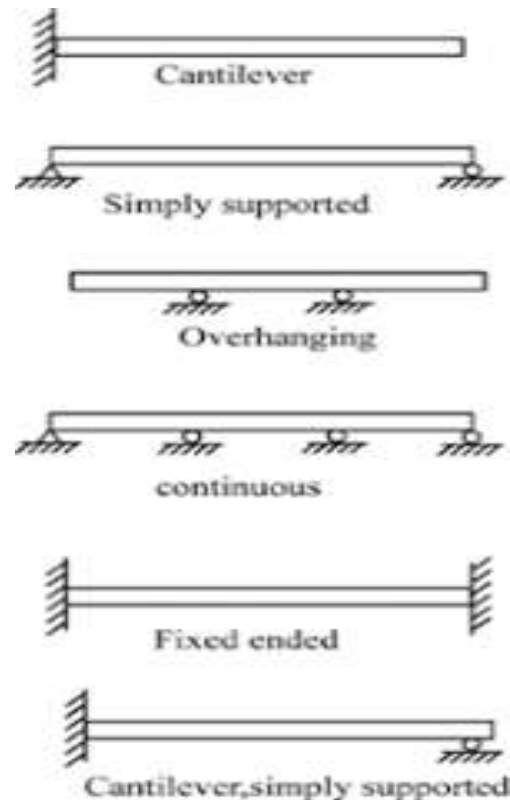
2. **Based on Spans**
    *   Single Span
    *   Multiple span

Let's take an example of SSB
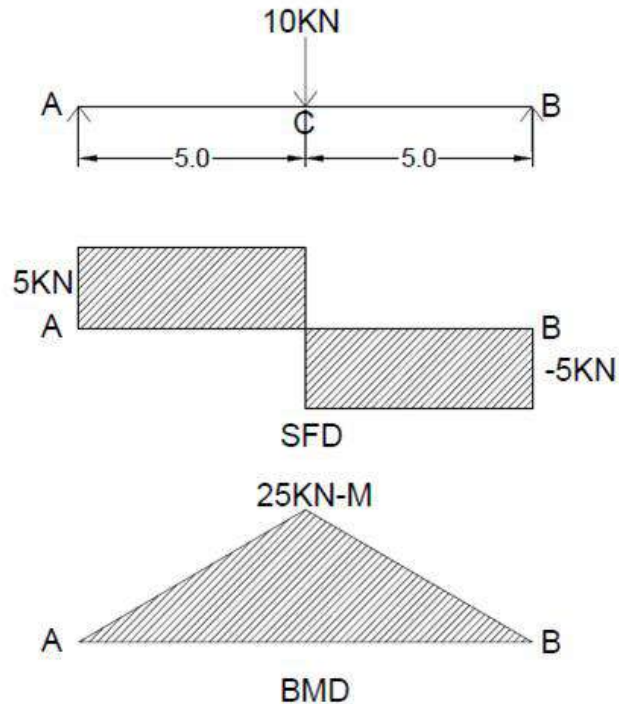
EXERCISE:

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**
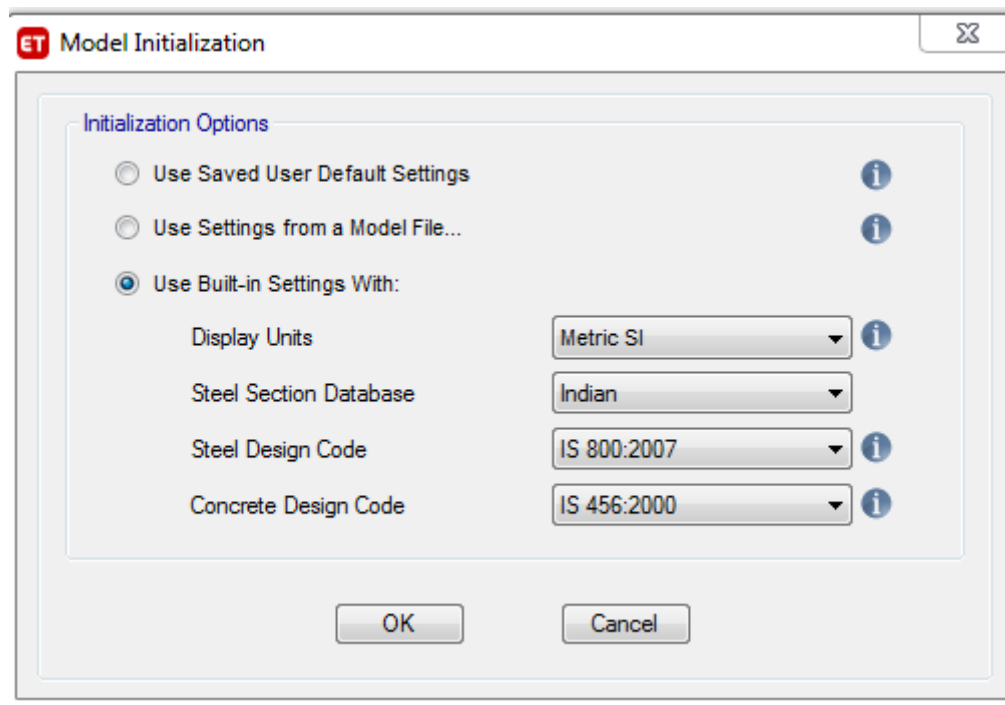

Fig: Model Initialization form

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.

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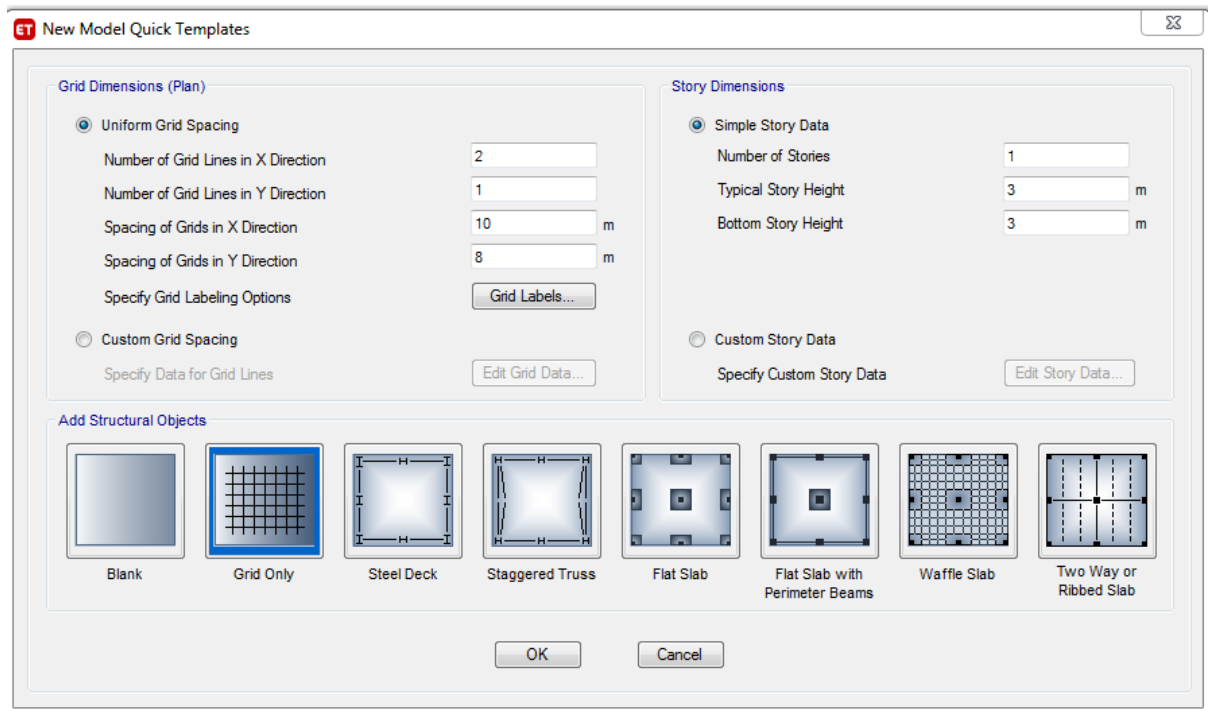
Fig: New Model Quick Templates form

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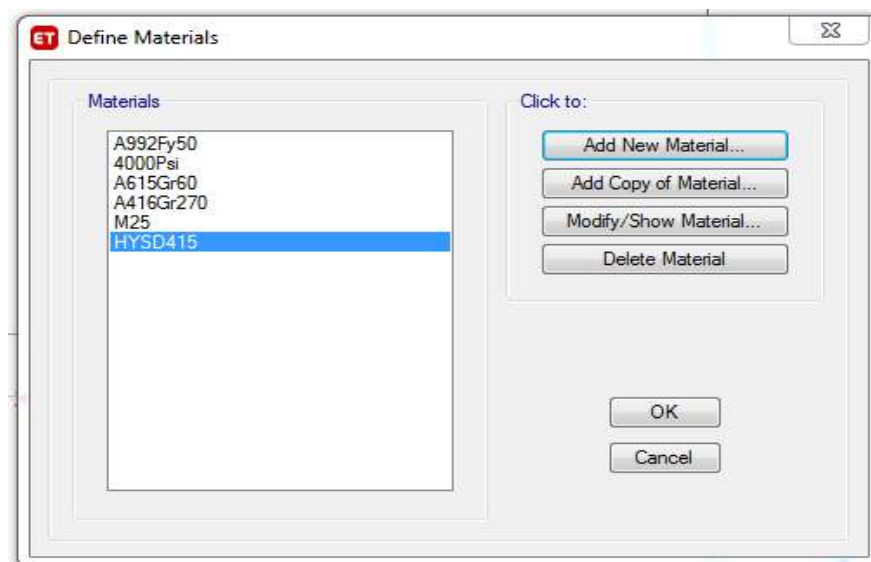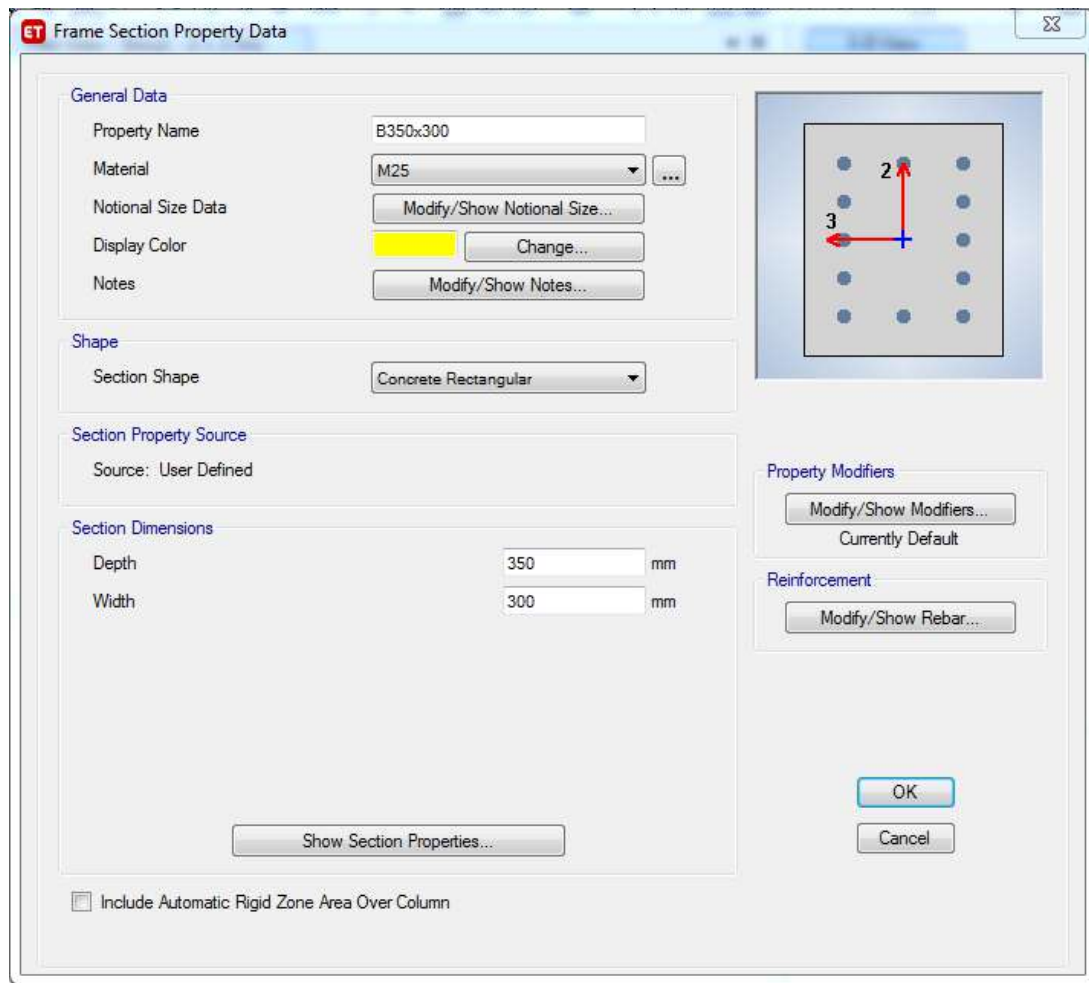

Fig: Define Materials form

7. 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 Property Data form

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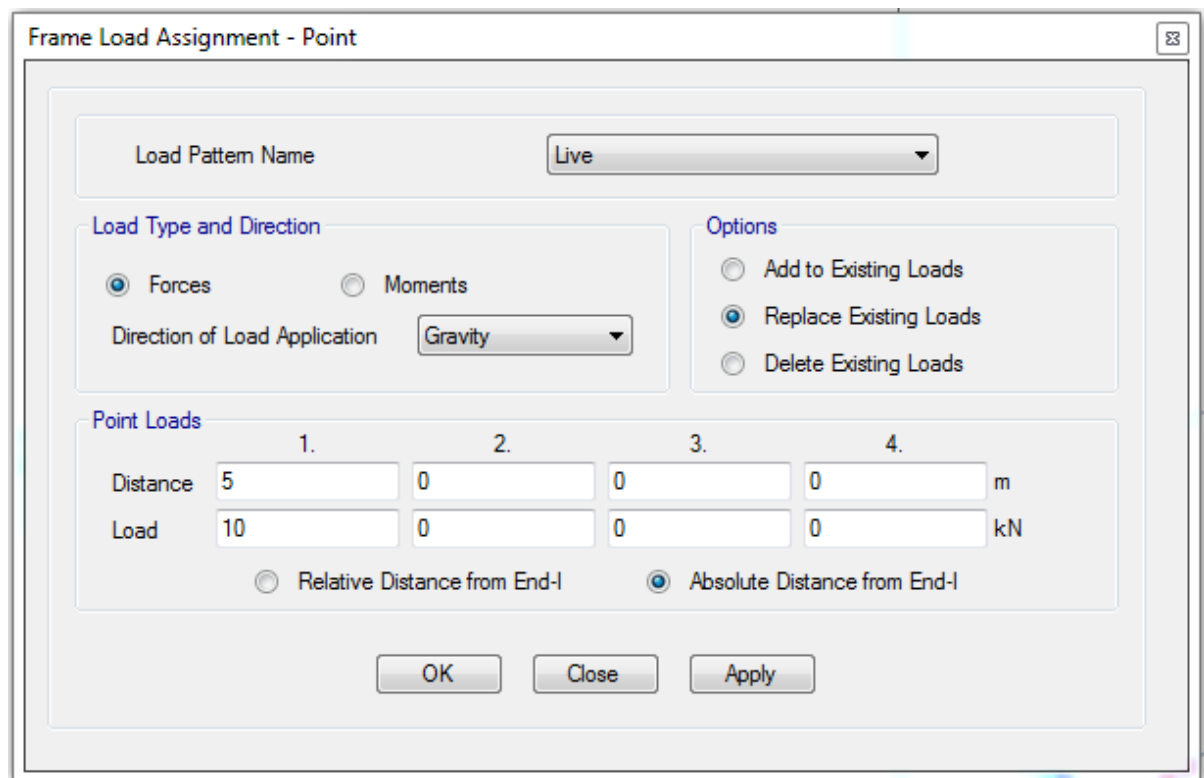

Fig: Frame Section Property Reinforcement Data form

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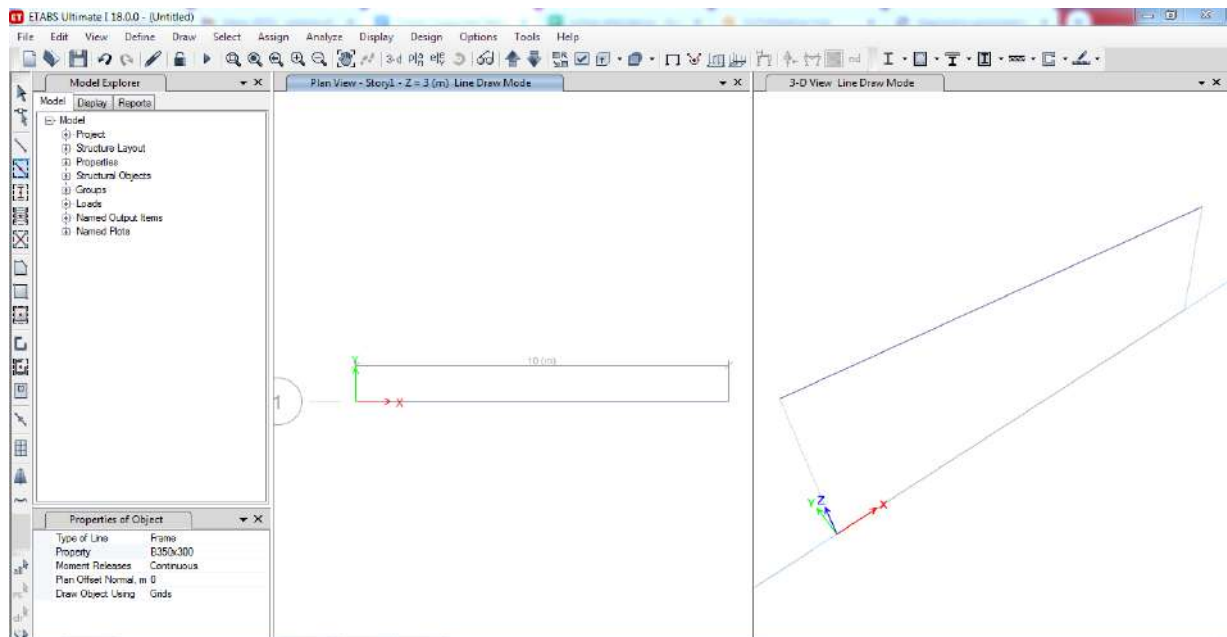
10. Select the frame and assign **Point load** of 10kN using **Frame Loads** in **Assign** menu

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.
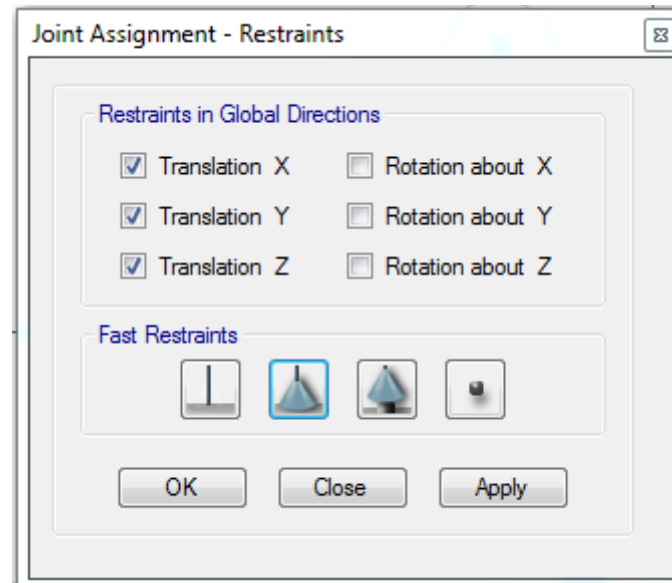
Fig: Joint Assignment-Restraints form

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**

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

Fig: BMD

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

Fig: SFD
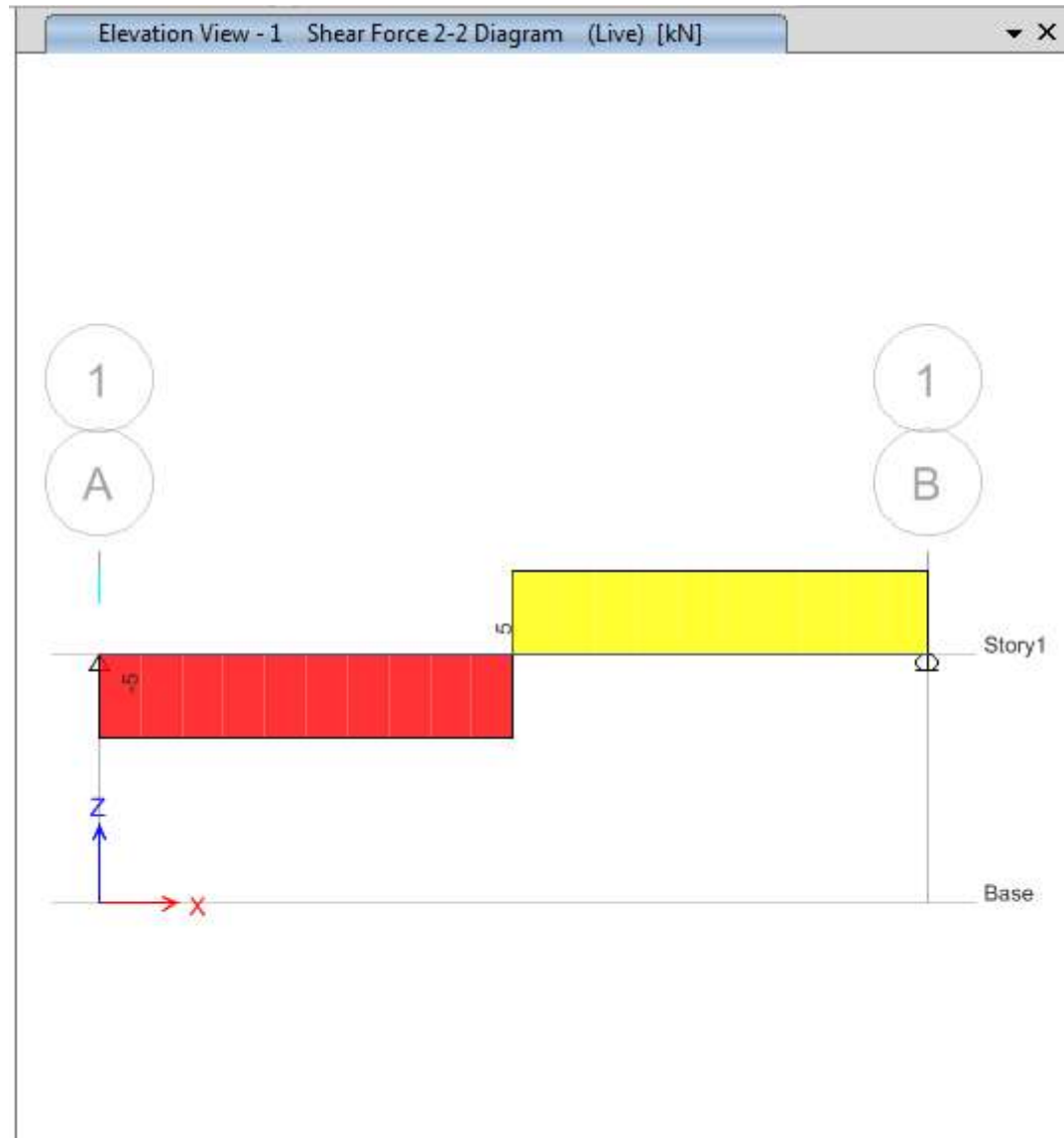
15. To see the results like BMD, SFD & Deflection for individual beams, select the beam and specify right click immediately it will display the form as shown below. Select the load case from the dropdown list for which you would like to view the results. And then click on the **Close** button to close the form.
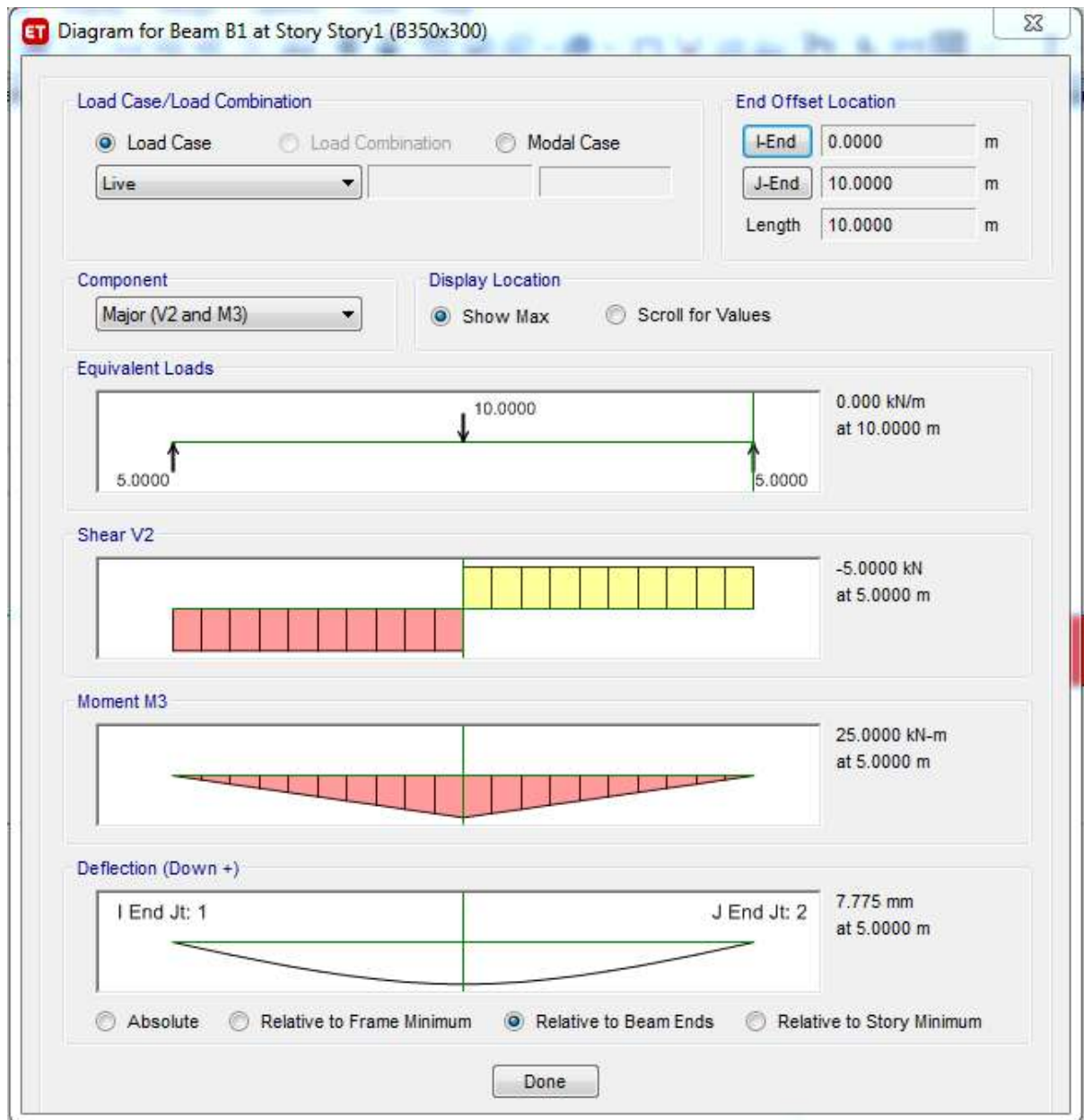
Fig: Diagram for Beam form

Here ends the process of SSB analysis in ETABS