



**Andhra Pradesh State Skill
Development Corporation**



Extended Three-Dimensional Analysis of Building System

ETABS

Continuous Beam Analysis

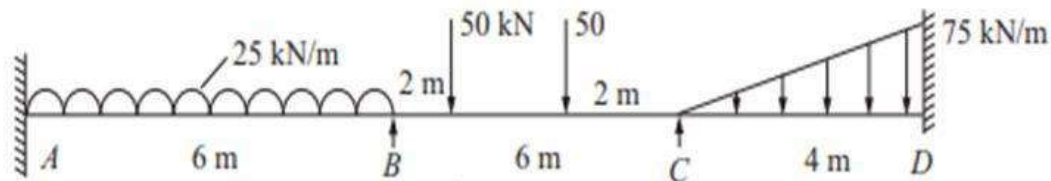
CONTINUOUS BEAM ANALYSIS

Objective

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

EXERCISE:

This is a textbook Example; the manual results are as shown below.



Ans:

$$(i) R_A = 75.39 \text{ kN}, \quad R_B = 127.59 \text{ kN}, \quad R_C = 97.85 \text{ kN}, \quad R_D = 99.17 \text{ kN}$$

$$M_A = -75.78 \text{ kNm}, \quad M_B = -73.44 \text{ kNm}, \quad M_C = -55.55 \text{ kNm}, \quad M_D = -55.2 \text{ kNm}.$$

Considerations:

Material Properties

Concrete: M30

Steel: HYSD415

Section Property

Beam Section: 300x300 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**

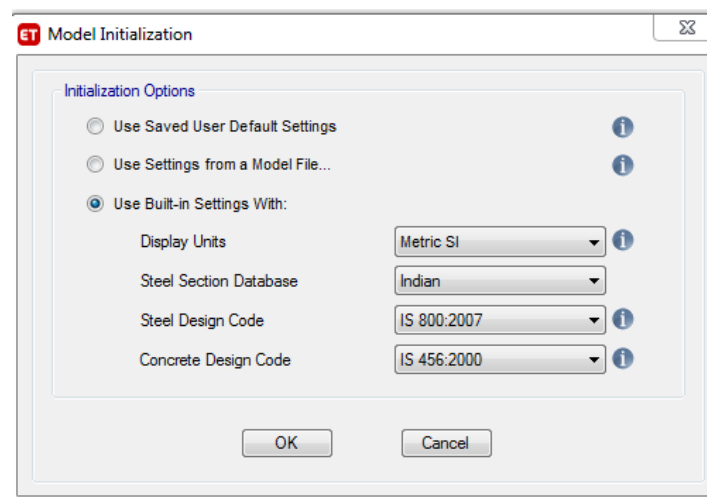
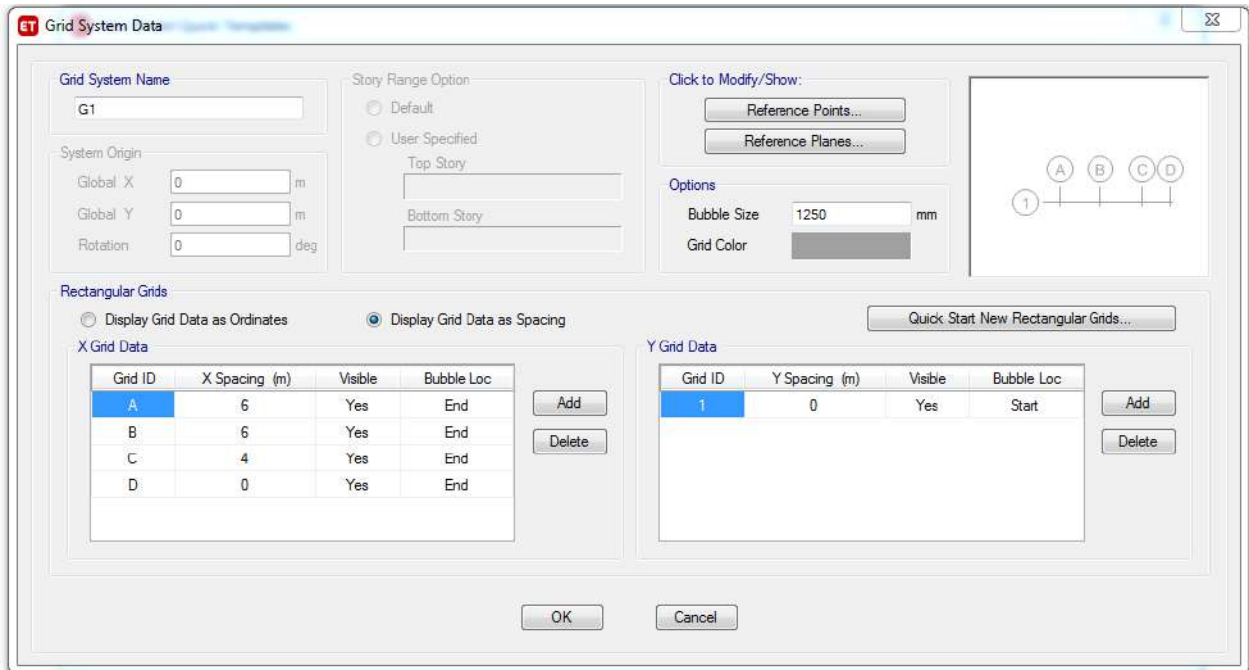


Fig: Model Initialization form

- Specify the grid dimensions as 4-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.



Grid System Data

Grid System Name: G1

System Origin:
Global X: 0 m
Global Y: 0 m
Rotation: 0 deg

Story Range Option:
☐ Default
☐ User Specified
Top Story:
Bottom Story:

Click to Modify/Show:
Reference Points...
Reference Planes...

Options:
Bubble Size: 1250 mm
Grid Color:

Rectangular Grids:
☐ Display Grid Data as Ordinates
☒ Display Grid Data as Spacing

X Grid Data

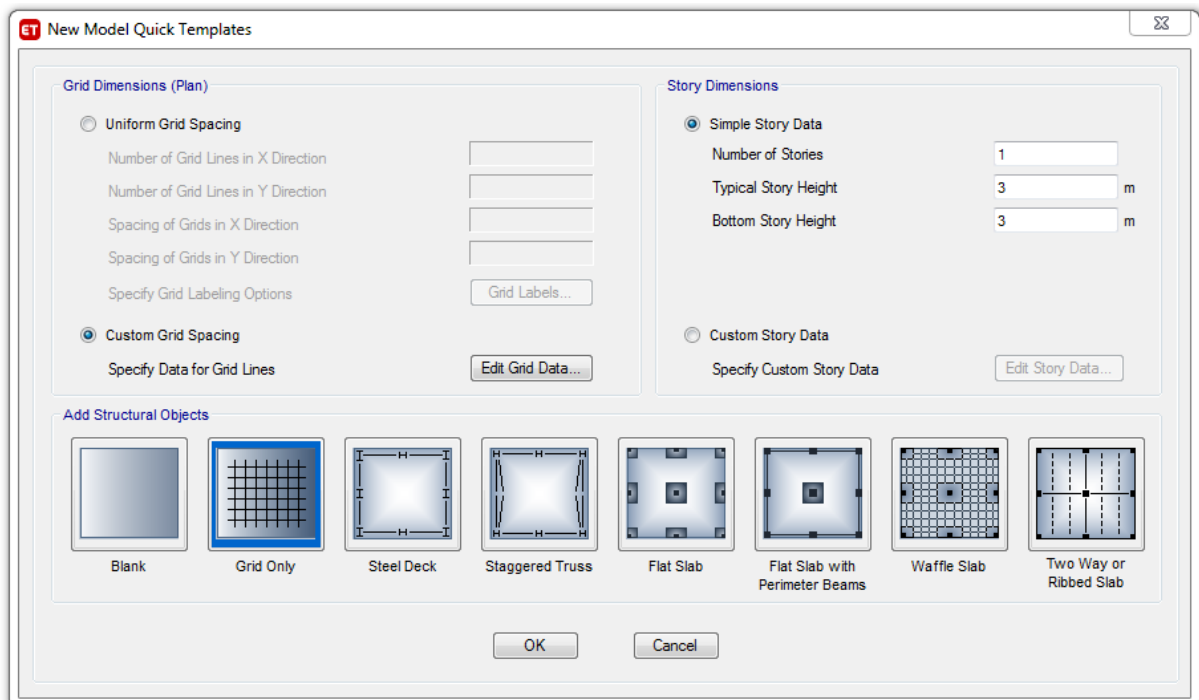
Grid ID	X Spacing (m)	Visible	Bubble Loc
A	6	Yes	End
B	6	Yes	End
C	4	Yes	End
D	0	Yes	End

Y Grid Data

Grid ID	Y Spacing (m)	Visible	Bubble Loc
1	0	Yes	Start

OK Cancel

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



New Model Quick Templates

Grid Dimensions (Plan)
☐ Uniform Grid Spacing
 Number of Grid Lines in X Direction:
 Number of Grid Lines in Y Direction:
 Spacing of Grids in X Direction:
 Spacing of Grids in Y Direction:
 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: 3 m
☐ Custom Story Data
 Specify Custom Story Data: Edit Story Data...

Add Structural Objects

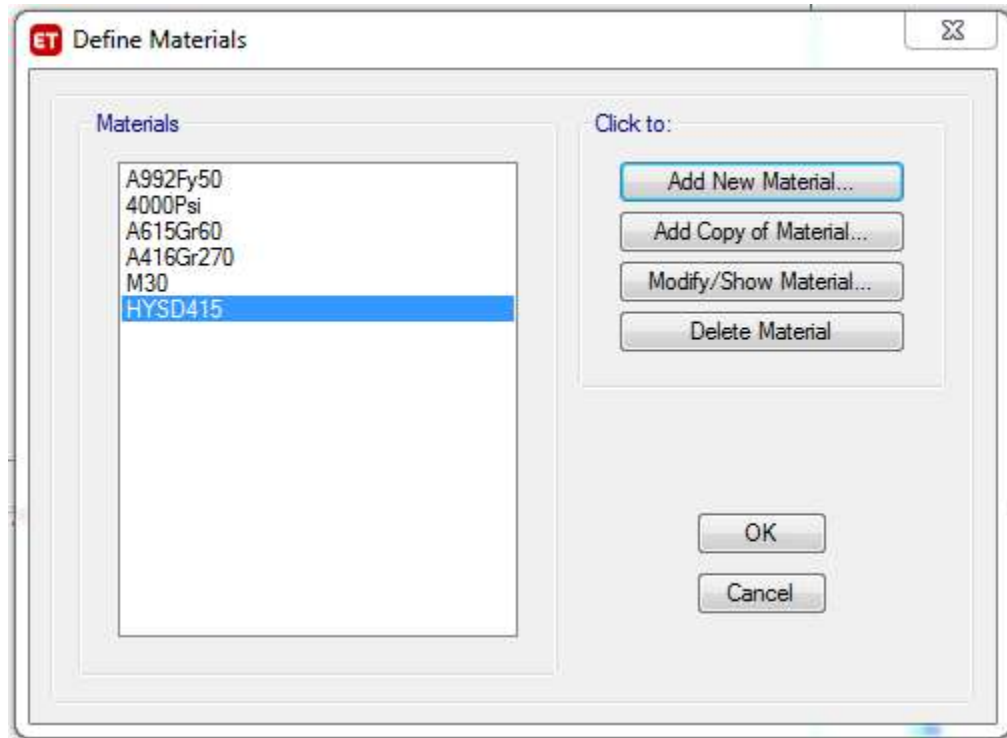
Blank Grid Only Steel Deck Staggered Truss Flat Slab Flat Slab with Perimeter Beams Waffle Slab Two Way or Ribbed Slab

OK Cancel

Fig: New Model Quick Templates form

- By keeping remaining settings as default click **Ok**.

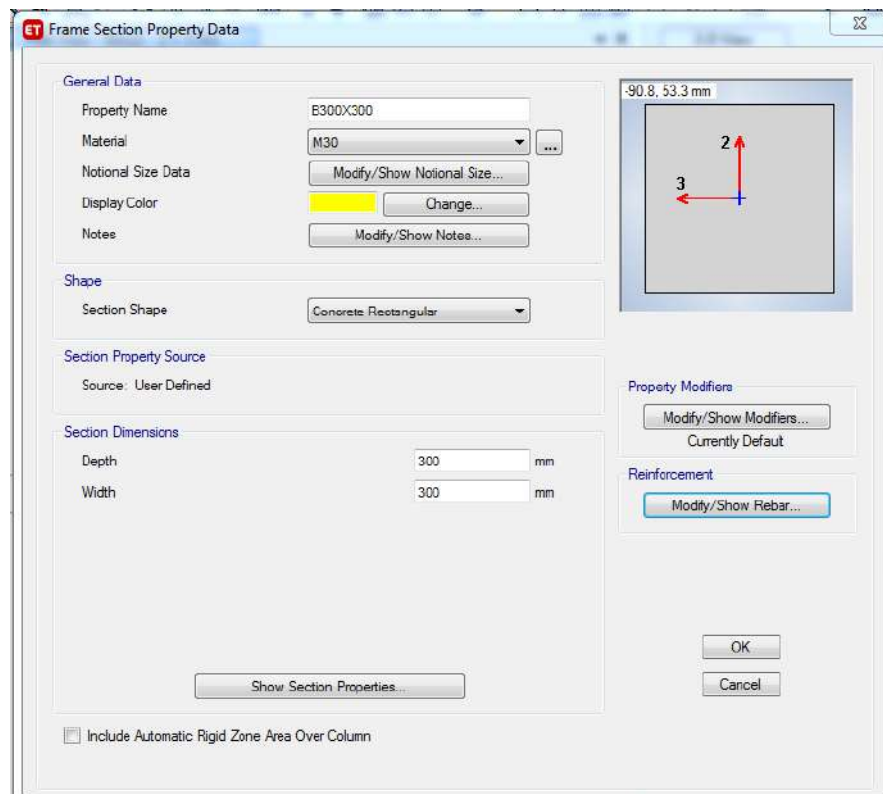
- Go to **Define menu > Material Properties**, Click on **Add New Material** option and add M30 grade concrete and HYSD415 grade rebar using **Add New Material Property** form.



The 'Define Materials' dialog box shows a list of materials on the left and action buttons on the right. The materials listed are A992Fy50, 4000Psi, A615Gr60, A416Gr270, M30, and HYSD415. The 'HYSD415' material is currently selected. The action buttons on the right are 'Add New Material...', 'Add Copy of Material...', 'Modify/Show Material...', and 'Delete Material'. At the bottom are 'OK' and 'Cancel' buttons.

Fig: Define Materials form

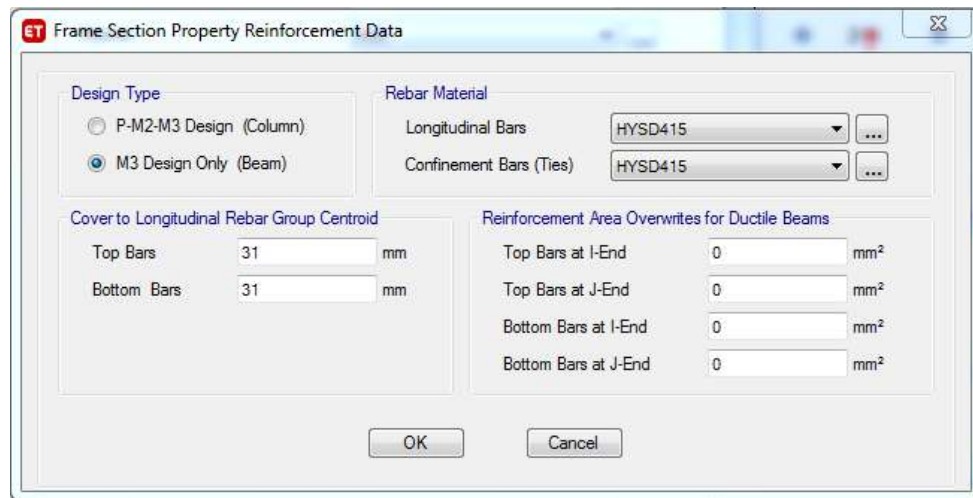
- Go to **Define menu > Section Properties > Frame Sections**, click on **Add New Property** and specify the parameters in **Frame Section Property Data** form as shown in following figures



The 'Frame Section Property Data' dialog box is divided into several sections. The 'General Data' section includes fields for Property Name (B300X300), Material (M30), Notional Size Data (Modify/Show Notional Size...), Display Color (yellow), and Notes (Modify/Show Notes...). The 'Shape' section has a dropdown for Section Shape set to 'Concrete Rectangular'. The 'Section Property Source' section shows 'Source: User Defined'. The 'Section Dimensions' section has input fields for Depth (300 mm) and Width (300 mm). On the right, there is a diagram of a rectangular section with dimensions 300 mm by 300 mm and a coordinate system. Below the diagram are buttons for 'Property Modifiers' (Modify/Show Modifiers... Currently Default) and 'Reinforcement' (Modify/Show Rebar...). At the bottom are 'OK' and 'Cancel' buttons, and a checkbox for 'Include Automatic Rigid Zone Area Over Column'.

Fig: Frame Property Data form

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



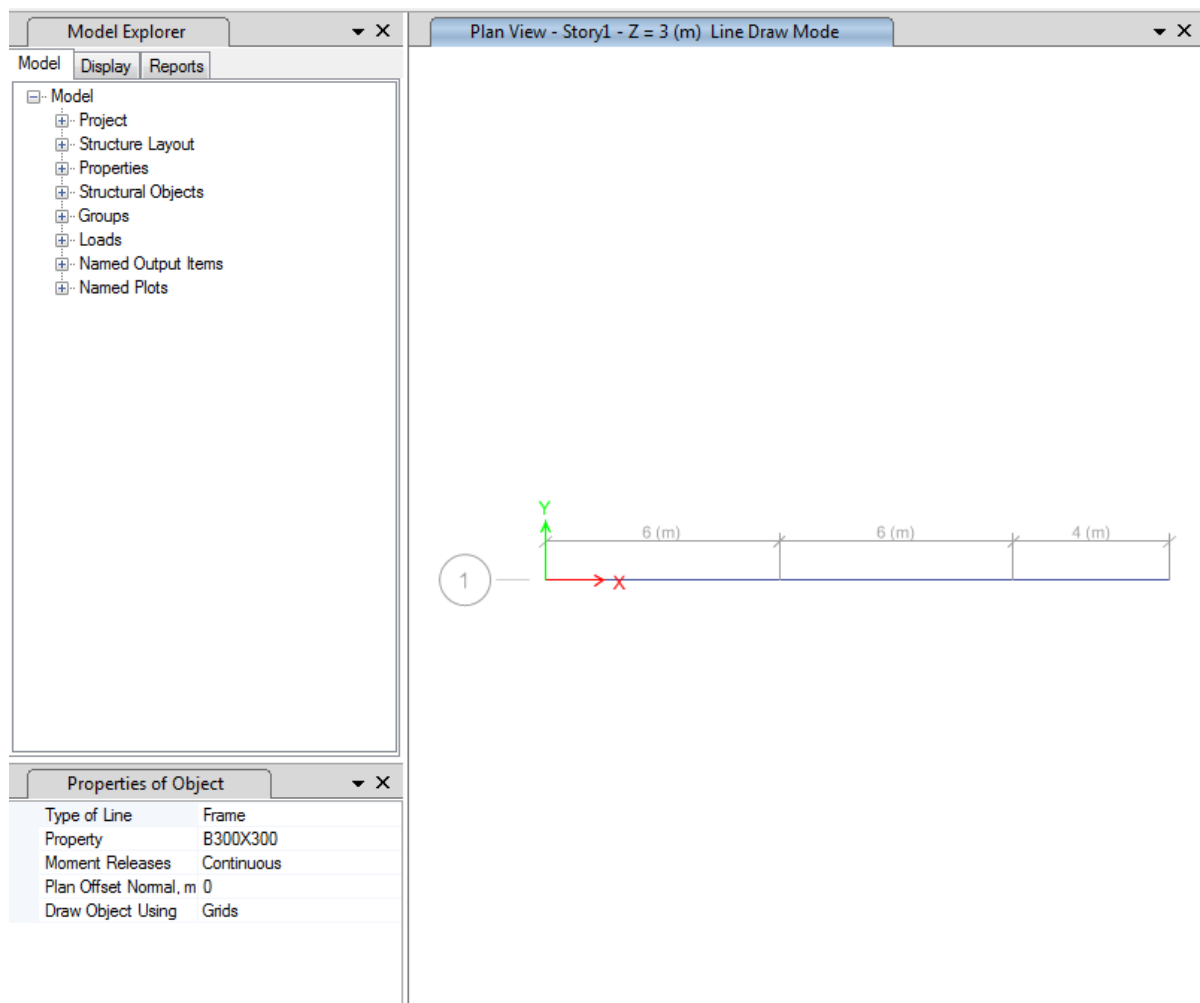
The dialog box titled "Frame Section Property Reinforcement Data" contains the following fields:

- 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

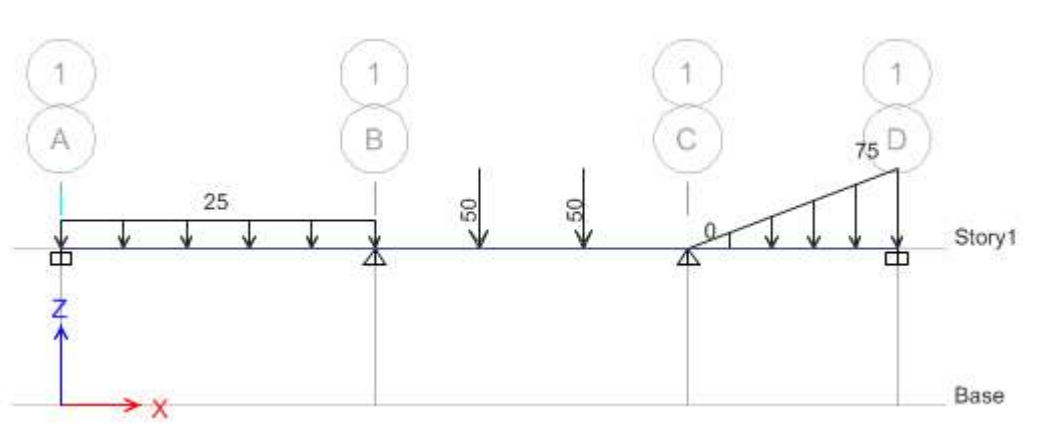
Buttons: OK, Cancel

Fig: Frame Section Property Reinforcement Data form

- 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 of 1st span and assign **UDL** of 25kN/m under Live load pattern, select the 2nd span and assign **Point loads** of each 50kN under Live load pattern and select 3rd span assign **UVL** OF 75kN/m under Live load pattern using **Frame Loads** in **Assign** menu



11. To Assign Supports select the joints one by one, go to **Assign > Joins > Restraints**, and assign Fixed support at end joints and hinged support at intermediate joints.

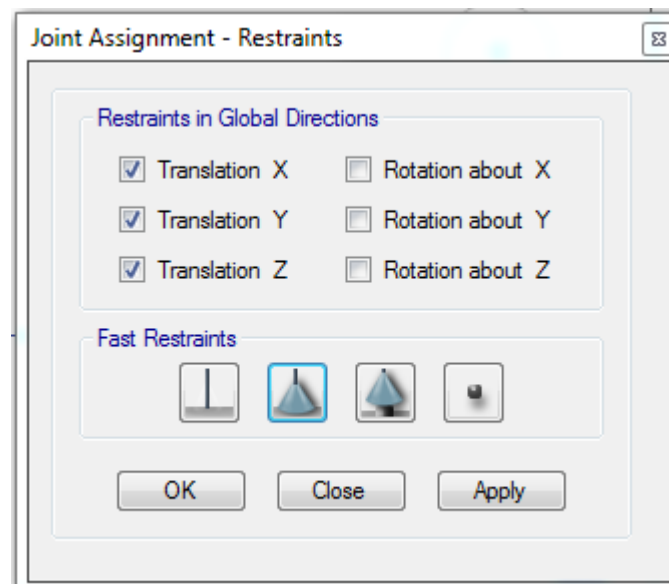
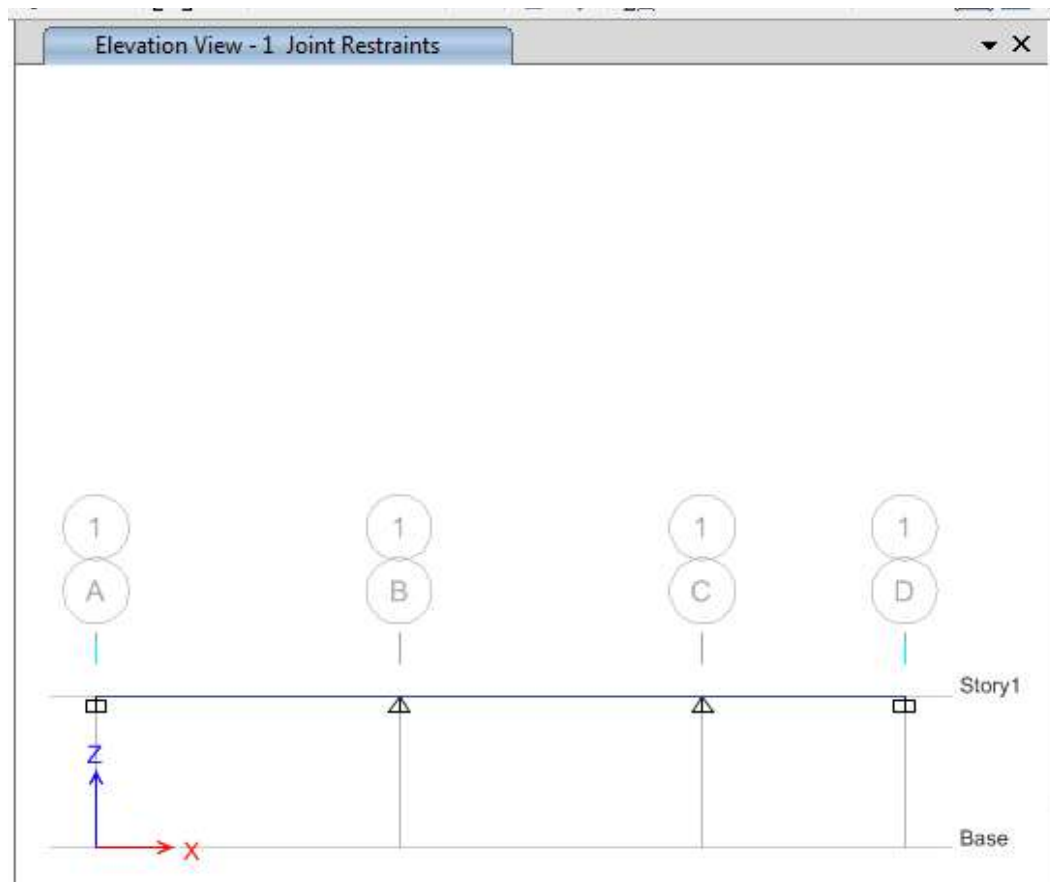


Fig: Joint Assignment-Restraints form



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 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**

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

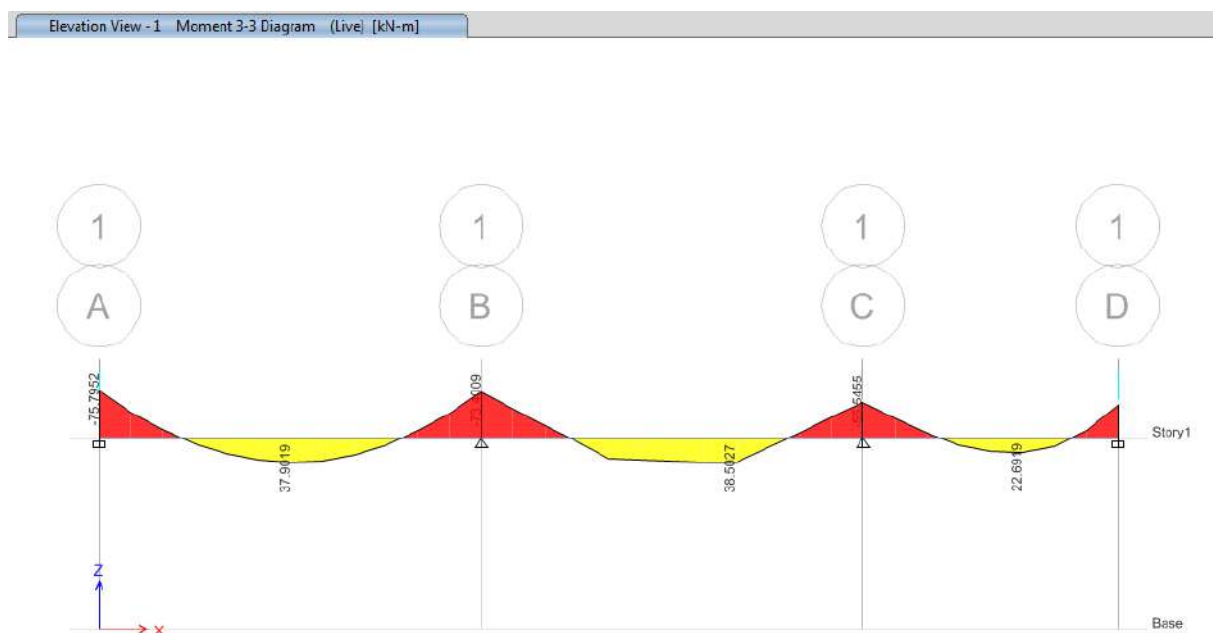


Fig: BMD

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

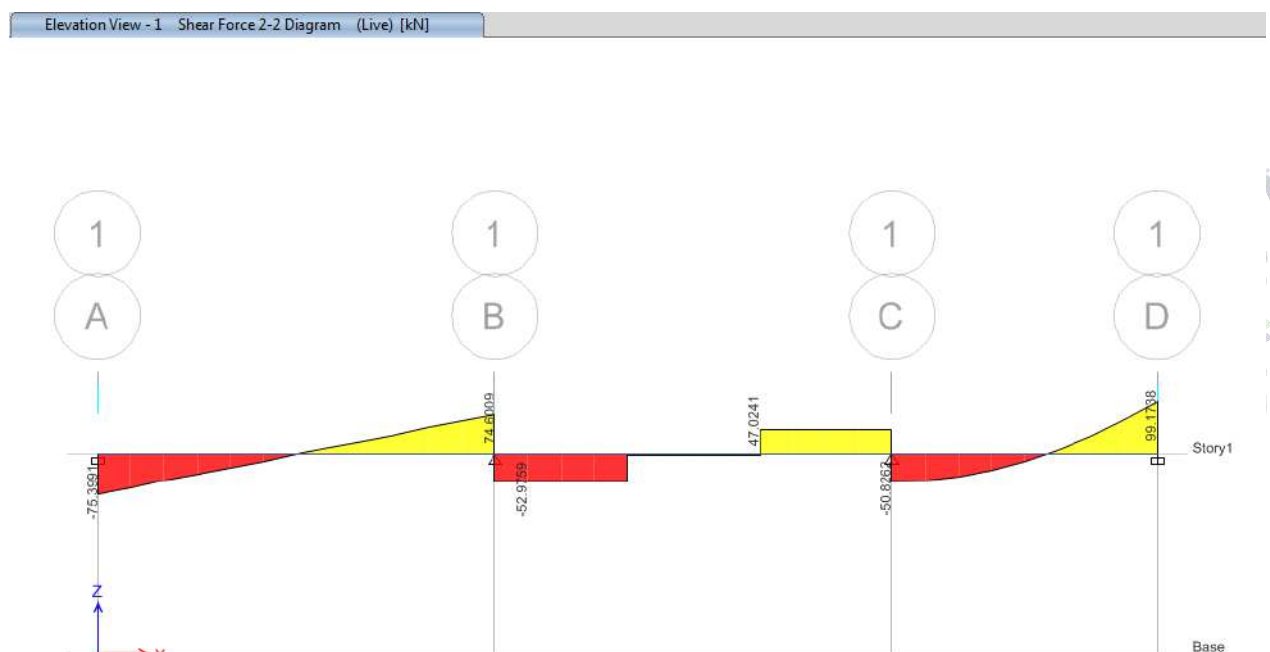


Fig: SFD

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.
16. To see the reactions at the supports click on **Display Support/ Spring Reactions** or **F7**

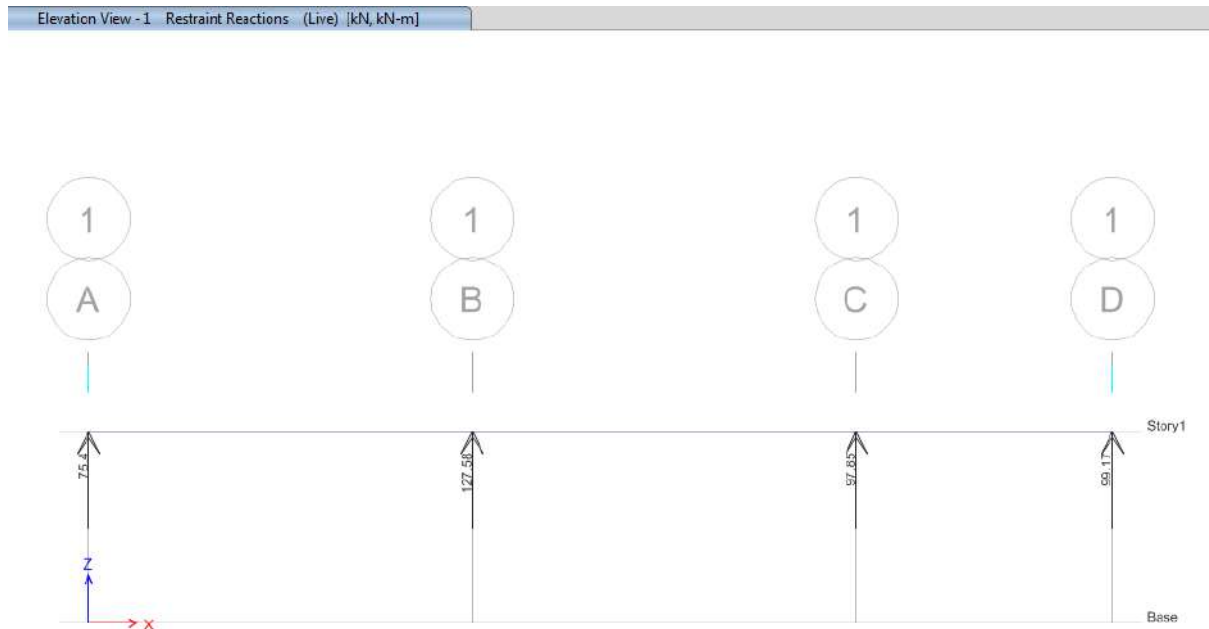


Fig: Showing Restraints Reactions

Here ends the process of Continuous Beam Analysis in ETABS.