



**Andhra Pradesh State Skill
Development Corporation**



Extended Three-Dimensional Analysis of Building System

ETABS

AutoCAD Plan Import and Complete Analysis & Design (Method-II)

AutoCAD Plan Import and Complete Analysis & Design (Method -2)

Objective

This chapter contains an explanation on AutoCAD plan import and complete analysis & design in ETABS.

Example Plan

CONSIDERATIONS

Material Properties

Concrete: M30

Steel: HYSD500

Section Property

Beam Section: 230mm X 350mm

Column Sections: 230mm X 300mm

Slab: 150mm

Procedure:

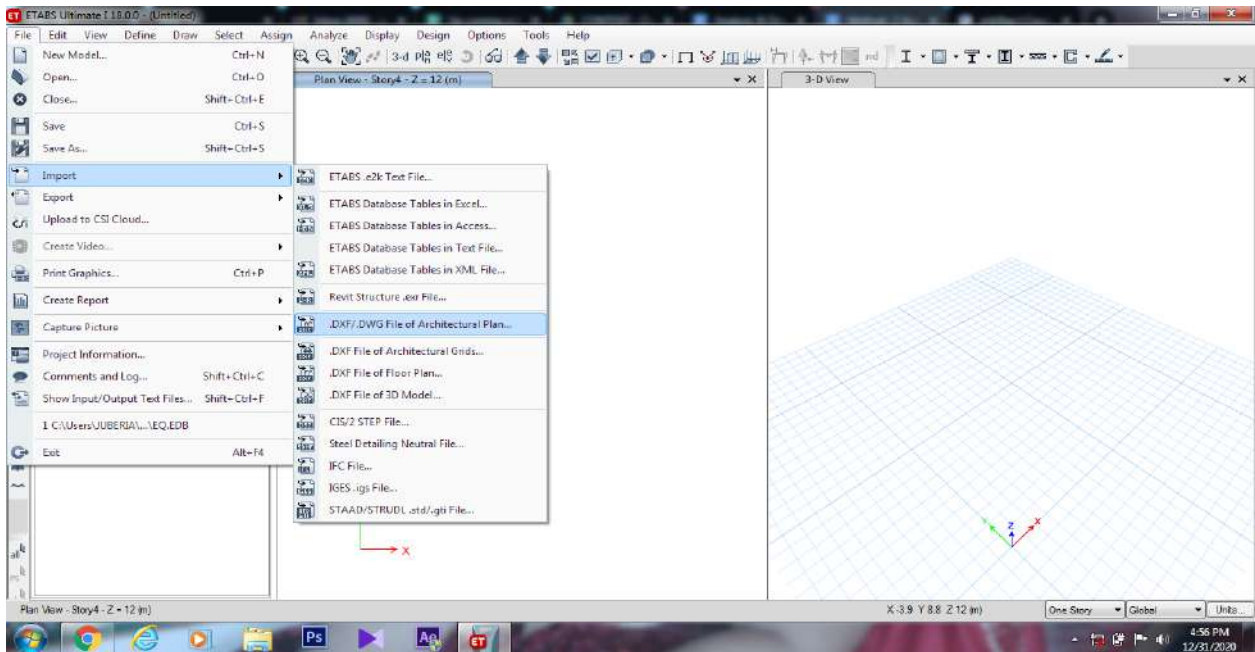
1. Plot the plan in AutoCAD using layers and save the file in .DXF format

Access the DXF Import form for importing .DXF/.DWG Architectural Grids as follows:

2. Click on **New Model** and 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

3. Select **Blank** template from **Add Structural Template**. By keeping remaining settings as default click **Ok**.
4. Click the **File menu > Import > .DXF/DWG Architectural Grids** command to access the **.DXF/DWG Import** form



5. Use the form to locate the filename/path of the .DXF file to be imported.
6. Highlight the filename and double click it or click the **Open** button to access the **DXF Import** form

ET Architectural Plan Import

CAD Entities Import

☒ Overwrite existing CAD entities
☐ Add to existing CAD entities

CAD Drawing Units

mm
Story4

Drawing Origin and Scale

Global X Ordinate: 0 m
Global Y Ordinate: 0 m
Rotation: 0 deg
Scale Factor: 1

Center Line Creation for Beams/Walls

☒ Add Center Lines

Max. Distance Between Parallel Lines: 1 m
Min. Distance Between Parallel Lines: 0.01 m

OK Cancel

Fig: Architectural Plan Import Form

7. Set DXF File units, Plan location and then click on **OK** to Import the plan
8. Go to **Define menu > Material Properties**, Click on **Add New Material** option and add M30 grade concrete and HYSD500 grade rebar using **Add New Material Property** form.

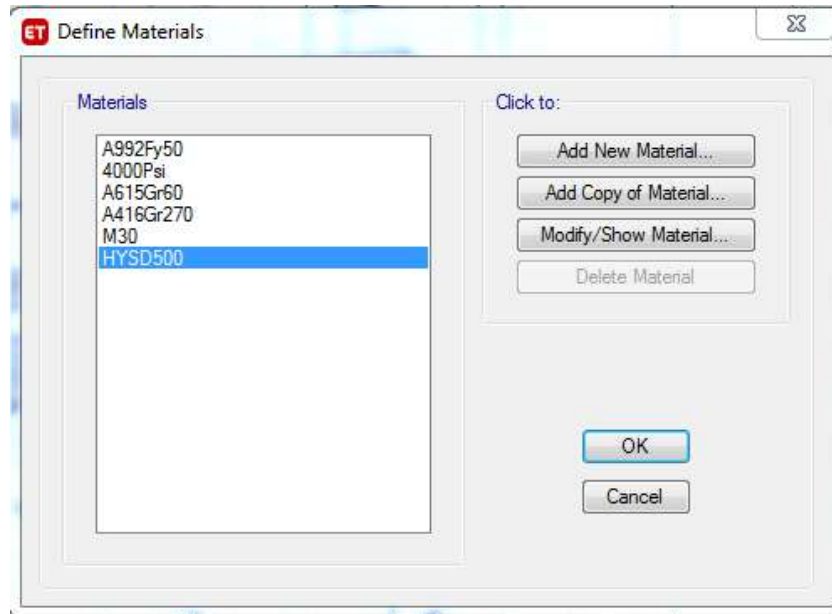


Fig: Define Material Form

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

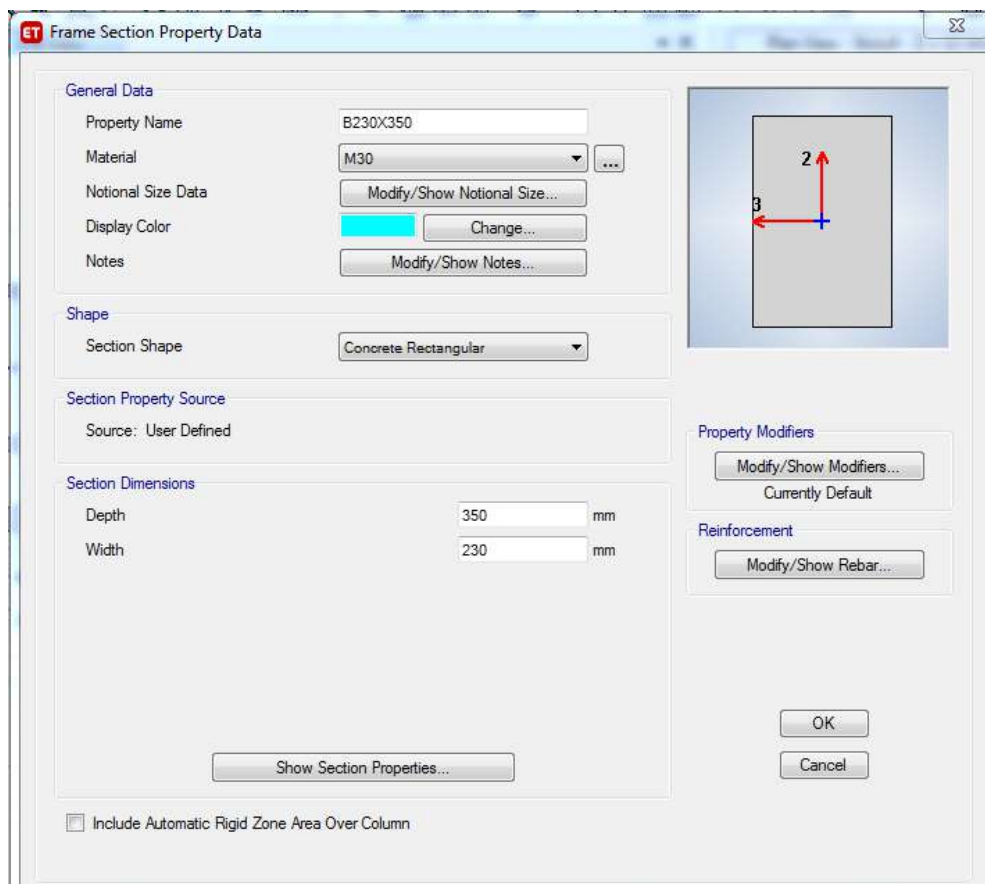
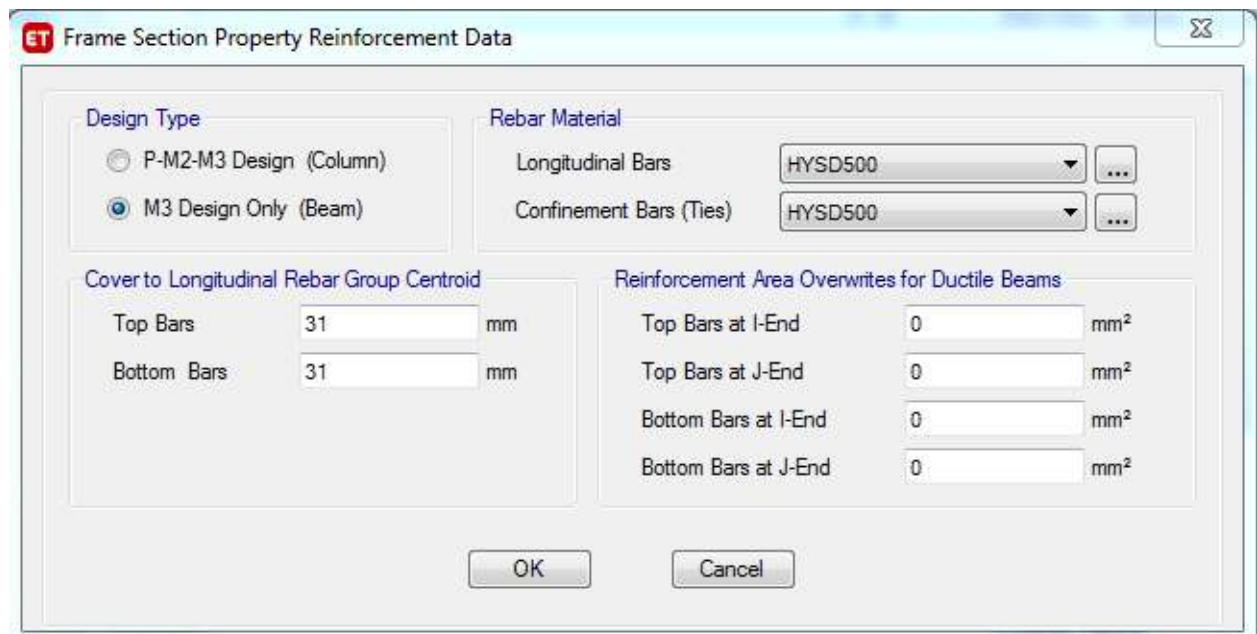


Fig: Frame Section Property Data form (Beam Definition)

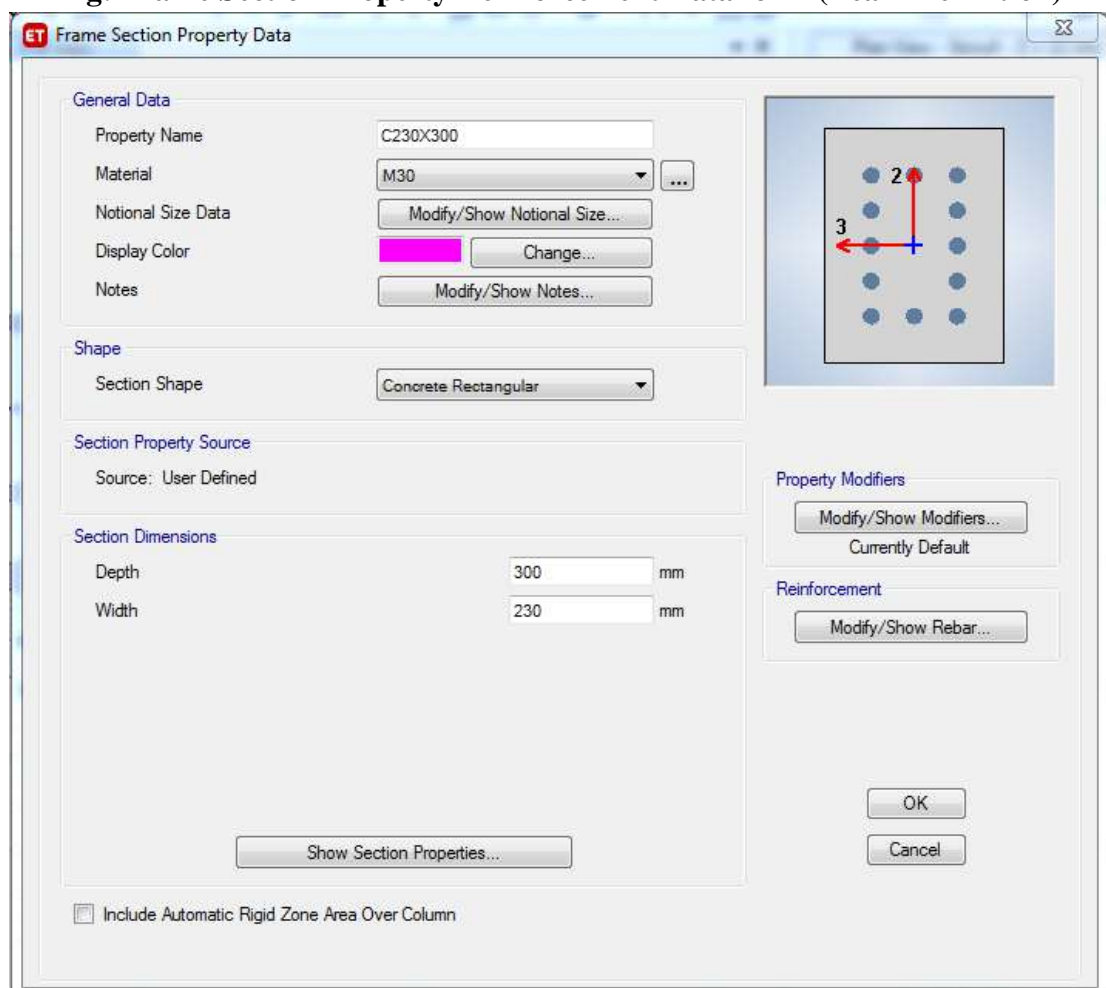
- Click on **Modify/Show Rebar** to Specify the design type, rebar materials & cover as per the requirements as shown in following figure



The image shows a software window titled "ET Frame Section Property Reinforcement Data". It contains several input fields and buttons for defining beam reinforcement.

- Design Type:** Two radio buttons are present: "P-M2-M3 Design (Column)" and "M3 Design Only (Beam)". The "M3 Design Only (Beam)" option is selected.
- Rebar Material:** Two dropdown menus are shown: "Longitudinal Bars" and "Confinement Bars (Ties)". Both are set to "HYSD500".
- Cover to Longitudinal Rebar Group Centroid:** Two input fields: "Top Bars" and "Bottom Bars", both set to "31 mm".
- Reinforcement Area Overwrites for Ductile Beams:** Four input fields: "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".
- Buttons:** "OK" and "Cancel" buttons are at the bottom.

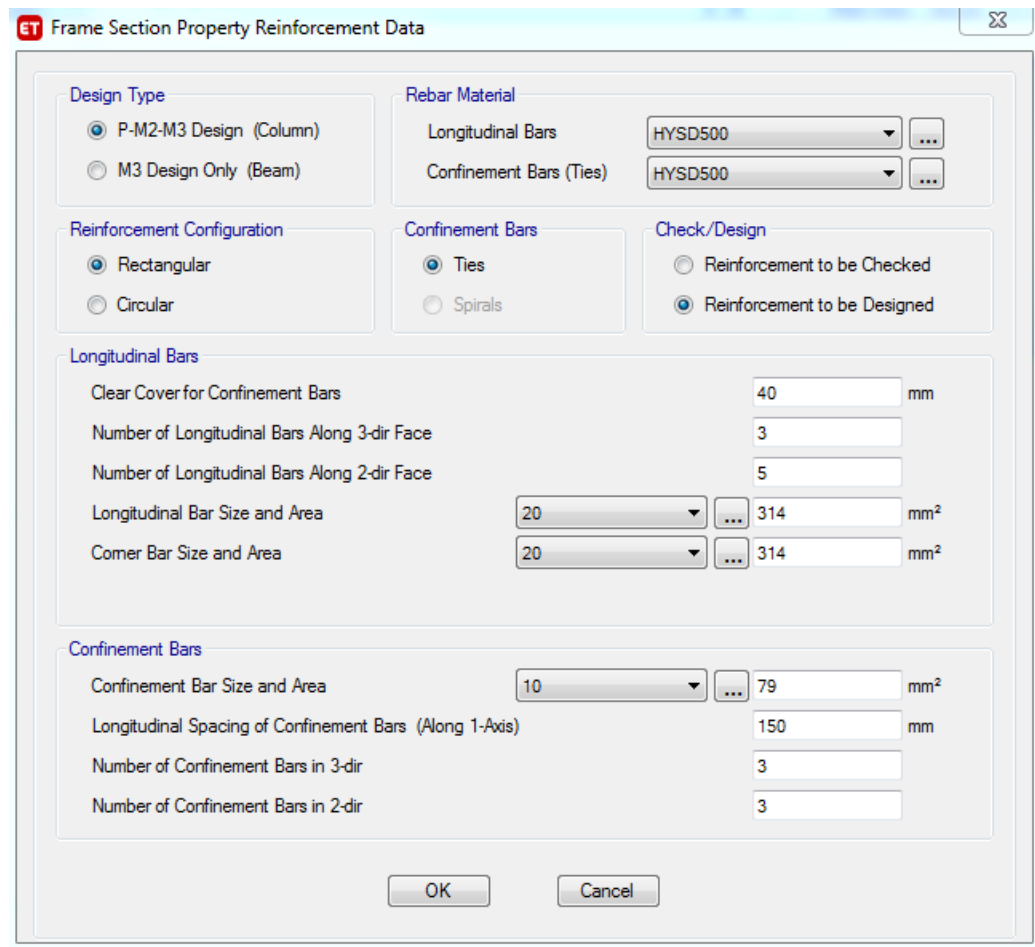
Fig: Frame Section Property Reinforcement Data form (Beam Definition)



The image shows a software window titled "ET Frame Section Property Data". It contains several input fields and buttons for defining column properties.

- General Data:**
 - Property Name: "C230X300"
 - Material: "M30"
 - 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: "300 mm"
 - Width: "230 mm"
- Property Modifiers:**
 - "Modify/Show Modifiers..."
 - "Currently Default"
- Reinforcement:**
 - "Modify/Show Rebar..."
- Buttons:** "OK" and "Cancel" buttons are at the bottom right.
- Checkbox:** "Include Automatic Rigid Zone Area Over Column" is at the bottom left.

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: HYSD500

Confinement Bars (Ties): HYSD500

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 mm²

Cornor Bar Size and Area: 20 314 mm²

Confinement Bars

Confinement Bar Size and Area: 10 79 mm²

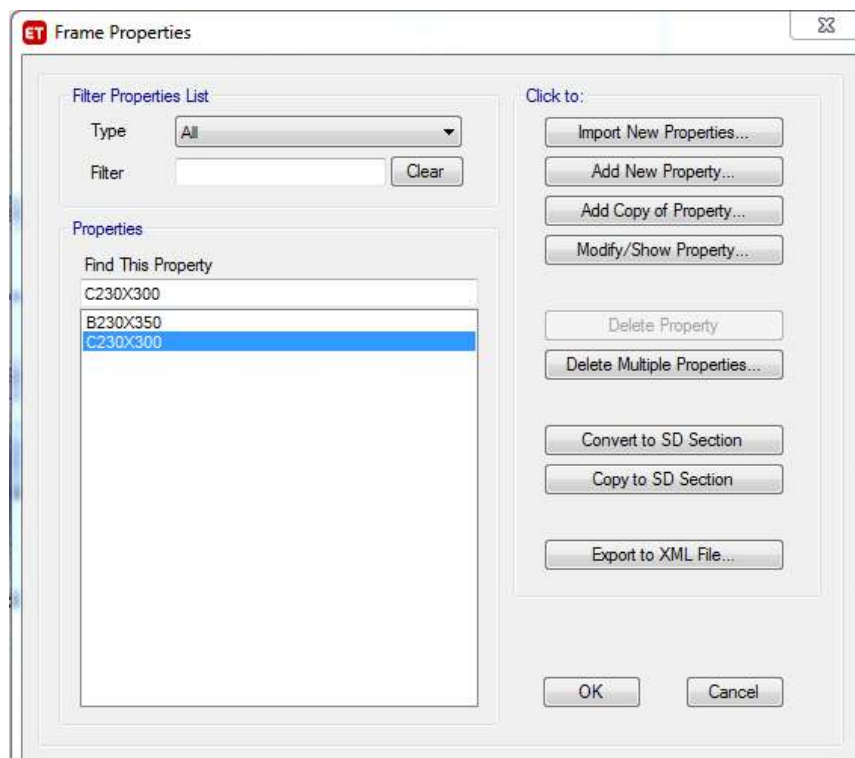
Longitudinal Spacing of Confinement Bars (Along 1-Axis): 150 mm

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)



ET Frame Properties

Filter Properties List

Type: All

Filter: Clear

Properties

Find This Property

- C230X300
- B230X350
- C230X300**

Click to:

- Import New Properties...
- Add New Property...
- Add Copy of Property...
- Modify/Show Property...
- Delete Property
- Delete Multiple Properties...
- Convert to SD Section
- Copy to SD Section
- Export to XML File...

OK Cancel

Fig: Frame Properties form

11. Click the **Define menu > Section Properties > Slab Sections** command to access the **Slab Properties** form. Click on **Add New Property** button to add Slab of 150mm thickness as shown below

Fig: Slab Property Data form

Fig: Slab Properties form



12. To assign the beam property go to **Options> Architectural Plan Options>** Deselect all the layers except beam. Now activate **Quick Draw Beam** tool and specify the properties of the beam, in the **Draw Object Using** select Ach.layers option to assign the beams to beam layer by drawing a window. Similarly assign the remaining structural elements.
13. Assign the main wall load & partition wall load on the respective beams as per the plan by using **Frame Loads**
14. Assign the Slab loads(Floor finish load & live load) using **Shell Loads**
15. Select all the elements in the viewport and replicate it to the remaining stories
16. To Assign Supports keep the story settings to **one story** >open plan view of **base story** and select the joints at base using windows selection, go to **Assign > Joints > Restraints**, and assign Fixed support to bottom joints.
17. To define the load combinations go to **Define Menu> Load Combinations> Add Default Design Combos**, select **Concrete Frame Design** and click **OK**
18. 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**.
19. As a result of **Check Model** you will receive a warning message stating the errors in model, if the model is free from errors and mistakes it display warning message
20. To run the analysis click the **F5** function key or choose **Run Analysis** from **Analyze** menu. As the analysis completes it shows the deflection diagram initially.
21. To check the results like BMD or SFD click on **Display Frames/Piers/Spandrels/Links** or **F8**, select Load case > select Moment 3-3 or Shear 2-2 respectively and click on **OK**
22. To check the results of slabs **Display Shell Stresses/Forces** or **F9**
23. After analysis go to **Design menu > Concrete Frame Design> View Preferences**, check the Design parameters.
24. Go to **Design menu > Concrete Frame Design > Start Design Check**
25. To check the failure members go to **Design menu > Concrete Frame Design > Verify all members Passed**