



Andhra Pradesh State Skill Development Corporation



Andhra Pradesh State Skill Development Corporation



Extended Three-Dimensional Analysis of Building System

ETABS

Workflow of ETABS



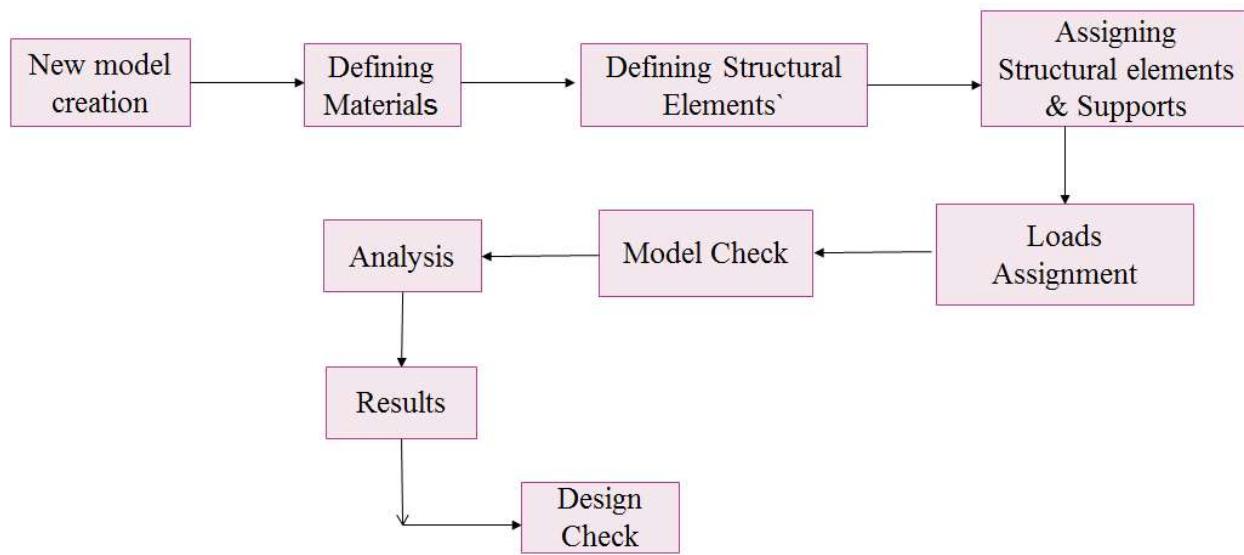
WORKFLOW

Objective

This chapter contains a brief explanation on ETABS workflow.

Workflow

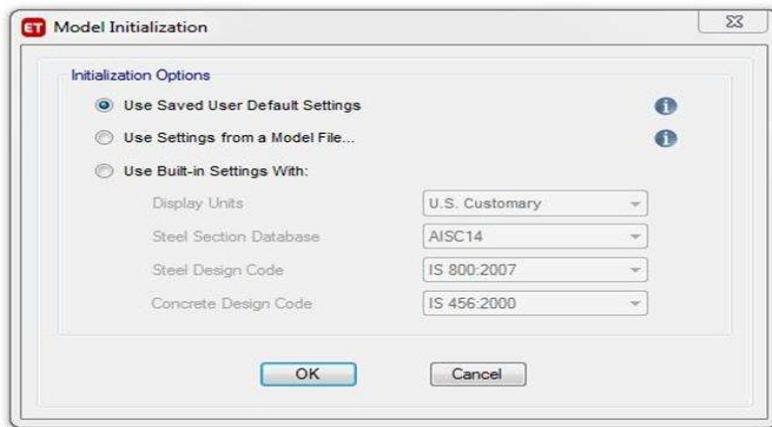
To perform any analysis and design in ETABS a standard procedure has to be followed. The following flowchart represents the steps involved in ETABS for the analysis and design of structures. It involves 9 steps as listed below



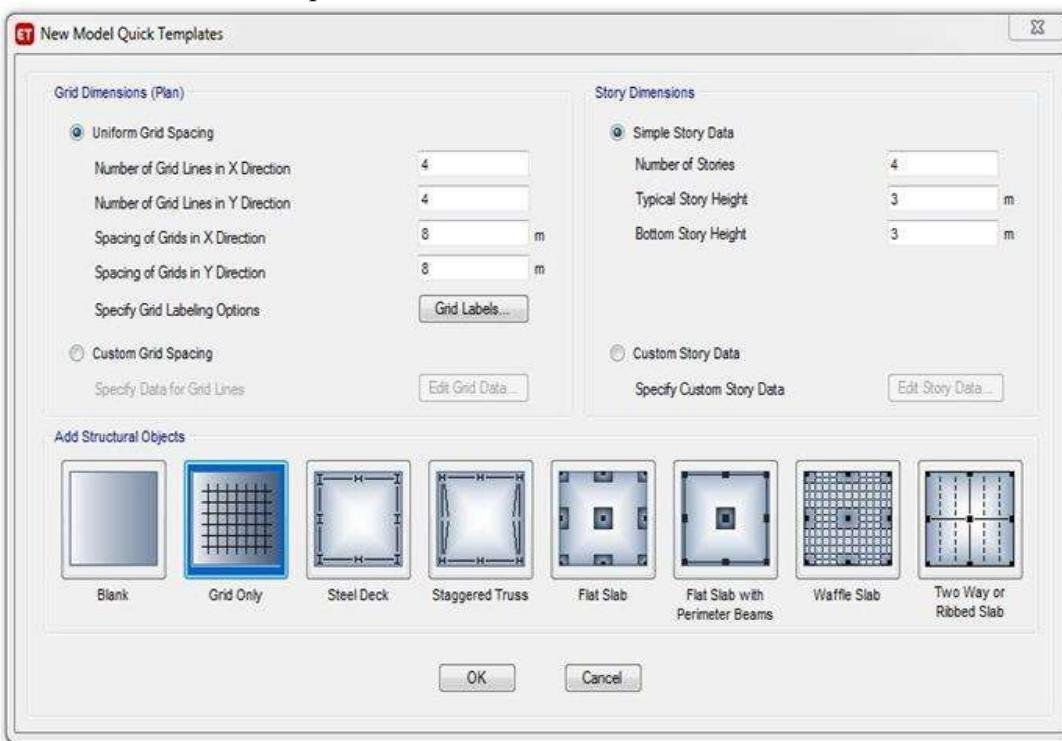
Level-1: New Model Creation

In this level you have to choose the method of opening new model and the units setup. This involves 2 steps

1. **Model Initialization:** In this step you have to specify the method to open **New Model** and you have to specify the Display Units & Design codes

**Figure: Model Initialization Form**

2. **New Model Quick Template:** In this step you have to specify the Grid Dimension, Story Dimensions and the Structural Template
3. As you specify grid dimensions and story dimensions, as a result you will get a 3D grid generated, which resembles the positions of columns and beams.

**Figure: New Model Quick Template Form**

Level-2: Defining Materials



Material property of any structural member refers to the engineering properties, such as cube compressive strength of concrete which decides the grade of concrete and tensile strength or yield stress of the steel, based on which the steel rebar grade is identified.

Eg: (I) For concrete M-20, M-30, M-40 etc.

(II) For Steel Fe 415, Fe 500 etc.

In this step we will define the materials (concrete & steel) as per the requirement

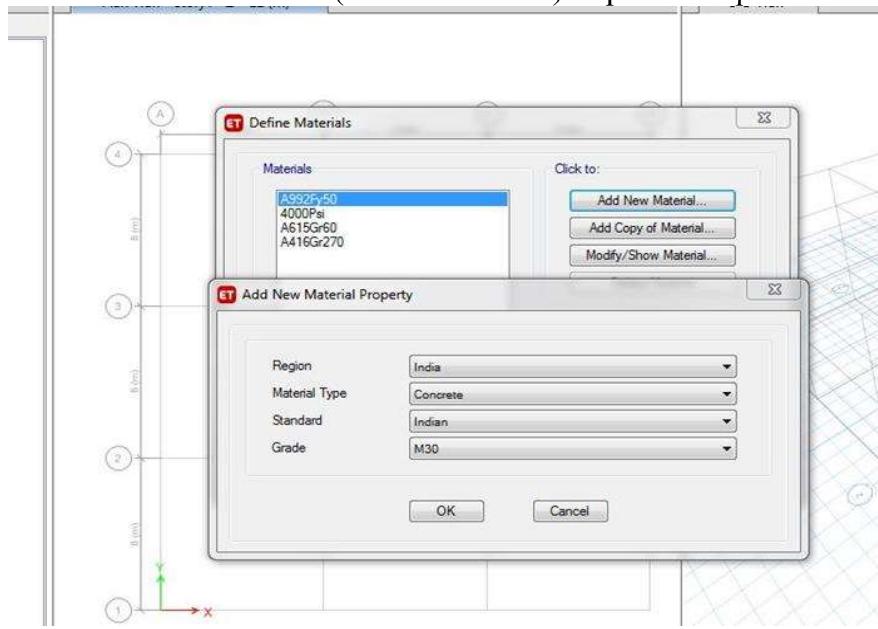


Figure: Define Material Form & Add New Material Property Form

Level- 3: Defining Structural elements

In this step you have to create the sections for beams, columns and slabs that you are going to use in the project.

To define beams and columns **Frame Sections** option is used as shown in the following fig, similarly to define slab sections **Slab Sections** option is used.

- Beam Sections

- Ex: B230x400, B300x450,

- Column Sections

- Ex: Col300x300, Col230x300,

- Slab Sections

- Ex: Slab 120mm,

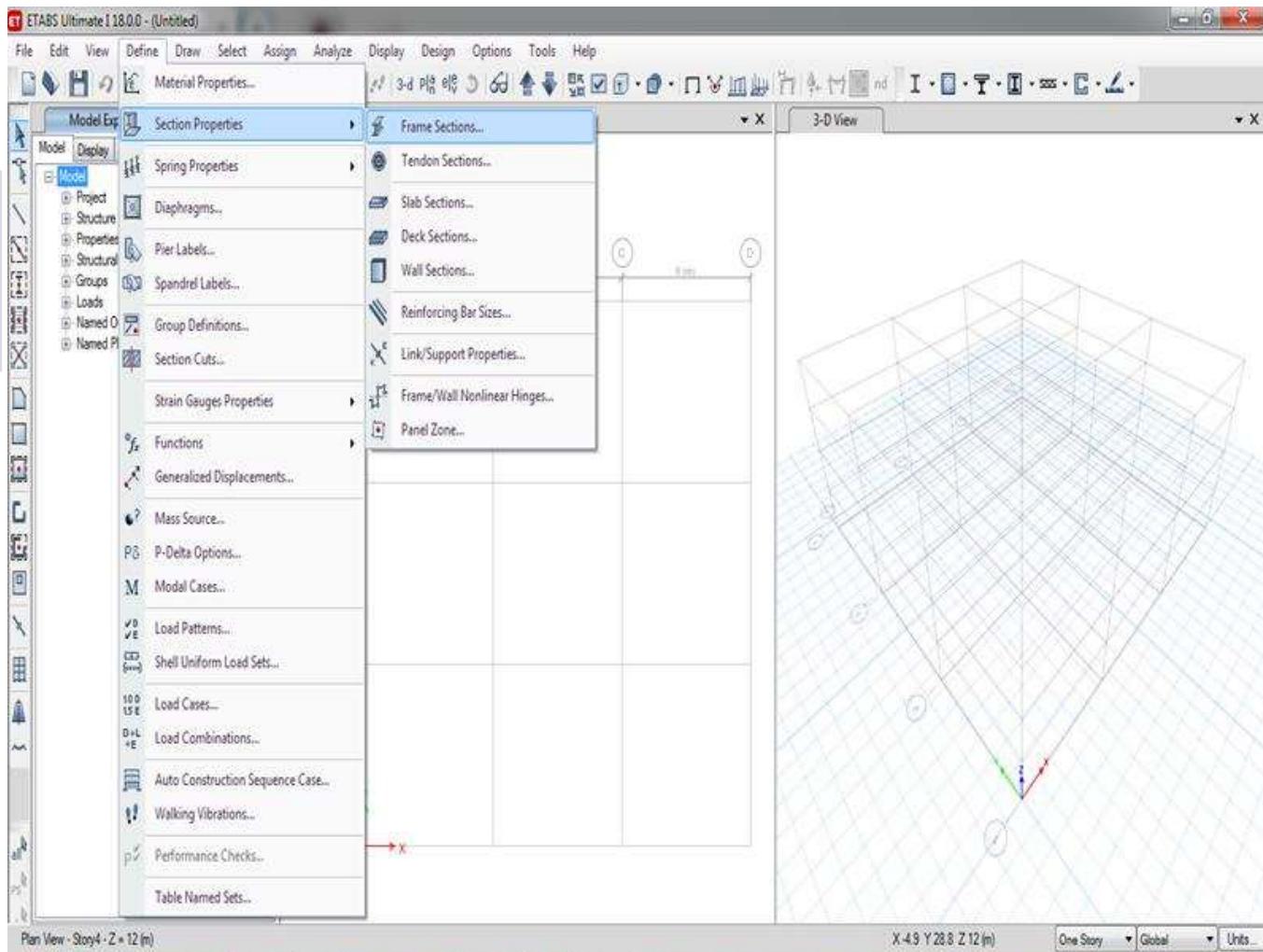


Figure: Define Menu

Level- 4: Assigning Structural elements & Supports

1. Assigning Structural elements :

This step involves assigning defined structural elements of **Level: 3** to the Empty grid generated in **level-1** by using various draw tools.

A. Fig-1 shows the empty 3d grid which was created in level-1

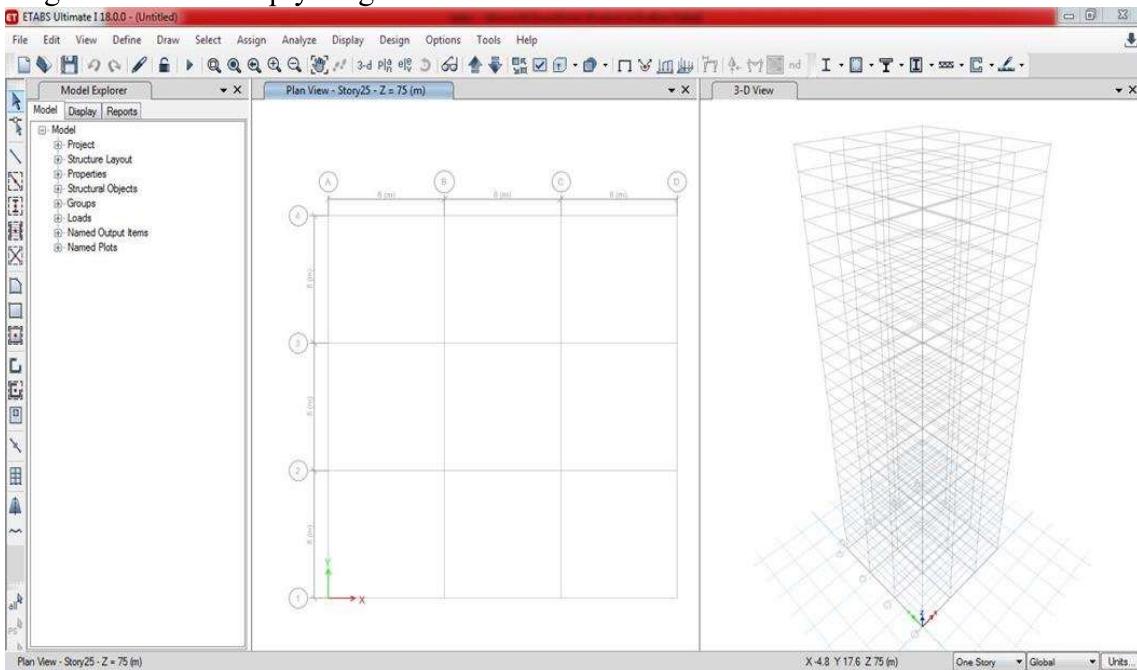


Fig-1

B. Fig 2 shows the draw tools which are used to assign the structural elements to the grid

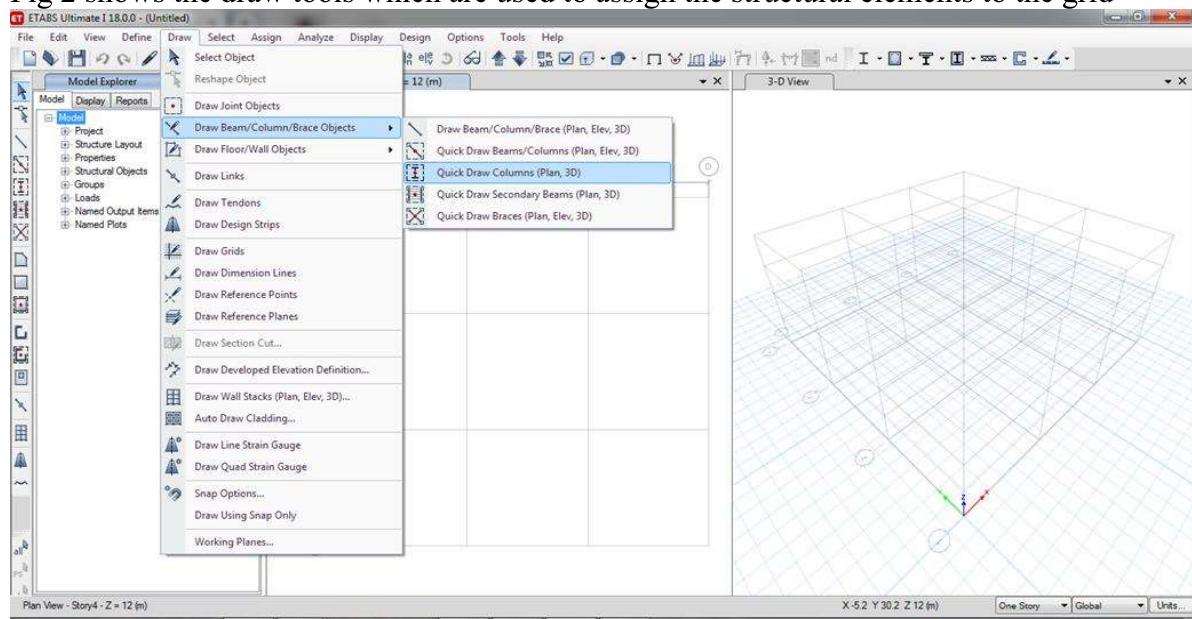


Fig-2

C. Fig-3 shows the final output of Level-4... i.e view after assigning the structural elements

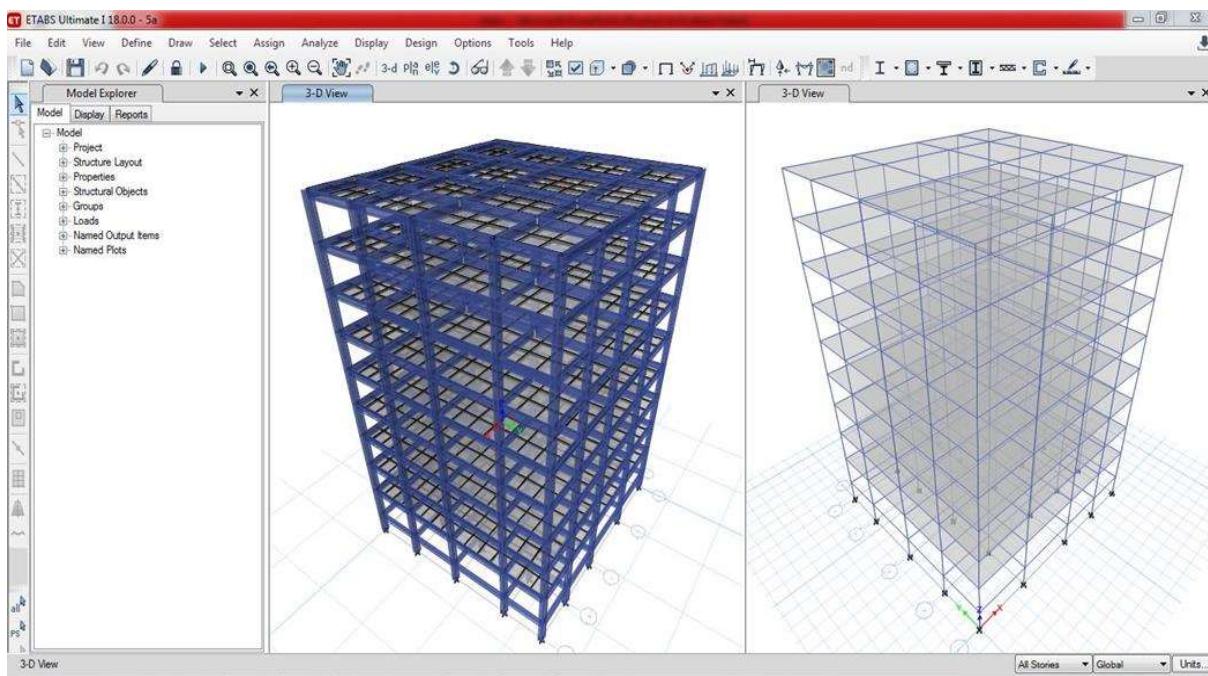


Fig-3

2. Assigning Supports:

In this step you have to assign the support conditions

Ex: Generally for a building at bottom fixed supports has to be assigned to indicate footings.

The following fig represents the various supports assignment to the structure

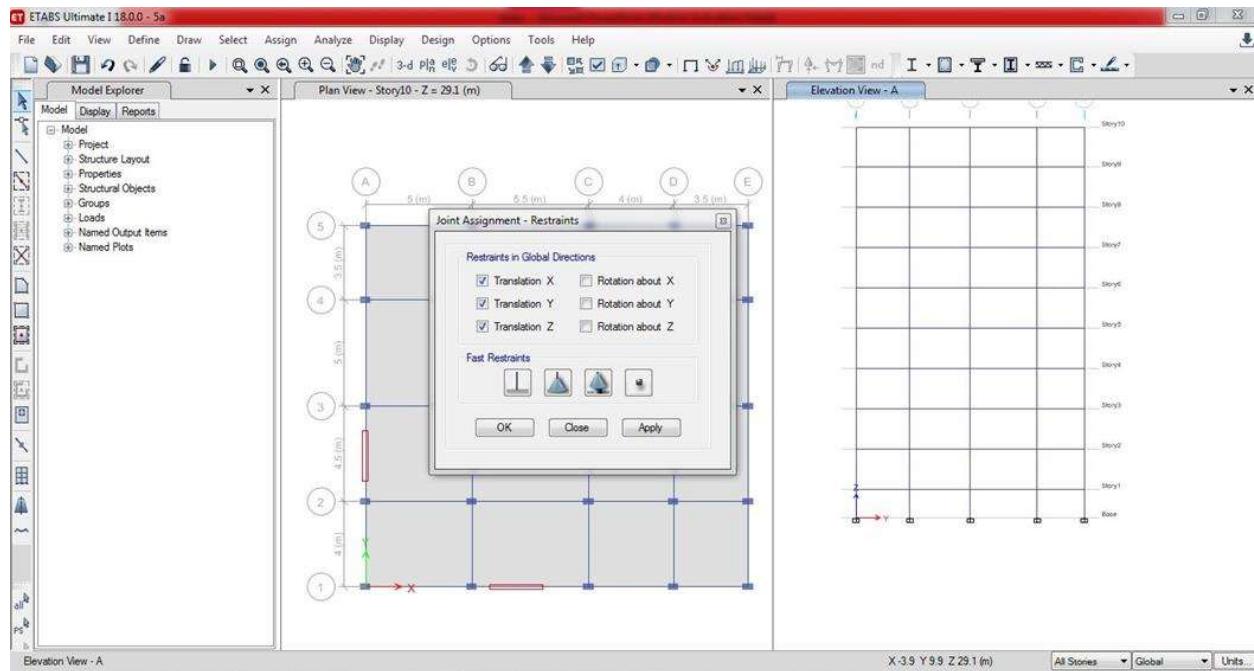


Fig-4

Level-5: Load Assignments

This step involves defining and assigning various loads on to the structure like dead load, live load and lateral loads. Along with loads Load combinations are also defined by using the **Define menu**. The following fig shows structure with assigned loads

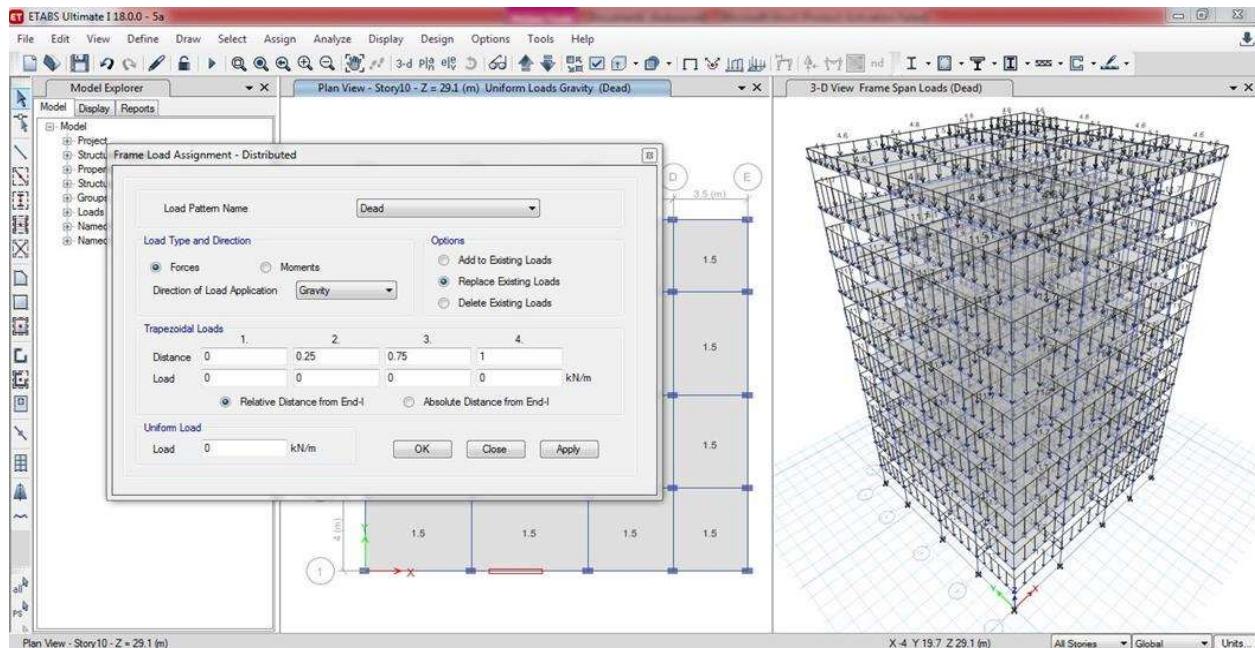
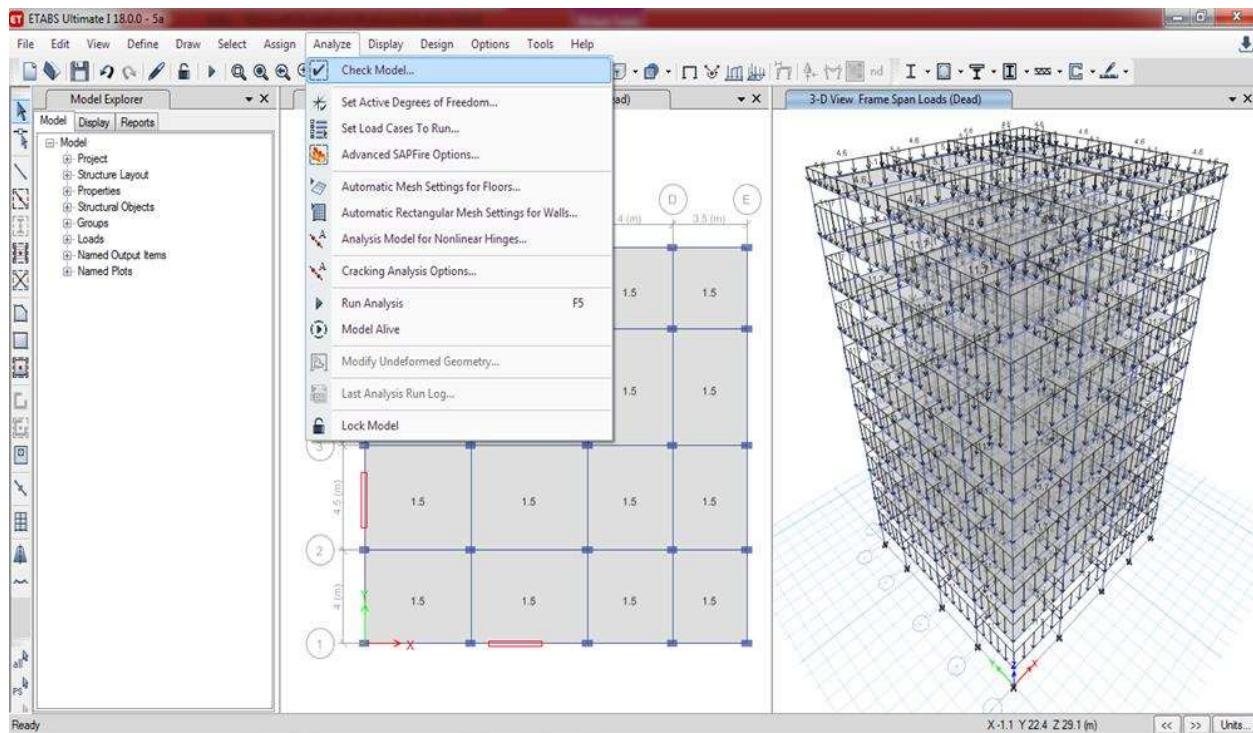


Fig-5

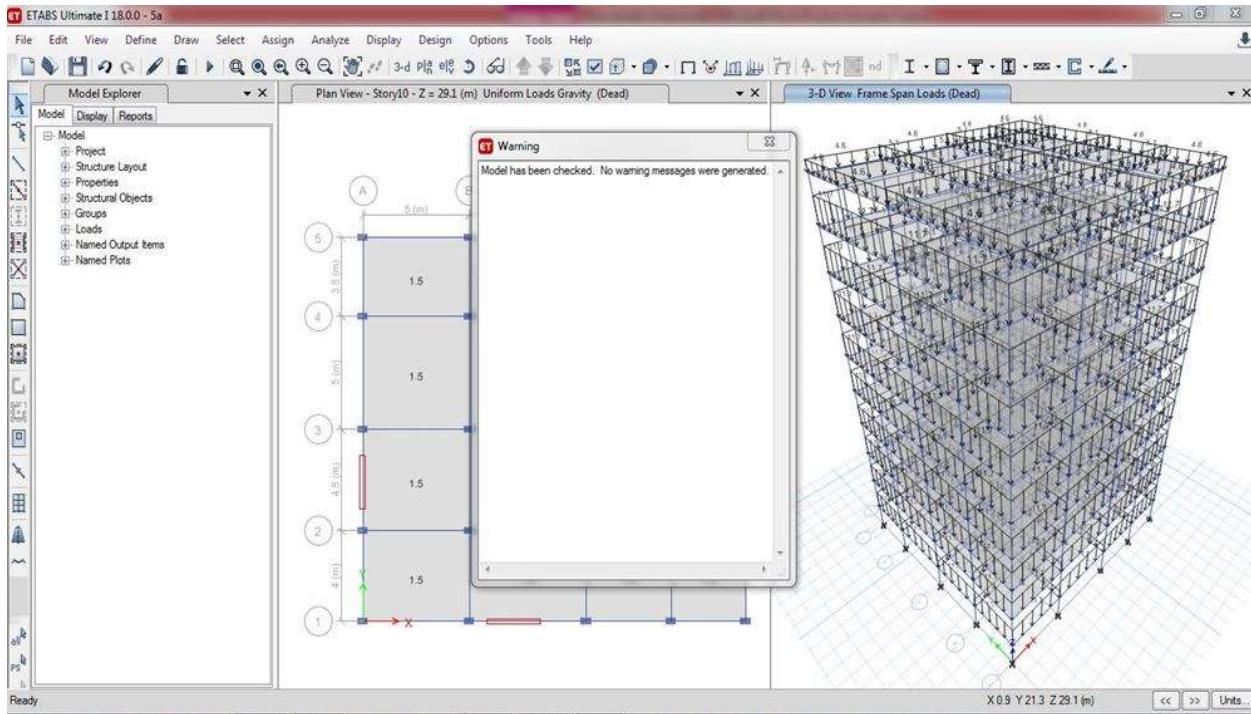
Level-6: Model Check

Fine tuning of the ETABS model before the analysis and design is necessary to make sure that our model is free of errors and warning messages. As these errors may affect the output results if we are not doing so. After we finish the ETABS modeling geometrically and assigned the necessary parameters, the next thing to consider is to review the model.

Check Model option will be present in **Analyze Menu** as Shown below

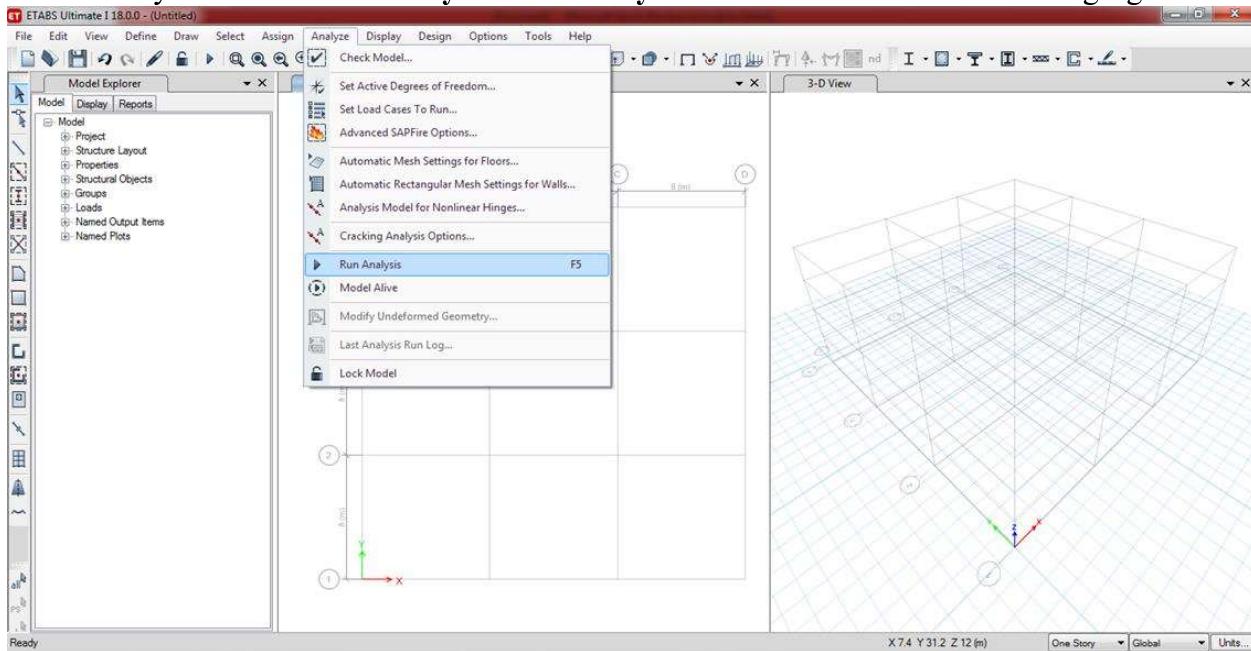


As a result of **Check Model** you will receive a warning message stating the errors in model, if the model is free from errors and mistakes it display warning message as shown in below fig



Level-7: Analysis

In this step you will run the analysis for the generated structure. To run the analysis click **F5** function key or choose **Run Analysis** from **Analyze** menu as shown in the following fig

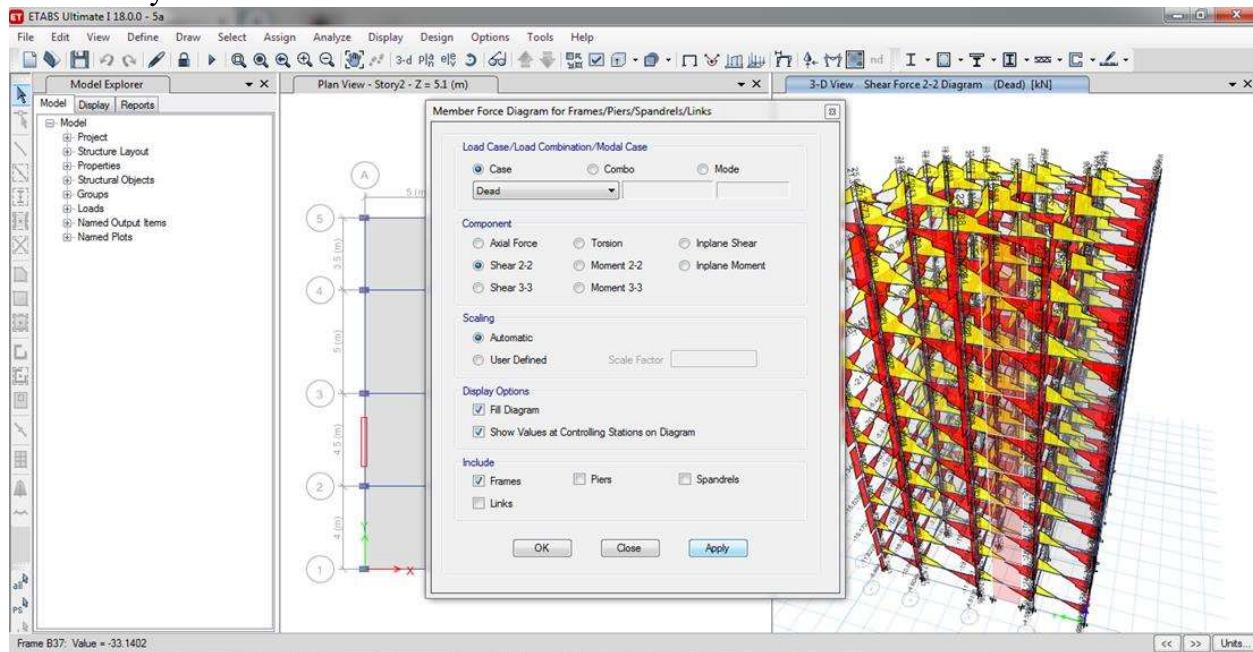


Level-8: Results Interpretation

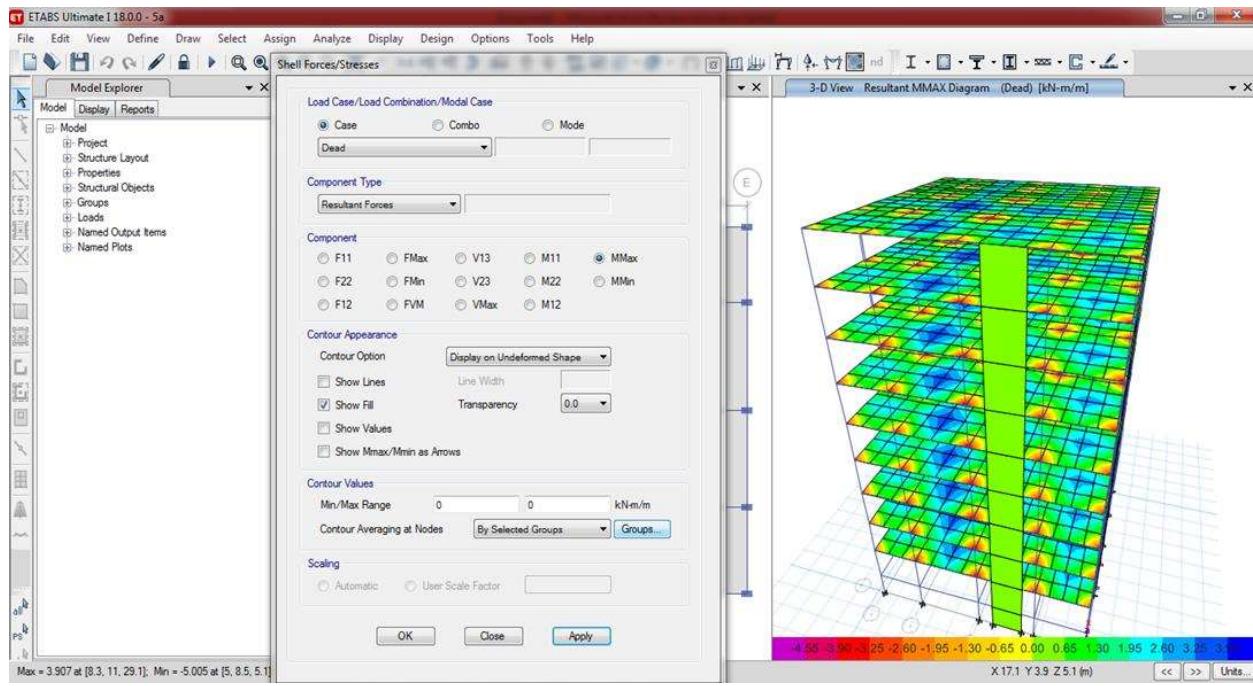
After analysis you are accessed to see results such as deflection, bending moment, shear forces and reactions.



The results can be graphically represented as shown in fig for each and every structural element individually.



It displays results for all the assigned loads and load combinations individually.



Level-9: Design Check

This step involves the design of structure for the obtained analysis results. As a result it shows the area of reinforcement required in the structural elements. And also it will detect the failure sections. Design can be performed for all the load cases and load combinations assigned to the structure. The following fig shows the design results of a structure

