International Journal of Engineering Sciences & Research Technology

(A Peer Reviewed Online Journal)
Impact Factor: 5.164

IJESRT

RESEARCHERID
THOMSON REUTERS

[Maruthi, *et al.*, 8(12): December, 2019]
IC™ Value: 3.00

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

ANALYSIS AND DESIGN OF COMMERCIAL BUILDING USING ETABS

Maruthi T*[1], Pruthvi Raj S R[2] , Pramod YE[3], Raghavendra S[4] & Krishnamurthy K A[5]

[1,2,3,4&5]PG Students. Department of Civil Engineering, University B D T College of Engineering, Davangere, Karnataka, India

DOI: 10.5281/zenodo.3596339

ABSTRACT

Structural Analysis is a branch which involves in the determination of behaviour of structures in order to predict the responses of different structural components due to effect of loads. Each and every structure will be subjected to either one or the groups of loads, the various kinds of loads normally considered are dead load, live load, earth quake load and wind load. ETABS (Extended Three Dimensional Analysis of Building System) is a software which is incorporated with all the major analysis engines that is static, dynamic, Linear and non-linear, etc. The main purpose of this software is to design multi-storeyed building in a systematic process. The effective design and construction of an earthquake resistant structures have great importance all over the world. Our project "Analysis and Design of Commercial building using ETABS software" is an attempt to analyze and design a commercial building using ETABS. Analysis is carried out by static method and design is done as per IS 456:2000 guidelines. Also an attempt has been made to design the structural elements manually.

KEYWORDS: ETABS, Analysis, Building, Design.

1. INTRODUCTION

The term building in Civil Engineering is used to mean a structure having various components like foundation, walls, columns, floors, roofs, doors, windows, ventilators, stairs lifts, various types of surface finishes etc. Structural analysis and design is used to produce a structure capable of resisting all applied loads without failure during its intended life. Etabs is a stand-alone structural analysis program with a special purpose features for structural design and analysis of building systems. ETABS is simple to use and user-friendly and it is unique in its ability to address the full spectrum of tasks involved in the process of structure analysis and design. ETABS is a very suitable package for, Multi-storied building analysis. The entire input data may be generated either graphically or by typing simple English language based commands. It is equipped with the sophisticated algorithms and state of the art graphics, residing in an extremely user-friendly environment.

1.1 Features and Benefits of ETABS

- The input, output and numerical solutions technique of ETABS are specifically designed to take advantage of the unique physical and numerical characteristics associated with building type structures.
- The need for the special purpose program has never been more evident as structural engineers put nonlinear dynamic analysis into practice and use the greater computer power available today to create a larger analytical model.
- Over the past decades, ETABS as numerous mega projects to its credit and as established itself as the standard of the industry. ETABS software is clearly recognized as the most practical efficient tool for the static and dynamic analysis of multi-storey frame and shear wall buildings.

1.2 Objectives

To perform analysis and design of the structure without any type of failures.

1. To understand the basic principles of structures by using Indian Standard Codes.
2. To design structural components like beam, slab, column and footing manually.
3. Comparison of results obtained from ETABS software with a manual method.

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

Scope of study
The scope of the study is to produce good structural work for performing analysis and design for a residential building.

2. METHODOLOGY

To achieve the objectives of the study that is to analyze and design commercial building using ETABS and by manual method, which meets the basic requirements such as safety, durability, economy, aesthetic appearance, feasibility, practicability and acceptability. It has been proposed to follow the following methodology.

- Site survey
- Soil investigation
- Structural planning
- Analysis in ETABS
- Verification by manual method

Procedure for determining the Etabs
Step - 1: Initial setup of Standard Codes and Country codes

Step - 2: Creation of Grid points & Generation of structure
Firstly, we have to opened the ETABS we select a new model and a window appears where we had entered the grid dimensions and story dimensions of our building.

Step - 3: Defining of property
Here we had first defined the material property by selecting define menu material properties and add new material for our structural components (like beams, columns, slabs) by giving the specified details in defining. After that we define section size by selecting frame, beams, column sections etc.

Step - 4: Assigning of Property
After defining the property of each material we draw the structural components using command menu. Draw line for beam for beams and create columns in region for columns by which property assigning is completed for beams and columns.

Step - 5: Assigning of Supports
By keeping the selection at the base of the structure and selecting all the columns we assigned supports by going to assign menu joint/frame Restraints fixed.

Step - 6: Defining of loads
Defining of loads in ETABS all the load considerations are first defined and then assigned. The loads in ETABS are defined as using static load cases command in define menu.

Step - 7: Assigning of Dead loads
After defining all the loads. Dead loads are assigned for external walls, internal walls in staad but in E-TABS automatically taken care by the software i.e., inbuilt only

Step - 8: Assigning of Live loads
After assigning of dead load we have to assign Live loads are assigned for the entire structure including floor finishing.

Step - 9: Assigning of wind loads
Wind loads are defined and assigned as per IS 875 1987 PART 3 by giving wind speed and wind angle. But since this is a Residential Building having total height less than 12 meters there is no need of assigning of wind loads or earth quake loads.

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

Step - 10: Assigning of Seismic loads
Seismic loads are defined and assigned as per IS 1893: 2002 by giving zone, soil type, and response reduction factor in X and Y directions. But since this is a G+1 Residential Building having total height less than 12 meters there is no need of assigning Seismic loads.

Step - 11: Assigning of load combinations
After assigning of all loads we have to take total load Using load combinations command in define menu 1.5 times of dead load and live load will be taken as mentioned in above.

Step - 12: Analysis
After the completion of all the above steps we have performed the analysis and checked for errors.

Step - 13: Design
After the completion of analysis, we had performed concrete design on the structure as per IS456:2000. ETABS performs the design for every structural element.

3. DESCRIPTION OF THE STUDY MODEL

Modelling is done by using ETABS (9.7 version) software. For this purposes the preliminary step is to prepare grid layouts. As our selected structure is (G+1) story, we prepared plinth beam layout, floor bema layout and roof bema layout and by using this layout we prepared grid system using ETABS. The next step is to define the materials properties, beam details, column details, slab details and the load combinations by referring to IS 456-2000.

3.1 Material properties:

- Material name: Concrete
- Type of material: Isotropic
- Density of concrete: 25 KN/m^3
- Poisson's ratio: 0.2
- Grade of concrete: M25
- Grade of steel: Fe500

3.2 Beam properties

- Beam size:
  - Plinth beam = (230x300) mm
  - Floor Beam = (230x300) mm
  - Roof Beam = (230x300) mm
  - Far beam = (115x300) mm
- Material: Concrete
- Grade of concrete: M25
- Grade of steel: Fe500

3.3 Column details:

- Column size:

Column = (230x300) mm

- Type of column: Rectangular
- Grade of concrete: M25
- Grade of steel: Fe 500

3.4 Slab properties:

- Material: Concrete
- Type: Membrane
- Thickness: 150mm

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

3.5 Load combinations:

Analysis is carried out for gravity loads using partial safety factor as 1.5. The following Loads have been considered in the structural analysis and design as per IS code 456-2000.

- Density of RCC= 25KN/m^3
- Density of burnt brick masonry = 19 KN/m^3
- Floor
    1. Live load= 3KN/m^2
    2. Floor finishing= 1KN/m^2
- Roof live load= 1.5KN/m^2
- Burnt brick masonry wall thickness= 230mm.
- Parapet wall thickness = 115mm.

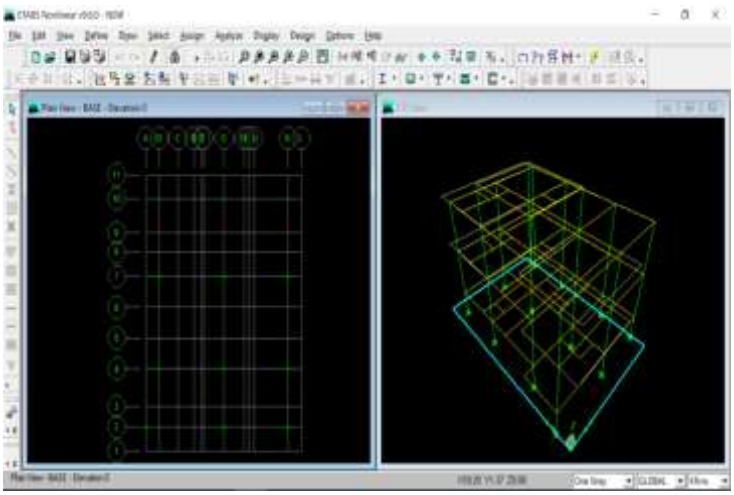

Fig 3.1: Grid line of the Architectural Plan

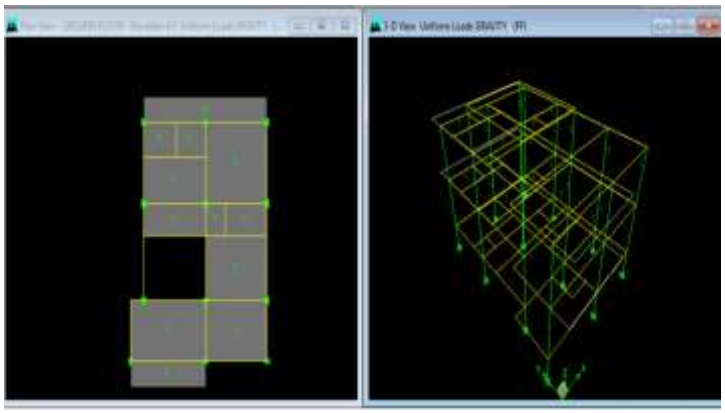

Fig 3.2: Model After Assigning Loads

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

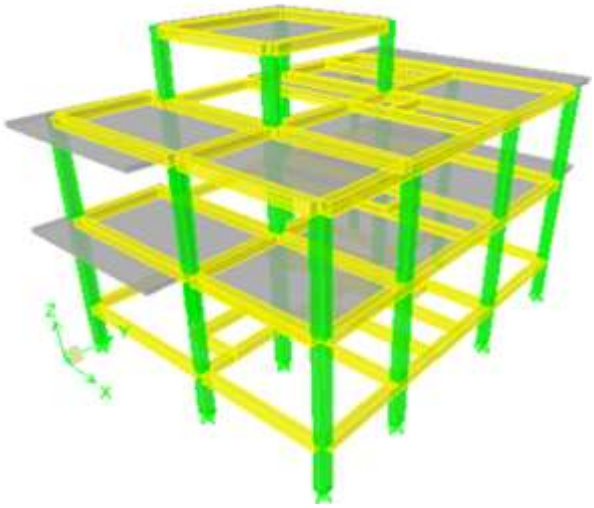

Fig 3.3: Rendered Model of the Structure

4. RESULTS AND DISCUSSIONS

The tests conducted for soil investigation are core cutter method (bulk density), oven dry test (water content), and direct shear test (shear parameter).

Table 4.1: Soil Survey Report

Density	25kN/m^3
Water content	15.50%
Cohesion, C	9kN/m^2
Angle of shearing resistance	200
Safe bearing capacity of soil	200kN/m^2

ETABS features an intuitive and powerful graphical interface coupled with modeling, analytical, and design procedures, all integrated using a common database. CAD drawings can be directly converted into ETABS models. Design of steel and concrete frames, beams, columns. Comprehensive and customizable reports are available for all analysis and design output and construction drawings of framing plans, details, and cross sections are generated for concrete and steel structures.

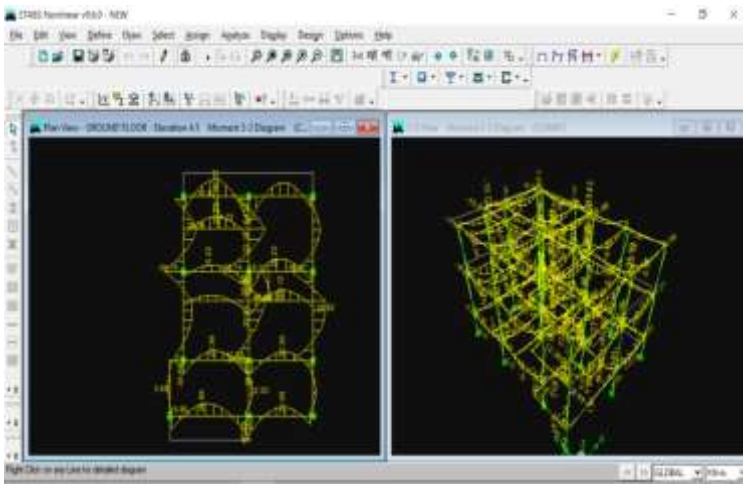

Fig 4.1: Bending Moment Diagram for first floor

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

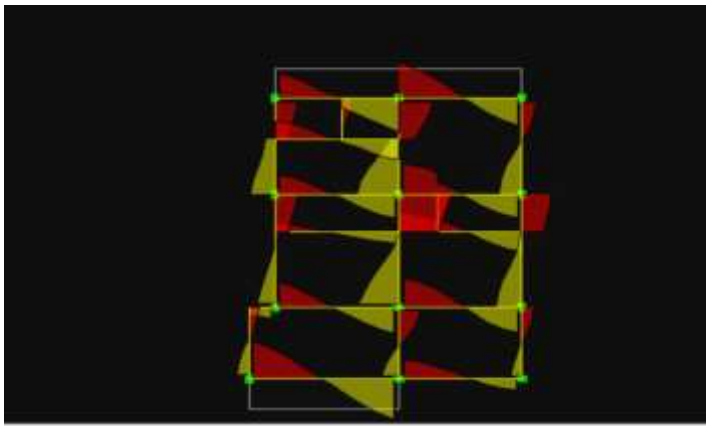

Fig 4.2: Shear Force Diagram for First Floor

Structural components like beam, slab, column and footing are designed manually. Results obtained from ETABS software is compared with manual method.

5. ACKNOWLEDGMENTS

We are grateful to our beloved Principal Dr. K S SRIDHAR, who always been thriving for enhancement and excellence for collage, department facilities and student welfare which a student desires.

We deeply indebted to Head of the Dept. DR. K. KRISHNEGOWDA and PG Coordinator Dr.H R PRABAKARA for his moral support and encouragement.

6. CONCLUSION

From the data revealed by the manual design as well as software analysis for the structures following conclusions are drawn:

- Analysis was done by using ETABS software and successfully verified manually as per IS456.
- Calculation by manual work and analysis gives almost same result.
- Usage of ETABS software minimizes the time required for analysis and design.

REFERENCES

[1] Varalakshmi V,G shivakumar and R S Sarma (2014) “Designed and d G+5 residential building by ETABS”, International Conference on Advance in Engineering and Technology.
[2] Chandrashekarand and Rajashekar (2015), “Analysis and Design of Multi Storied Building by Using ETABS Software”, International journals of scientific and research vol.4: issue.7: ISSN no. 2277-8179.
[3] Balaji and Selvarasan (2016), “Design and Analysis of multi-storeyed building under static and dynamic loading conditions using ETABS”, International Journal of Technical Research and Applications e-ISSN: 2320-8163, www.ijtra.com Volume 4, Issue 4, PP. 1-5
[4] Geethu S N, Depthi M, Abdul Nasir N A and Izzudeen K M(2016) “Comparative study on design and analysis of multi storied building by STAAD.Pro and ETABS softwares”.