

ANALYSIS AND DESIGN OF COMMERCIAL BUILDING USING ETABS

Ragy Jose¹, Restina Mathew², Sandra Devan³, Sankeerthana Venu⁴, Mohith Y S⁵

^{1,2,3,4}UG Students, Department of Civil Engineering, KVG College of Engineering, Sullia D.K

⁵Assistant Professor, Department of Civil Engineering, KVG College of Engineering, Sullia D.K

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. and especially this Software is used to analyze and design the buildings. Our project "Analysis and Design of Commercial building using ETABS software" is an attempt to analyze and design a commercial building using ETABS. A G+3 storey building is considered for this study. 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. Drawing and detailing are done using Auto CAD as per SP 34.

Key Words: ETABS, Building, Analysis, 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. Prior to the analysis and design of any structure, necessary information regarding supporting soil has to be collected by means of geotechnical investigation. A geotechnical site investigation is the process of collecting information and evaluating the conditions of the site for the purpose of designing and constructing the foundation for a structure. Structural engineers are facing the challenges of striving for most efficient and economical design with accuracy in solution while ensuring that the final design of a building and the building must be serviceable for its intended function over its design life time. Now a days various software packages are available in market for analyzing and designing practically all types of structures viz. RISA, STAADPRO, ETABS, STRUDL, MIDAS, SAP and RAM etc.

2. LITERATURE REVIEW

Varalakshmi V et.al (2014) ^[1] analyzed a G+5 storey residential building and designed the various components like beam, slab, column and foundation. The loads namely dead load and live load were calculated as per IS 875(Part I & II)-1987 and HYSD bars i.e. Fe 415 are used as per IS 1986-1985. They concluded that the safety of the reinforced concrete building depends upon the initial architectural and structural configuration of the total building, the quality of the structural analysis, design and reinforcement detailing of the building frame to achieve stability of elements and their ductile performance.

Chandrashekar et.al (2015) ^[2] analyzed and designed the multi-storeyed building by using ETABS software. A G+5 storey building under the lateral loading effect of wind and earthquake was considered for this study and analysis is done by using ETABS. They have also considered the chances of occurrence of spread of fire and the importance of use of fire proof material up to highest possible standards of performance as well as reliability. They suggested that the wide chances of ETABS software which is very innovative and easier for high rise buildings so that time incurred for designing is reduced.

Balaji.U and Selvarasan M.E (2016) ^[3] worked on analysis and design of multi-storeyed building under static and dynamic loading conditions using ETABS. In this work a G+13 storey residential building was studied for the earth quake loads using ETABS. They assumed that material property to be linear, static and dynamic analyses were performed. The non-linear analysis was carried out by considering severe seismic zones and the behaviour was assessed by considering type II soil condition. Different results like displacements, base shear were plotted and studied.

Geethu et.al (2016) ^[4] made a comparative study on analysis and design of multi storied building by STAAD.Pro and ETABS softwares. They provided the details of both residential and commercial building design. The planning was made in accordance with the national building code and drafted using Auto CAD software. They concluded that while comparing both software results, ETABS software shows higher values of bending moment and axial force.

2.1 Present Study

The present study is to analyze and design a G+3 storey commercial building of 3441.87sq.ft floor area located at Jatipalaya, Sullia (2km from KVG campus, Sullia).

2.2 Objectives

From the above literature review the following objectives are made;

- The main objective of this study is to analyze and design a G+3 commercial building using ETABS software.
- To design structural components like beam, slab, column and footing manually.
- Comparison of results obtained from ETABS software with a manual method.
- To draw and give reinforcement details of structural components by using Auto CAD.

3. 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 and design in ETABS
- Verification by manual method
- Detailing

Surveying is a basic tool for a Civil engineering science. Before any civil engineering work has to start, surveying has to be done and then we must prepare a plan or map of the area showing topographical details related to design of structure etc. Good planning and management of a geotechnical site investigation is the key to obtaining sufficient site information for designing a structure in a timely manner and with minimum cost for the effort needed. The engineering properties of soil like water content, density and SBC are calculated by conducting tests in laboratory. The structural plan is prepared using auto cad.

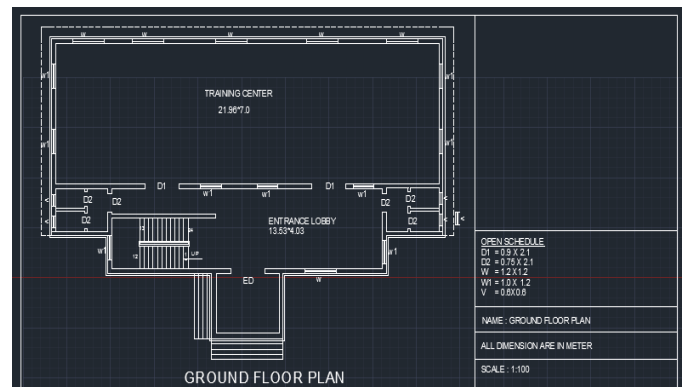


Fig - 1 Ground floor plan

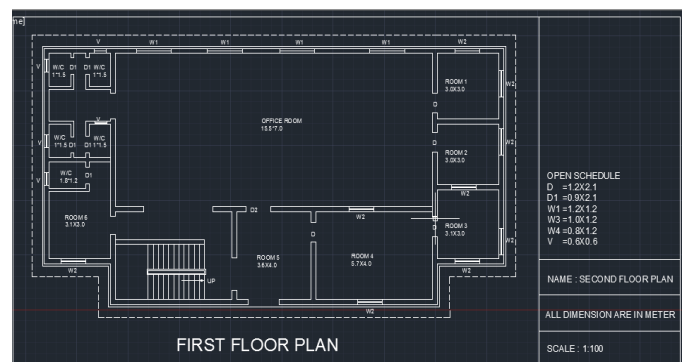


Fig - 2 First floor plan



Fig - 3 Second and third floor plan

The center line diagram is prepared and imported to ETAB model, and the following step by step procedures are followed:

Step - 1: Defining of property

Select define menu > material properties. Add new material in the defining material property the concrete of M25 and steel of grade Fe415. For our work the size of structural components (beams, columns, and slabs) are taken as per the requirement.

Table -1: Beam details

Beam No.	Size
Beam 1	230x450mm
Beam 2	230x600mm
Beam 3	230x750mm

Table -2: Column details

Column No.	Size
Column 1	230x450mm
Column 2	230x600mm
Column 3	230x750mm

Table -3: Slab details

Material	Concrete
Type	Membrane
Thickness	150mm

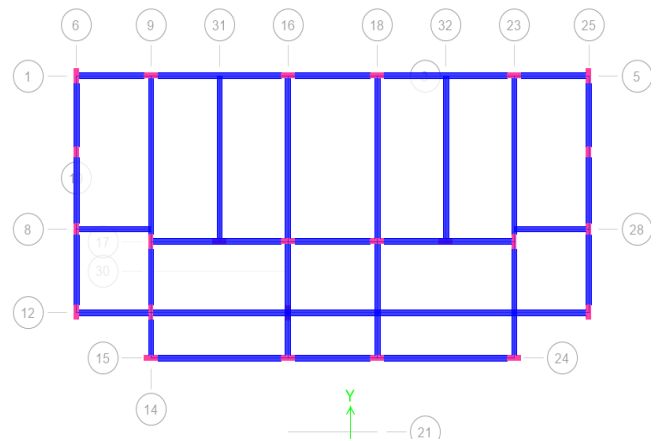


Fig - 4 plan of model

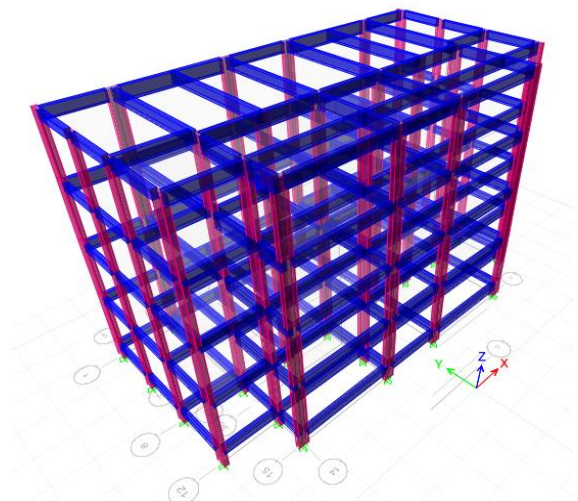


Fig -5 3D view of model

Step - 2: Assigning of Property

After defining the property we have to draw the structural components using command menu > Draw line for beam for beams and create columns in region for columns.

Step - 3: Assigning of Supports

By keeping the plan at the base of the structure and selecting all the columns supports are assigned by using assign menu > joint\frame > Restraints (supports) > fixed.

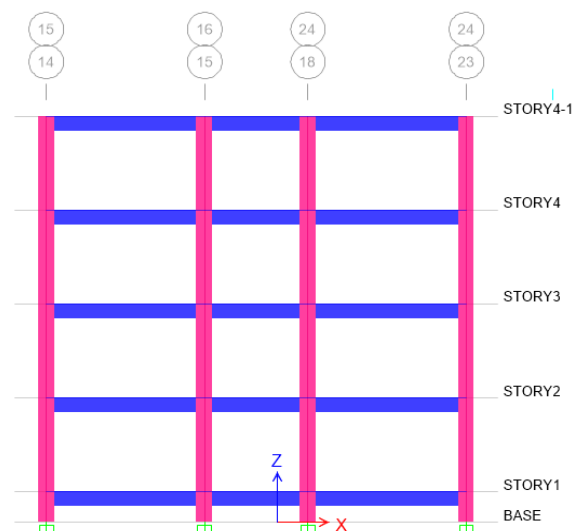


Fig -6 Elevation of model

Step - 4: Defining of loads

The loads in ETABS are defined as using static load cases command in define menu. In the Load Cases dead load of wall (230mm thick) 13.8kN/m, live load of 4 kN/m² and floor finish of 1.5kN/m² was defined.

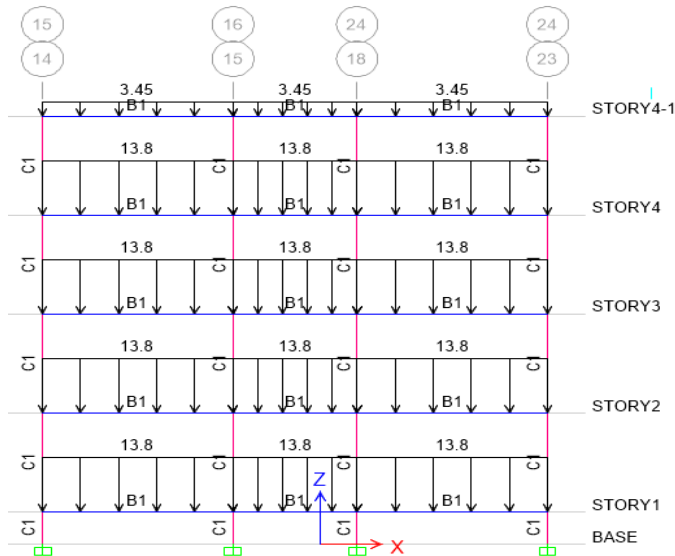


Fig-7 Brick wall load on beam.

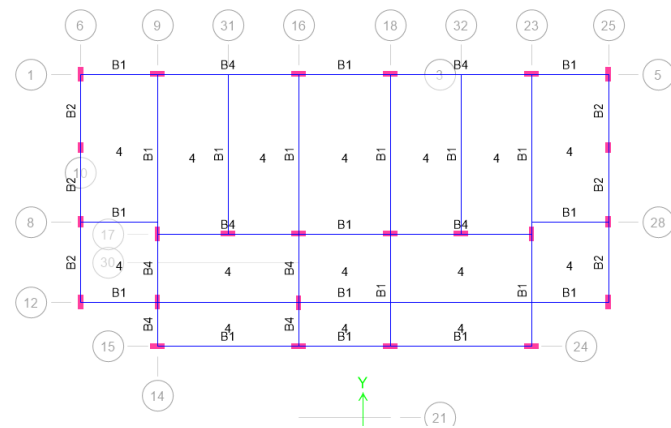


Fig - 8 Slab load on slab

Step - 5: Analysis

After the completion of all the above steps analysis was performed and checked for errors.

Step - 6: Design

Once after the analysis is over design of the structural elements was done as per IS 456:2000 guidelines.

For this go to Design menu >concrete design >select design combo. After this again go to design menu >concrete frame

design >start design \ check of structure then ETABS performs the design for every structural element.

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: Soil Survey Report

Density	13.2kN/m ³
Water content	13.78%
Cohesion, C	9kN/m ²
Angle of shearing resistance	20 ⁰
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.

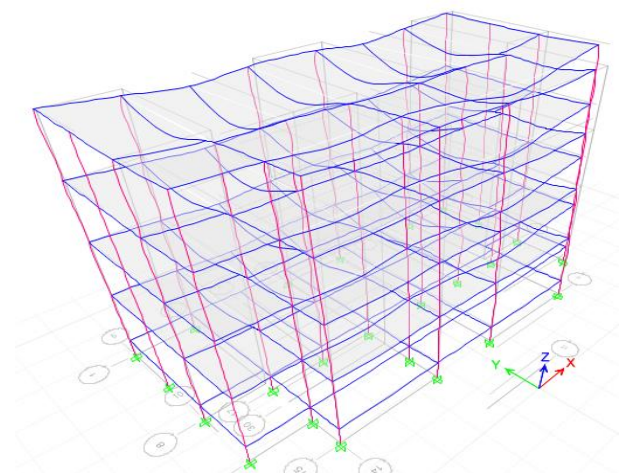


Fig - 9 3D View after analysis.

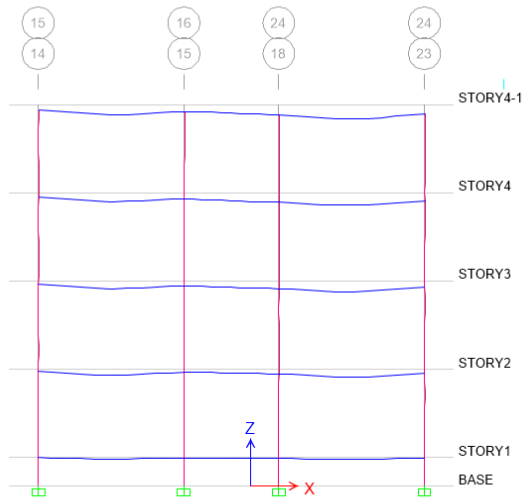


Fig-10 Elevation after analysis.

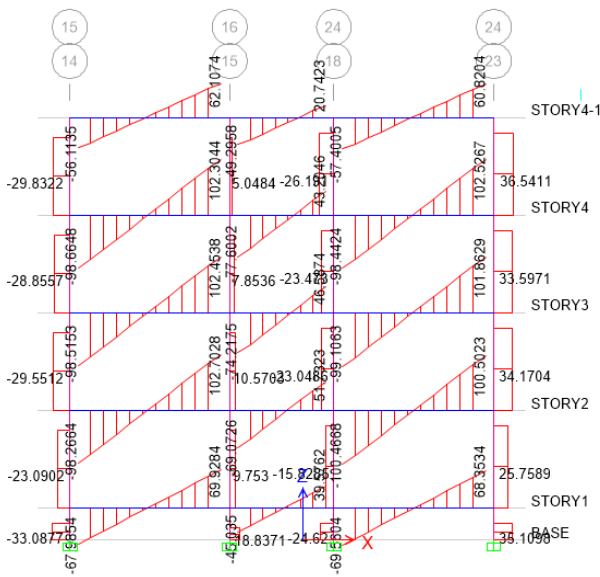


Fig-11 Shear force diagram.

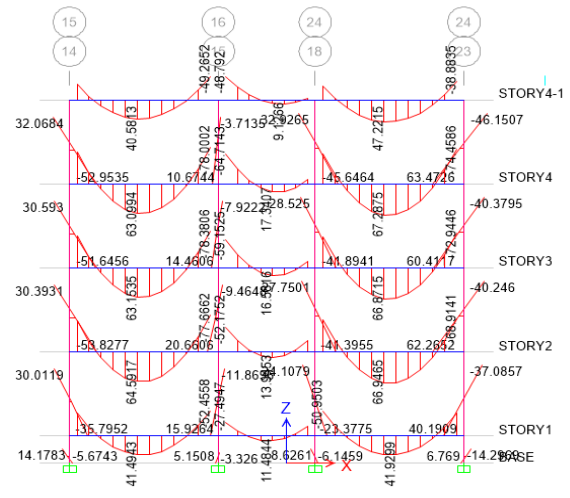


Fig- 12 Bending moment diagram.

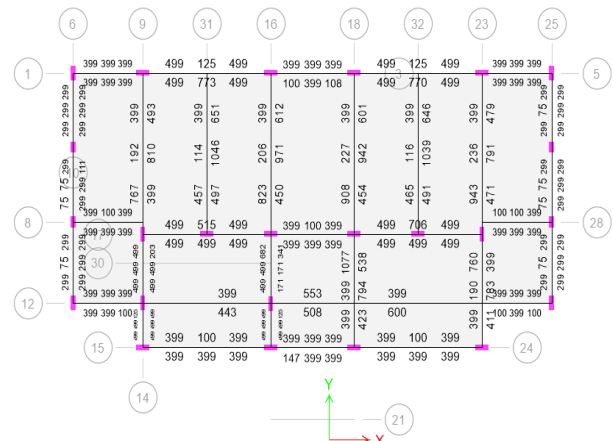


Fig-13 Design values of Beams.

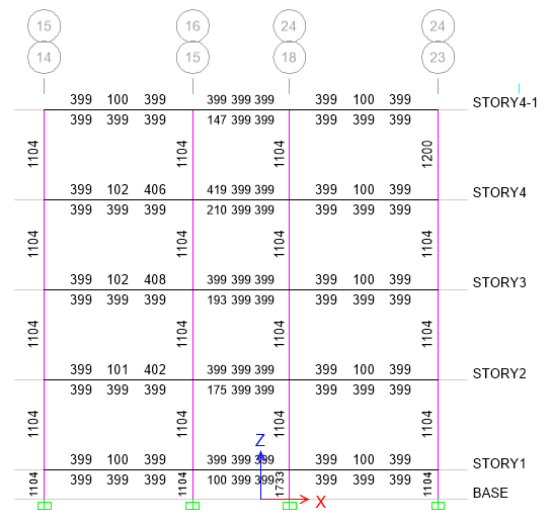


Fig-14 Design values of column.

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

5. CONCLUSION

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

1. Analysis was done by using ETABS software and successfully verified manually as per IS456.
2. Calculation by both manual work as well as software analysis gives almost same result.
3. Further the work is extended for a 4 storey building and found that the results are matching.
4. As the 4-storey building has similar floors ETABS is the perfect software which can be adopted for analysis and design.
5. 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".