EasyChair Preprint
№ 5921

Analysis And Design Of Structure Residential Building G+5 Using Etabs

V. Lakshmi Shireen Banu, Sheikh Mohd Salar, Mohd Ghouse,
Sohel Jawad, Md Afak and Shaik Mohd Sadiq

June 27, 2021

Analysis And Design Of Structure Residential Building G+5 Using Etabs

V.L.S Banu[*1], Second AMD Shafiuddin Siddiqui[2], Sheikh Mohd Salar[3], MA Ghouse[4], Sohel Jawad[5], Md Afak[6], Shaik Mohd Sadeq[7]

[*1]Civil Engineering, Lords Institute Of Engineering And Technology, Hyderabad, Telangana, India
Example@ijsrce.com[1]

[2] Civil Engineering, Lords Institute Of Engineering And Technology, Hyderabad, Telangana, India
Example@ijsrce.com[2]

[3] Civil Engineering, Lords Institute Of Engineering And Technology, Hyderabad, Telangana, India
Example@ijsrce.com[3]

[4] Civil Engineering, Lords Institute Of Engineering And Technology, Hyderabad, Telangana, India
Example@ijsrce.com[3]

[5]Civil Engineering, Lords Institute Of Engineering And Technology, Hyderabad, Telangana, India
Example@ijsrce.com[3]

ABSTRACT

We are living in the 21st Century number of complex and irregular structure and Designed to resists the earthquake, Wind and needs to analyze, design the structure by the various software like ETABS, STAAD.Pro, TEKLA and to design the structure in this project we used the ETABS software due to company suggestion and to find stress analysis in a slab, the shear force for the beam, and area reinforcement for the Column and design the foundation depends upon the reaction a height of the Foundation level depends upon the site and safe bearing capacity of the soil due to Stability purpose-designed the retaining wall in this project.

INTRODUCTION

Structural are design to resist earthquake, wind load and stable the structure and the damage in the structure causes loss of peoples and the high raise buildings and to check the strength Stiffness and resists the displacement of the building by proper designs and detail ductile of the building and can design the proper gravity loads and depend on the design of the building, the paper deals the analysis, design is done by using the software package called as E-TABS. ETABS is 3D structural software. ETABS is the abbreviation of "Extended 3D Analysis of Building System. Hence revisions are done depend upon the code provisions and the result given by the analysis. Stand always gives higher demand for steel reinforcement. Also after proving the credibility of ETABS by manual calculations. Once correctly define materials in ETABS, and the necessary steps taken in the modeling process and discussed below.

Objective

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

1. To understand the basic principles of structures by using Indian Standard Codes

2. To understand the parameters of the design for beams, columns, slabs, and other structural components.

3. To prepare the 3D model of the structure by using the E-TABS Software for detailed analysis and design.

Scope of study

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

Loads and Combination

As per the limit state design of reinforced concrete structures and pre-stressed concrete structures, the following load combination has been taken:
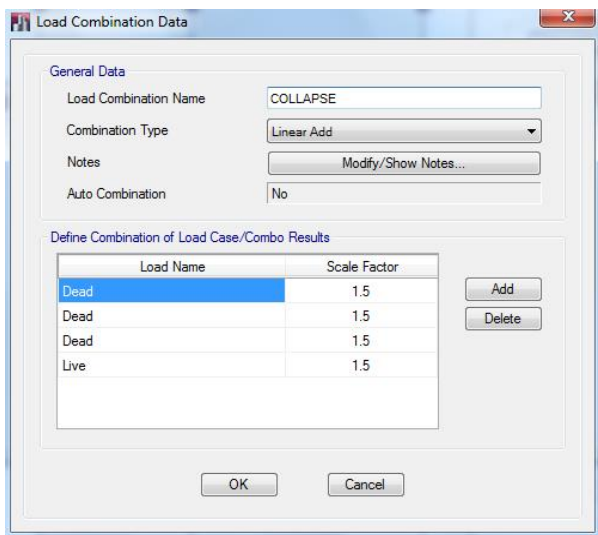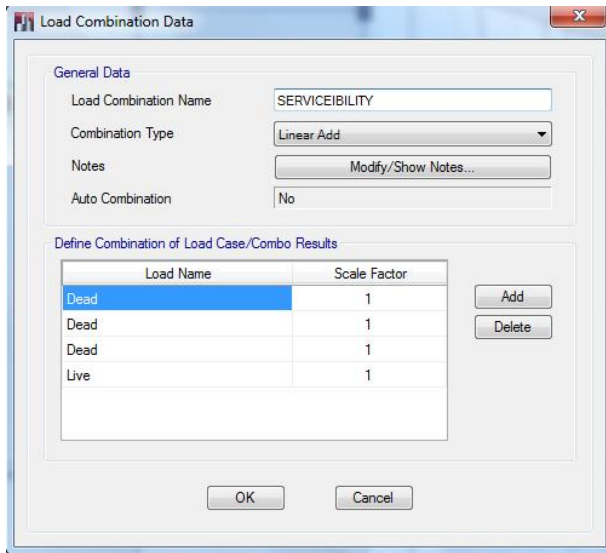

Components of Structure and Size

MEMBER		DIMENSIONS (mm)
SLAB	S1	150 (Thickness)
COLUMN	C1	450 x 230
	C2	700 x 230
	C3	500 x 230
	C4	900 x 250
	B1	450 x 230
BEAM	B2	420 x 230

Loads and calculations

Sl. No.	Types of Load Applied on Structure	Beams	Slabs
1.	Dead Load	2kN/m	-
2.	Live Load	-	2kN/m
3.	Wall Load 230mm	12KN/m	-
4.	Wall load 115mm	6KN/m	-
5.	Floor Finish	-	1.5kN/m
6.	Staircase Dead Load	25KN/m	-

Centre Line of the Structure as per ETABS:

2
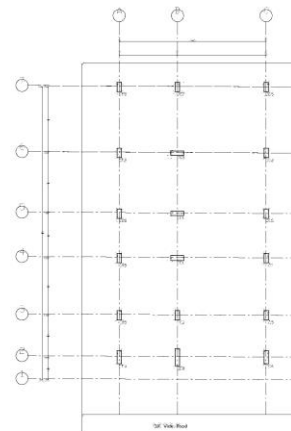
Methodology / Procedure :

Step - 1: Initial setup of Standard Codes and Country codes

Step - 2: Creation of Grid points & Generation of structure

After getting opened with 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. We add new material for our structural components (beams, columns, slabs) by giving the specified details in defining. After that, we define section size by selecting frame sections as shown below & added the required section for beams, columns, etc.

Step - 4: Assigning of Property

After defining the property we draw the structural components using the command menu. Draw a line for the beam for beams and create columns in the 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 (supports) fixed.

Step - 6: Assigning of Dead loads

After defining all the loads. Dead loads are assigned for external walls, internal walls by taken as mention above table.

Step - 7: Assigning of Live loads

Live loads are assigned for the entire structure including floor finishing.

Step - 8: 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 G+5 Residential Building having a total height of fewer than 17 meters there is no need of assigning wind loads or earthquake loads.

Step - 9: Assigning 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+5 Residential Building having a total height of less than 17 meters there is no need of assigning Seismic loads.
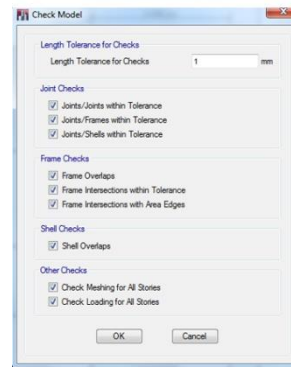
Step - 10: Assigning of load combinations

Using load combinations command in define menu 1.5 times of dead load and live load will be taken as mentioned above.
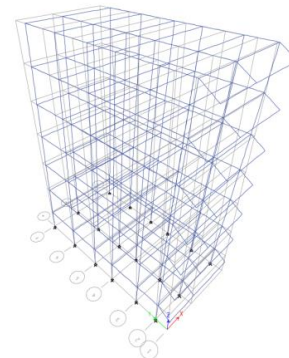
Step - 11: Analysis

After the completion of all the above steps, we have performed the analysis and checked for errors.
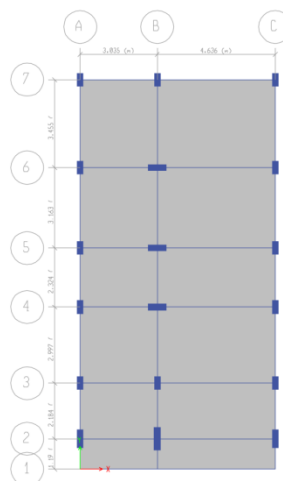

Step - 12: Design

The design of after completion of analysis of structure deform the shape of a model

Step - 12: Design

After the completion of the analysis, we have performed a concrete design on the structure as per IS 456:2000. ETABS performs the design for every structural element.

By using the centerline diagram the grid system is given by X and Y coordinates and the spacing method of the grid system is adopted for the convenience of the user.


3

Results of Analysis

Shear Force for the analysis part

Shear and **bending moment** diagrams are analytical tools **used** in conjunction with structural analysis to help to perform structural **design** by determining the value of **shear force** and **bending moment** at a given point of a structural element such as a beam.
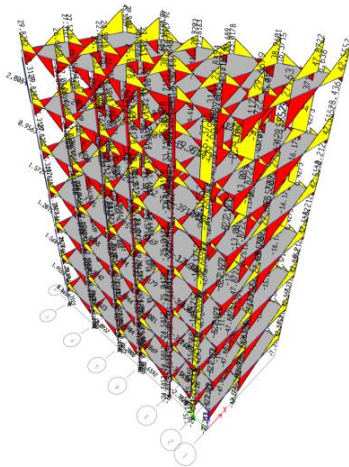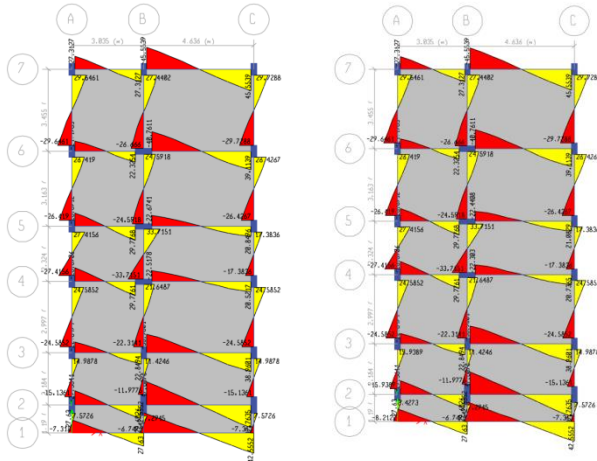
The bending moment diagram indicates the bending moment resisted by the beam.
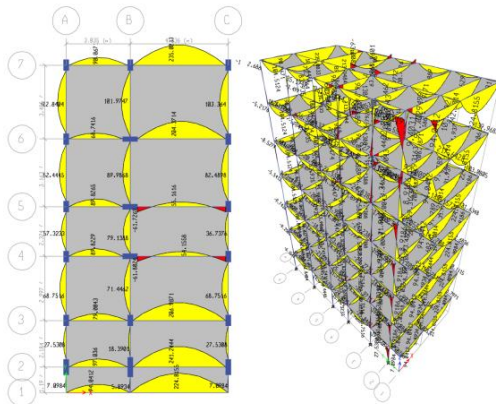
For the area of longitudinal reinforcement, go to the design menu in ETABS, concrete frame designand start design check.
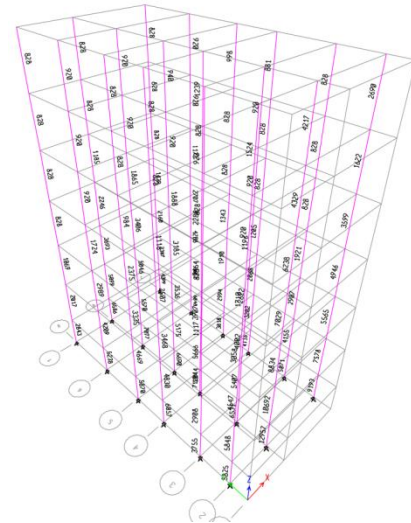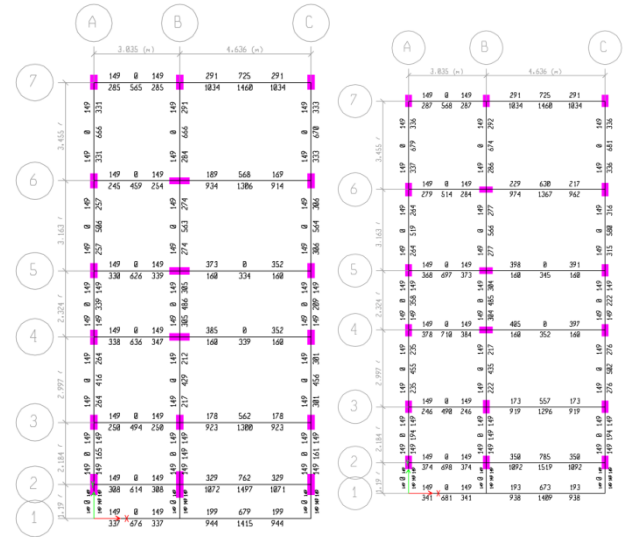
CONCLUSION

`   The structure is a design based on the E-TABS, and the theory of LIMIT STATE METHOD which provide adequate strength, serviceability, and durability besides the economy. The displacement, shear force, bending moment variation has been shown. If any beam fails, the dimensions of the beam and column should be changed and reinforcement detailing can be produced.

4

Reference

[1] Structural Design of concrete structure using E-Tabs, Shivam Asawa, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) Volume 14, Issue 1 ver. 4 (Jan – Feb 2017), PP 49-51

[2] Design and analysis of multi-storeyed building under static and dynamic loading conditions by using E-TABS by Balaji and Selvarasan in International Journal of Technical Research and Applications, Volume 4 , Issue 4(July-Aug, 2016), PP.1-5

[3] Effect of base isolation in a multistoried reinforced concrete building by M. Rajesh Reddy, Dr.N. Srujana, N.Lingeshwaran, IJCIET International Journal of civil engineering & technology, Volume 8, Issue 3, March 2017, PP. 878-887

[4] Mahesh N. Patil, Yogesh N. Sonawane, "Seismic Analysis of Multi-storied Building", International Journal of Engineering and Innovative Technology, ISSN: 2277-3754, Volume 4, Issue 9, March 2015.

[5] Piyush Tiwari, P.J.Salunke, "Earthquake Resistant Design of Open Ground Storey Building", International Research Journal of Engineering and Technology, ISSN: 2395 - 0056, Volume: 02 Issue: 07-Oct-2015.

[6] Pardeshi Sameer, "Study of seismic analysis and design of the multi-story symmetrical and asymmetrical building", International Research Journal of Engineering and Technology, ISSN: 2395 -0056, Volume: 03 Issue: 01-Jan-2016.

[7] IS: 1893:2000, Part 1, "Criteria for Earthquake Resistant Design of Structures - General Provisions for Buildings", Bureau of Indian Standards, New Delhi, 2002.