International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 06 Issue: 05 | May 2019 www.irjet.net p-ISSN: 2395-0072

ANALYSIS AND DESIGN OF MULTISTOREY BUILDING (G+3) BY USING ETABS SOFTWARE

Nagaratna. S.A[1], Ranjita. N.H[2], Asif. R.S[3], Vijaykumar. K.J[4], Sudha. P.H[5], Prof. D.S. Maganur[6]

[1,2,3,4]U.G. Student, Dept of Civil Enginneering SJT Institute of Technology Ranebennur, KA, India.
[5]Assistant Professor, Dept of Civil Engineering STJ Institute of technology Ranebennur, KA, India.
[6]Proffesor, H O D, Dept of Civil Engineering STJ Institute of Technology Ranebennur, KA, India.

Abstract - Etabs stands for extended three-dimensional analysis of building systems. The main purpose of this software is to design multistoried building in a systematic process. The effective design and construction of an earthquake resistant structure have great importance al over the world. This project presents multi-storied residential building analyzed and designed with lateral loading effect of earthquake using ETABS. This project is designed as per INDIAN CODES-IS 1893-PART 2:2002, IS 456-2000.

Every structural engineer should design a building with most efficient planning and also be economical. They should ensure that it is serviceable, habitable in healthy environmental for its occupants and have longer design period. structurally robust and aesthetically pleasing building are beginning constructed by combining the best properties of any construction material and at the same time meeting a specific requirement like type of building and its loads, soil condition, time, flexibility and economy. In the view of above, the high-rise buildings are best suited solution. This paper discusses the analysis of a commercial building (G+3 hospital building). Shear force and bending moments of beams and columns are observed and concluded that large span having more shear forces and bending moment.

1.0 INTRODUCTION

Project on structural analysis and design of multi-storey RCC building focuses on the structural analysis of multi-storey building using appropriate methods of structural analysis and software (E-TABS).

In this project we have taken an architectural plan of the building on the basis of which the analysis will be done for the whole structure. For analysing a multi-storied building, one has to consider all the possible loadings and see that the structure is safe against all possible loading conditions.

Analysis of beams & columns has been done by using E-TABS software and slabs & footings are designed using “LIMIT STATE METHOD” according to IS: 456-2000. Materials used are of M-25 concrete and Fe-415 steel.

The building is designed as a framed structure with brick infill walls. Keeping in view the requirement & utility of the building the dead load, live load & super imposed loads have been considered for the analysis & design of the structures in accordance with the specification of IS: 456-2000 and IS:875-1987 (Part 1 & Part2).

Subsequently, the beam column layout was prepared with the help of which slabs were identified as One Way or Two-Way Slabs. The slabs were designated names in a series as S1 &S2. The slabs were designed as per the moments obtained using the Bending Moment Co-efficient as per Annex D of IS 456: 2000.

The present project deals with the analysis of a multi-storied residential building. The dead load and live loads are applied and the design for beams, columns, footing and slabs is obtained. For design calculation MS Excel has been used.

1.1 Stages in structural design

1. Drawing study
2. Load combinations
3. Analysis of structures
4. Structural design

2.0 Objectives

The structure should be able to carry all expected loads without failure. Carry out a complete analysis and design of the main structural elements of a multi-stored building including slabs, columns, beams etc. Getting familiar with structural software's (E-TABS, AUTOCAD) The objectives include structural analysis of multi-stored RCC building using software (ETABS) prior to that manual calculation will be done by appropriate structural analysis methods. Design of beam, column, footing and slab are done. Later calculations are done using MS-Excel.

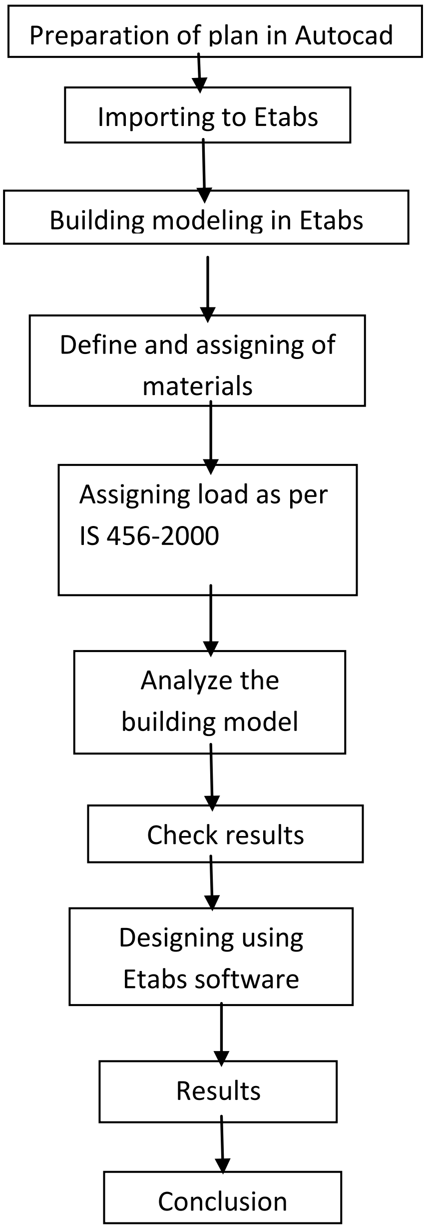
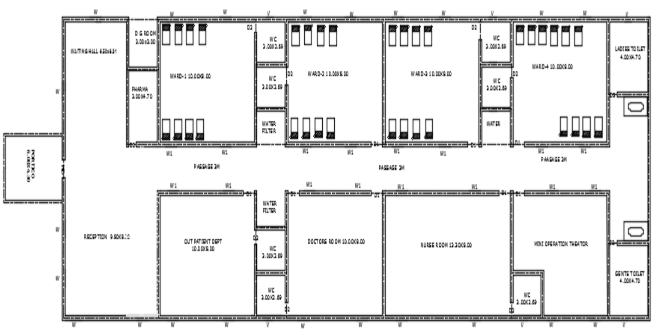
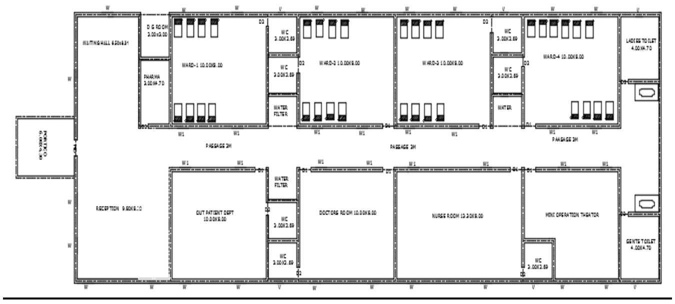


Fig -1 METHODOLOGY

3.0 Plan of Hospital Building



GROUND FLOOR PLAN



FIRST, SECOND & THIRD FLOOR PLAN

Table -1: SILENT FEATRES OF PROPOSED BUIDING

Sl.no	Description	Features
1	TYPE OF BUIDLING	G+3
2	STRUCTURE	R.C.C FRAME
3	BUILT UP AREA	1262.5m^2
4	HEIGHT BETWEEN THE FLOOR	3.0m
5	DEPT OF FOUNDATION BELOW GROUND LEVEL	600mm
6	THICKNESS OF WALL	300mm

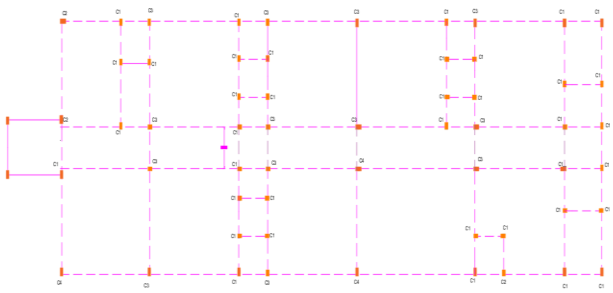


Fig -2(a): BEAM LAYOUT

BEAM SIZE

B1=(300x330) mm

B2=(300x800) mm

B3=(300x830) mm

B4=(300x1300) mm

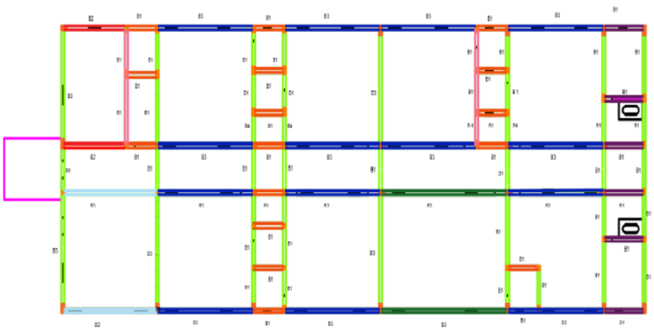


Fig -2(b): COLUMN LAYOUT

COLUMN SIZE

C1=(230x300) mm

C2=(450x450) mm

C3=(450x600) mm

C4=(600x600) mm

4.0 Analysis Methods

Etab is the primer FEM analysis and design tool for any type of project including towers, culverts, plants, bridges, stadium and marine structures. With an array of advanced analysis capabilities including linear static, response spectra, time history, cable and push over and non-linear analysis, Etab provides good compatibility with a scalable solution that will meet the demands of project every time .

5.0 Steps in Modelling

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

Display units: KN-m.

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

File → New model → Custom Grid Spacing → Edit Gird

After getting opened with ETABS we select a new model and a window appears where we had entered the grid dimensions and storey dimensions of our building.

Step - 3: Defining of Material property.
Define → Material Properties → Add New Material
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.

Step –4: Define Frame Sections.
Define → Frame Sections → Add Rectangular Section
After defining the property we define section size by selecting frame sections & added the required section for beams, columns etc.

Step - 5: Slab Details
Define → Wall/Slab Section → Add New Slab
We have to define the slab properties after defining frame sections.
Step - 6: Assigning of Property
After defining the property we draw the structural components using command menu. After defining the columns and beams, the columns and beams are placed on the grid lines, using various **"line object"** options under the command **"Draw".**

Step - 7: Defining of loads
Define → Static Load Cases → Add New Load
In ETABS all the load considerations are first defined and then assigned. The loads in ETABS are defined as using static load cases.

Step - 8: Assigning of Supports
Select Plan Level →Base →Select all columns
Assign →Joint Points →Restraints →Fixed Support
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 - 9: Assigning Loads
Slab loads
Select slabs → Assign →Shell area loads →Uniform

6.0 ANALUSIS RESULTS

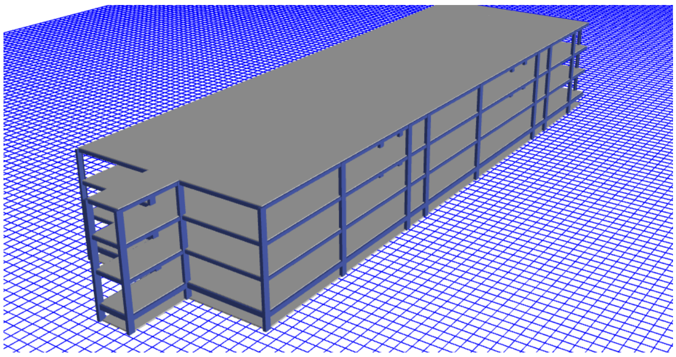


Fig -3: MODEL

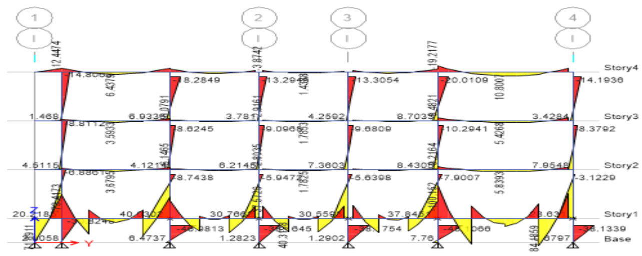


Fig -4: BENDING MOMENT DIAGRAM

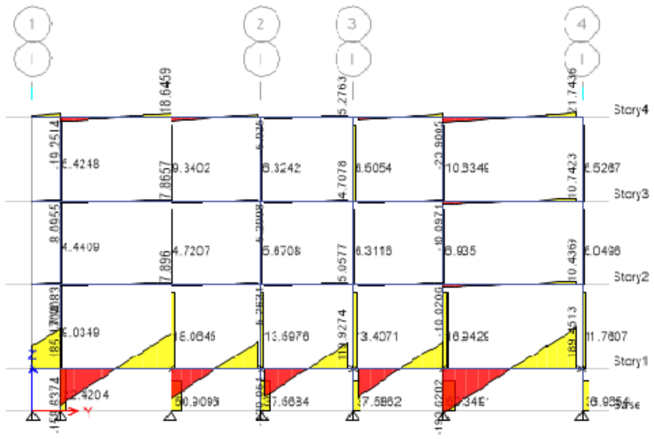


Fig -5: SHEAR FORCE DIAGRAM FROM ETAB

7.0DESIGN DETAILS

Design of slabs

Shorter span Lx=8.0m

Longer span Ly=6.5m

d=l/28 (for simply supported beam)

d=6500/28=232.14mm ≈ 250mm

Equating $Mux = Mu\ lim = 0.138\ fck\ b\ d^2$

$=0.138 x 20 x 250^2 x 1000$

$=172.5 x 10^6 N/mm^2$

$$Mu = 0.87\ fy \times Ast \times d \left(1 - \frac{fy \times Ast}{fck\ b\ d}\right)$$

$$172.5x10^6 = 0.87x415xAstx250 \left(1- \frac{Astx415}{20x250x1000}\right)$$

$172.5x10^6 = -7.49\ Ast^2 + 90.26x10^3 Ast$

$Ast = 2381.97mm^2$

Take $ast = 113.0mm^2$

$ast = .14x12^2/4=113.0mm^2$

$$Spacing = \frac{113.0}{2381.97} x 1000 = 47.49mm \approx 100mm$$

Shorter span Lx=10.0m

Longer span Ly=8.0m

d=l/28 (for simply supported beam)

d=8000/28=285.71mm ≈ 300mm

Equating $Mux = Mu\ lim = 0.138\ fck\ b\ d^2$

$=0.138x20x300^2x1000$

$=248.4x10^6 N/mm^2$

$$Mu = 0.87\ fy \times Ast \times d \left(1 - \frac{fy \times Ast}{fck\ b\ d}\right)$$

$$248.4x10^6 = 0.87x415xAstx300 \left(1- \frac{Astx415}{20x300x1000}\right)$$

$248.4x10^6 = -7.49\ Ast^2 + 108.31x10^3 Ast$

$Ast = 1255.75mm^2$

Take $ast = 113.0mm^2$

$ast = 3.14x12^2/4=113.0mm^2$

$$Spacing = \frac{113.0}{1255.75} x 1000 = 89.98mm \approx 100mm$$

Shorter span Lx=10.0m

Longer span Ly=3.0m

d=l/28 (for simply supported beam)

d=3000/28=107.14mm ≈ 110mm

Equating $Mux = Mu\ lim = 0.138\ fck\ b\ d^2$

$=0.138x20x110^2x1000$

$=33.396x10^6 N/mm^2$

$$Mu = 0.87\ fy \times Ast \times d \left(1 - \frac{fy \times Ast}{fck\ b\ d}\right)$$

$$33.396x10^6 = 0.87x415xAstx110 \left(1 - \frac{Astx415}{20x110x1000}\right)$$

$33.396x10^6 = -7.49\ Ast^2 + 39.715x10^3 Ast$

Ast = 2112.82mm^2

Take ast = 113.0mm^2

ast = 3.14x12^2/4=113.0mm^2

Spacing = $\frac{113.0 \times 1000}{2112.82}$ =53.48mm≈100mm

Design of footing:

Column size = 300 x 600mm

Grade of Concrete = M25

Grade of steel = Fe415

SBC of soil =160kN / mm^2

P = 597kN

M_{xx} = 19kN-m

M_{yy} = 68kN-m

Area of Footing = P/SBC

= 597/160

= 3.73 m^2.

Provide a footing of 2.4 x 2.4 m

Provide area = 4 m^2

Factored soil pressure = 597/4

Qu = 149.25kN/m^2

Hence the footing area is adequate since the soil pressure developed at the base is less than the factored bearing capacity of soil.

Factored Bending Moments:

Cantilever projection from the short side face of

The column = 0.5(2.4-0.23)

= 1.085 m

Cantilever projection from the long side face of

The column = 0.5(2.4-0.450)

= 0.975 m

Bending moment

@short side face of the column

= (1.5x149.25x1.085^2)/2

= 131kN-m

Bending moment at long side face

of the column = (1.5x149.25x0.975^2)/2

=106kN-m

Depth of footing:

(a) From BM consideration

Mu = 0.133xfckxbxd^2

d = 362mm

(b) From shear stress considerations we have the critical section for one-way shear is located at a distance d from the face of the column.

Shear force per meter width is

V_u = 149.25 (1085-d)

Assuming the shear strength of τc = 0.36 N/mm2 for M-20 grade concrete with nominal percentage of reinforcement Pt = 0.25

τ_c = Vu /bd

0.36 = 149.25(1085-d)/1000d

d_{min} = 317 mm

D = 600 mm

d_{prov} = 550mm

Reinforcement in footing:

(a)Longer direction:

M_u $=0.87xf_yxA_{st}xdx(1- f_yxAst/f_{ck}xbd)$

$131x10^6$ $=0.87x415xA_{st}x550(1-415xA_{st}/20x1000x550)$

A_{st} = 678 mm^2

Hence provide 12 mm dia bars @ 200 mm c/c.

Provide Ast = 708 mm^2

(b) Shorter direction

$106x10^6= 198577.5A_{st}-7.49\ A_{st}^2$

Ast = 544mm^2

Hence provide 12 mm dia bars @ 200 mm c/c.

Provide Ast = 565 mm^2

Check for shear stress:

One Way Shear Along X-direction:

Vu = 1.5x149.25x2(1.085-(550/1000))

= 239.54kN

τ_v = 239.54/2400x550

= 0.1814

τ_c = 0.36N/mm^2

V_{c1} = 0.36x2400x550

= 475kN

One Way Shear Along Y-direction:

Vu =1.5x149.25x2(0.975- (550/1000))

= 190.2kN

τ_v = 190.2/2400x550

= 0.144

τ_c = 0.36N/mm^2

V_{c1} = 0.360x2400x550

= 475kN.

$\tau_c > \tau_v$

Hence, safe under one-way shear.

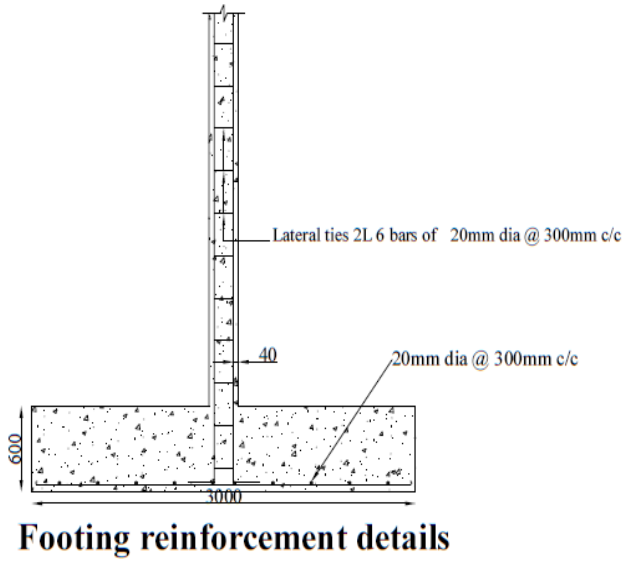


Fig -6 Footing reinforcement details

CONCLUSIONS

1. The provided member size in the Hospital building are found safe when structure is analysed using ETABS.
2. By observing results of design data we can adopt different sizes of members at different part of the structure.

REFERENCES

1. Chandrashekar and Rajasekar (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.
2. Balaji and Selvarasan (2016), "Design and Analysis of multi-storied 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. 15
3. Geethu S N, Deepthi 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".
4. IS 456-2000, Code of practice for plain and reinforced concrete
5. IS-875(PART-1): code for dead loads
6. IS-875(PART-2): code for imposed loads
7. IS-1893: code for seismic loads
8. SP 16 Design aid for reinforced concrete to IS-456.

BIOGRAPHIES

VIJAYKUMAR K J
Student
Dept of Civil Engineering
S T J I T Ranebennur

NAGARATNA S A
Student
Dept of Civil Engineering
S T J I T Ranebennur

ASIF R S
Student
Dept of Civil Engineering
S T J I T Ranebennur

RANJITA N H
Student
Dept of Civil Engineering
S T J I T Ranebennur

SUDHA P HADADI
Professor
Dept of Civil Engineering
S T J I T Ranebennur

D S MAGANUR
H O D,
Dept of Civil Engineering
S T J I T Ranebennur