

Structural Design and Analysis of Multistoried Building

Akshay K.J.¹, Mohammed Taaha¹, Prajwal B.S.¹, Ranjitha¹, Bharath H.K.², Ashish Dubay B³

B.E. Students, Department of Civil Engineering, Rajeev Institute of Technology, Hassan, India¹

M.Tech Student, Department of Civil Engineering, Rajeev Institute of Technology, Hassan, India²

Assistant Professor, Department of Civil Engineering, Rajeev Institute of Technology, Hassan, India³

ABSTRACT: Reinforced concrete (RC) is a composite material in which concrete has relatively low tensile strength and ductility which are counteracted by the inclusion of reinforcement having high tensile strength or ductility. Concrete frame structures are a very common or perhaps the most common type of modern buildings. As the name suggests this type of building consists of a frame or skeleton of concrete. Horizontal members of this frame are called beams, and the vertical members are called columns. Humans walk on flat planes of concrete called slabs. The structural design process starts once the form of the structure is selected. The structural design is the art and science of understanding the behaviour of structural members subjected to loads and designing them with economy and elegance to give safe, serviceable and durable structure. Structural planning involves position and orientation of column, positioning of beams, spanning of slabs, layout of stairs, selecting proper type of footing. Analysis of structure is used to verify the fitness of the structure for use. To perform a accurate analysis information such as structural loads, geometry, support condition and material properties should be determined. The results of such an analysis typically include support reactions, stresses and displacements. Advanced structural analysis may examine dynamic response, stability and nonlinear behaviour.

KEYWORDS: Reinforced concrete, Tensile strength, Ductility, Stability, Horizontal members, Dynamic response.

I. INTRODUCTION

The project is carried out with the help of both manual calculations and ETABS software for a new block (G+3) in the campus of RAJEEV INSTITUTE OF TECHNOLOGY, Hassan.

Analysis of structure is used to verify the fitness of the structure for use. To perform a accurate analysis information such as structural loads, geometry, support condition and material properties should be determined.

ETABS is the Acronym of EXTENDED 3D ANALYSIS OF BUILDING SYSTEMS is software developed by Computers and Structures, Inc. (CSI); a Berkeley, California based engineering software company founded in 1975. ETABS is an engineering software product that can be used to analyze and design multi-story buildings using grid-like geometry, various methods of analysis and solution techniques, considering various load combinations.

ETABS can also handle the largest and most complex building models, including a wide range of nonlinear behaviors, making it the tool of choice for structural engineers in the building industry.

ETABS can be effectively used in the analysis and design of building structures which might consists of structural members like beams, columns, slabs, shear walls etc. With ETABS you can easily apply various construction materials to your structural members like concrete, structural steel, Reinforced Concrete etc. ETABS automatically generates the self weights and the resultant gravity and lateral loads.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 5, May 2017

II. PROBLEM FORMULATION

Types of Load Used

1. Dead load (DL): Dead load defined as the load on a structure owing to it is possessed weight (self-weight). It also adds other loads if some permanent structure is added to that structure.
2. Live load (LL): Live load or impose load is distinct load on the structure due to distressing weight. The live load varies according to the type of structure. For example, usually for a Residential Building the live load taken as 3 to 5 KN/m²

Calculation of Loads

1. Main wall load: (Commencing above plinth area to below the Roof) should be the area of the cross sectional wall multiplied by unit weight of the brick. (Unit weight of is brick taken as 12 KN/m³). Usually understanding concept of the plinth load to the IS-code be supposed to be partially of the main wall load. Internal plinth load should be half of the plinth load.
2. Slab load: Arrangement of slab load+ floor finishes of slab. Slab load be able to be deliberate by breadth and width of slab multiplied by unit weight of material (according to IS-code unit weight of concrete is taken as 2.5 KN/m³).
3. Floor finish load: Floor load is a live load acting on the floor in a building, which taken as 1.5 KN/m³.
4. Live load: Live load is functional all over the burnt brick masonry structure with the exception of plinth. Usually Live load varies according to the types of structure. For inhabited, building live load is taken as -5 KN/m² on each floor. Negatives sign indicates its acting on downward direction.

Load combination

Combination of load.

- 1.5(DL + IL)

Service load.

- 1DL+1LL

III. MATERIALS PROPERTIES

Steel:

- Modulus of elasticity of steel, E = 200 Gpa
- Poisson's ratio, $\mu = 0.3$
- Steel density = 77 kN/m³(7.850 Kg/m³)
- Yield strength of steel, $f_y = 500$ Mpa
- Ultimate yield strength of steel, $f_u = 300$ Mpa

Concrete:

- Characteristic strength of concrete, $f_{ck} = 30$ Mpa
- Density of concrete = 25 kN/m³

IV. BUILDING PARAMETERS

- Length and width =45mx39m
- No of story =3
- Story height=3.5m
- Beam dimensions

BEAM	SPAN(m)	BREADTH(mm)	DEPTH(mm)	NO
Beam 1	9	300	600	30
Beam 2	4.5	300	300	52
TOTAL				104

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

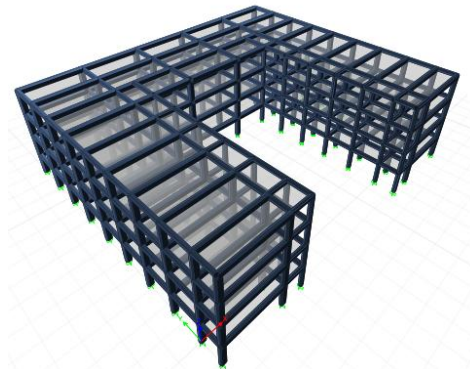
Website: www.ijirset.com

Vol. 6, Issue 5, May 2017

➤ Column dimensions

COLUMN	BREADTH(mm)	DEPTH(mm)	NO
Column 1	300	600	14
Column 2	300	300	48
TOTAL			62

- Slab thickness =150mm
- Support condition =fixed
- Beam release =axial force



Beam and column layout from ETABS

Loading Conditions:

- live load = 5 kN/m²
- Floor load=1.5kN/m²

IV. METHODOLOGY

- Creation of Grid points & Generation of structure
- Defining of property
- Assigning of Property
- Assigning of Supports
- Defining of loads
- Assigning of Dead loads
- Assigning of Live loads
- Assigning of load combinations
- Analysis
- Design

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

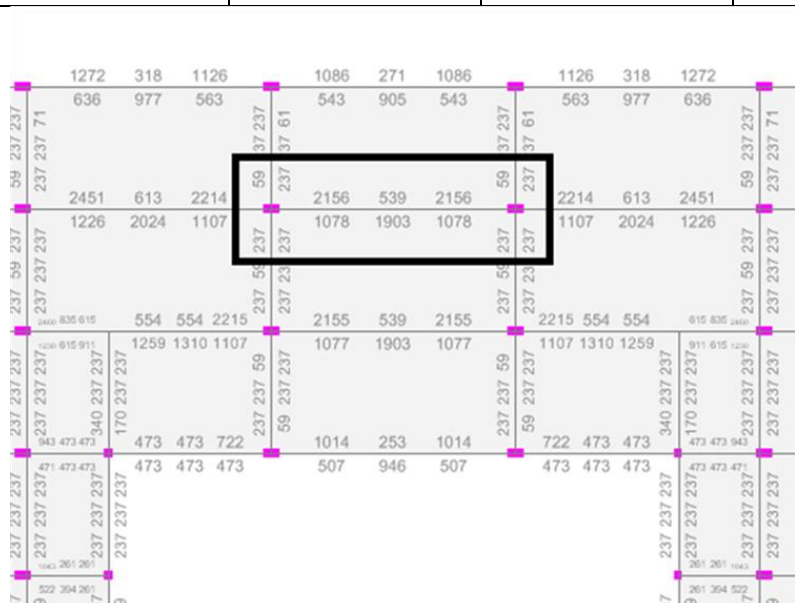
Website: www.ijirset.com

Vol. 6, Issue 5, May 2017

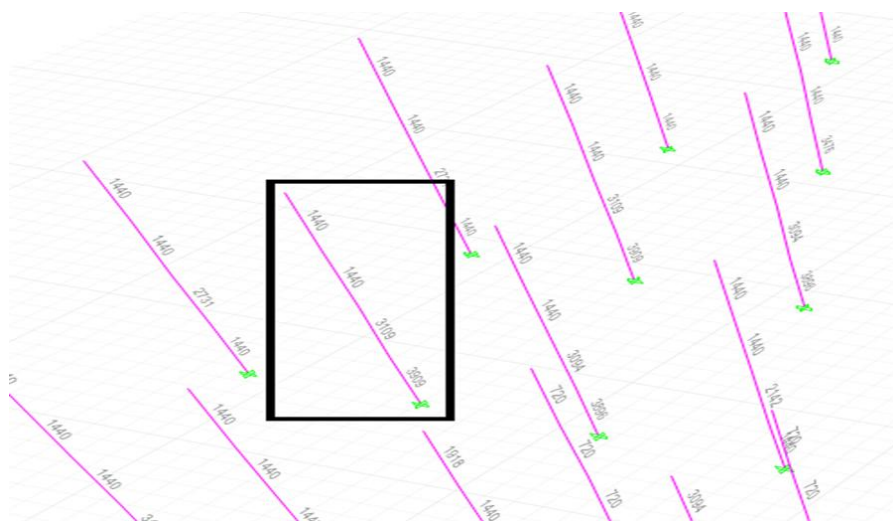
V. RESULTS AND DISCUSSIONS

Comparison of results of Ast

Particulars	ETABS results for Ast (mm ²)	Manual results for Ast (mm ²)	Percentage variation of Ast (%)
Support section of beam B80	2156	2199.11	1.96
Midspan section of beam B80	1903	1884.95	0.95
Column C42/C47	3909	3927	0.45



Area of steel of the designed beam obtained from ETABS



Area of steel of the designed column obtained from ETABS

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: www.ijirset.com

Vol. 6, Issue 5, May 2017

Comparison results of Moments

Particulars	ETABS results for Moments (kNm)	Manual results for Moments (kNm)	Percentage variation of Moments(%)
Support section of beam B80	362	337.17	6.86
Midspan section of beam B80	443	422.20	4.7
Top section of Column C42/C47	362	337.17	6.86
Bottom section of Column C42/C47	181	168.58	7.37

$$\text{Average \% difference in Ast} = (1.96+0.95+0.45)/3 \\ = 1.12 \%$$

$$\text{Average \% difference in Moments} = (6.86+4.7+6.86+7.37)/4 \\ = 6.44 \%$$

Therefore, the average % difference in Area of steel and Moments of the portal frame from ETABS software and manual calculation are obtained as 1.12 % and 6.44% respectively.

VI. CONCLUSIONS

The design and analysis of the multistoried building is carried out by ETABS software and the same is carried out for a portal frame of the structure by manual calculation. The analysis of the structure is carried out manually with the help of Kani's Method and the design is carried out by Limit State Method. The results obtained from ETABS software and manual calculation are compared and it shows the average % difference of Area of steel and moments of the portal frame as 1.12 % and 6.44 % respectively.

Hence, after comparing ETABS software with manual calculation, it can be concluded that ETABS software gives nearly accurate results and can be used for the design and analysis of multistoried structures.

REFERENCES

- [1] RCC Designs-BC Punmia
- [2] Structural Analysis-SS Bhavikatti
- [3] Structural analysis-CS Reddy
- [4] www.ascelibrary.org
- [5] www.caddcentre.com
- [6] www.wikipedia.com
- [7] www.google.com
- [8] AutoCAD
- [9] ETABS
- [10] IS 456:2000 plain and reinforced concrete
- [11] SP16 (For design of concrete structures)
- [12] IS 800:2007