IJERST

International Journal of Engineering Research and Science & Technology

www.ijerst.org

ISSN : 2319-5991

Vol. 22 No. 3 (2026)

ijerst.editor@gmail.com

editor@ijerst.com

ISSN 2319-5991 www.ijerst.com

Peer Reviewed, Referred & Indexed Journal

Research Paper

ETABS-BASED FINITE ELEMENT MODELLING AND STRUCTURAL PERFORMANCE EVALUATION OF TALL BUILDINGS

CH. NAGARAJAKUMAR[1], Mrs. P. SUSHMITHA[2], Dr. B. SHARATH CHANDRA[3]
[1]M.Tech Student, [2]Assistant Professor, [3]HOD & Assistant Professor
Department of Civil Engineering, Ellenki College of Engineering and Technology, Patelguda, Hyderabad, Telangana.

Abstract - To guarantee safety, stability, and serviceability under gravity and lateral loading situations, precise structural analysis is necessary for the ever-increasing demand for high-rise structures in fast-growing metropolitan areas. This work details the structural analysis and finite element modeling of an ETABS-based reinforced concrete skyscraper. After establishing grid systems, storey data, reinforced concrete beams, columns, slabs, and shear walls using M30 grade concrete and HYSD 415 reinforcing steel, a three-dimensional analytical model was created. The requirements of IS 456:2000, IS 875, and IS 1893 (Part 1):2016 were followed for the assignment of gravity loads, which include dead load, live load, floor finish load, and lateral loads caused by earthquakes and winds. In order to replicate realistic structural behavior during analysis, rigid diaphragm action, mass source definition, and P-Delta effects were used. Base shear, member forces, support reactions, storey displacement, and overall structural stability were all reported by the finite element study. In order to ensure that the analytical model was accurate, we compared the findings from the relevant ETABS calculations with those from hand calculations based on tributary-area load transfer for one representative column. The generated model was confirmed to be accurate as the comparison revealed close agreement with a difference of roughly 1.3%. The research proves that ETABS is a solid foundation for analyzing reinforced concrete structures with several stories, and it can help with safe, cost-effective, code-compliant structural design.

Keywords - Buildings with many stories, earthquakes, wind loads, seismic analysis, base shear, and finite element analysis are all part of structural design.

I. INTRODUCTION

The demand for towering, multi-story structures in contemporary cities has skyrocketed due to the city's rapid urbanization and persistent population expansion. Vertical building has been a popular way to accommodate residential, commercial, and institutional growth in urban areas with limited land. Tall structures experience more than just the usual stresses from gravity; they also have to contend with the significant lateral pressures caused by wind and earthquakes. The structural behavior of tall buildings is mostly dictated by these lateral activities, which affect internal forces, storey displacement, drift between floors, and overall structural stability. Hence, thorough structural study is necessary to guarantee that tall structures meet the necessary strength, serviceability, and safety standards over the duration of their design. Complex multi-story buildings render traditional manual techniques of structural analysis very wasteful, since they are only applicable to relatively basic structures. As a result, finite element analysis (FEA) is a mainstay of contemporary structural engineering. This method involves breaking down a building into its constituent parts and then solving the governing equilibrium equations computationally. With finite element modeling, massive structural systems exposed to diverse loading conditions may be quickly handled while precise displacements, stresses, support responses, and member forces can be estimated.Due to their extreme inefficiency when used to complicated multi-story buildings, traditional manual techniques of structural analysis are only applicable to relatively basic structures. Finite element analysis (FEA) is a crucial tool in modern structural engineering. It breaks a structure down into its constituent parts and uses numerical methods to solve the equilibrium equations that control it. A complex structural system exposed to different loading conditions may be easily handled with the help of finite element modeling, which accurately estimates displacements, stresses, support responses, and member forces.Accurate material attributes, acceptable modeling assumptions, boundary conditions, and load assignment are the four cornerstones of a reliable finite element model. Even while sophisticated programs may do complicated numerical calculations with great accuracy, it is still vital to validate results using separate engineering calculations. One way to ensure that the analytical model is accurate and that the numerical forecasts are reliable is to compare the results produced by the program with those obtained by hand.

This research uses ETABS to model and analyze a reinforced concrete tall skyscraper. Beams, columns, slabs, and shear walls constructed from M30 grade concrete and HYSD 415 reinforcing steel are all part of the structural model. In accordance with the requirements of IS 456:2000, IS 875, and IS 1893 (Part 1):2016, wind loads, seismic loads, and gravity loads are all calculated. To represent realistic structural behavior, the analytical model includes mass source definition, P-Delta effects, and stiff diaphragm action. A sample structural part is subjected to manual gravity load computations to further confirm the base responses and internal forces generated by ETABS. Results from both the analytical and manual approaches show that the proposed finite element model is reliable, and ETABS is a good fit for analyzing the structural integrity of tall reinforced concrete structures.

II. LITERATURE REVIEW

The structural behavior of tall structures under gravity and lateral loading circumstances has been the focus of increased study due to the fast expansion of urban infrastructure. The effectiveness of a building's structure is less affected by gravity loads and more affected by its ability to withstand wind and seismic forces as the building's height rises. In order to ensure the safe and cost-effective design of skyscrapers, a great deal of research has concentrated on enhancing analytical methods, structural systems, and computer tools.Conventional moment-resisting frames provide sufficient resistance to gravity loads, according to early studies on reinforced concrete framed buildings. However, these frames may suffer from excessive lateral displacement and inter-storey drift when subjected to seismic excitation. Shear walls, core walls, and braced frames made of reinforced concrete were created as structural methods to circumvent these restrictions. The total seismic performance, structural rigidity, and reduction of horizontal displacement are all greatly improved by these lateral load-resisting devices.According to several studies, the behavior of tall structures is significantly affected by the location and arrangement of shear walls. Shear walls that are symmetrically positioned and situated in the middle of a structure help to minimize torsional impacts and provide consistent lateral stiffness all around. Contemporary reinforced concrete high-rise construction, especially in areas with moderate to high seismic activity, often incorporates shear walls.

When it comes to computer analysis of complicated architectural structures, Finite Element Analysis (FEA) is now the gold standard. By breaking the structure down into several linked parts, FEA is able to accurately forecast stresses, internal forces, support responses, and structural displacements under various loading circumstances, in contrast to traditional analytical approaches. Computer technology's never-ending improvement has made finite element-based structural analysis for multi-story structures even more efficient and reliable. When it comes to structural engineering software, ETABS is among the most popular choices for analyzing and designing buildings. 3D modeling, FEM, load generating, design verification, and structural details are all part of the software's integrated platform. Researchers have shown that ETABS successfully simulates steel and reinforced concrete structures according to a number of international design standards for things like gravity loads, wind loads, earthquake forces, storey displacements, member forces, and support responses.Research comparing ETABS with STAAD.According to Pro and SAP2000, ETABS's storey-based modeling environment, automated load generation, diaphragm assignment, and effective handling of seismic load combinations make it a great choice for building-oriented applications. Evaluating the behavior of tall structures exposed to complicated loading circumstances is made possible by the software's support for sophisticated analytical processes such response spectrum analysis, nonlinear analysis, and P-Delta analysis.

The significance of precisely characterizing seismic mass, structural member sections, and material qualities has been highlighted in earlier research as well. The analytical model's stiffness, natural period, and load-carrying capacity are directly impacted by the accuracy with which the concrete and reinforcing parameters are represented. Similarly, for accurate estimate of seismic forces and dynamic properties, mass sources, including self-weight and relevant sections of applied loads, must be defined appropriately. In reinforced concrete structures with regular floor systems, researchers have also shown that rigid diaphragm modeling gives a realistic distribution of lateral loads among the vertical structural parts.An essential part of structural engineering is still manually calculating the validation of analytical models. Multiple investigations have contrasted the gravity load transfer computed separately utilizing tributary-area techniques with the support responses and member forces produced by ETABS. The studies proved that finite element modeling is reliable with the right assumptions and codal provisions applied, as the analytical and manual findings agreed closely in every case.According to the research, towering reinforced concrete structures may be effectively and accurately analyzed using finite element analysis using ETABS. Nevertheless, correct structural modeling, realistic load assignment, suitable boundary conditions, and validation of analytical outputs are crucial for the findings to be accurate. Based on these well-established principles, this study constructs a comprehensive finite element model of a reinforced concrete skyscraper, analyses its structural behavior under gravity and lateral loading, and verifies the calculated reactions with separate manual calculations to prove the model's reliability.

III. METHODOLOGY

A. Apps Employed

Extended Three-Dimensional Analysis of Building Systems, or ETABS, is a piece of software specifically made for analyzing and designing building structures, and it was used to do the structural modeling and finite element analysis. Code-based structural design, finite element analysis, load assignment, and three-dimensional modeling are all part of ETABS's integrated environment. The program has sophisticated analytical capabilities such response spectrum analysis, nonlinear analysis, and P-Delta analysis, and it supports reinforced concrete, steel, and composite structures. To make sure it would work with the Indian Standard codes that were used for building construction, this research used ETABS with the metric SI unit system.

B. Analyzing Structures

By outlining the building's grid system and story arrangement in accordance with the desired layout, a three-dimensional model of the reinforced concrete structure was created. All structural members were given material qualities that match to HYSD 415 reinforcing steel and M30 grade concrete. The frame parts were used to depict the reinforced concrete beams and columns, while the shell elements were used to represent the slabs and shear walls. As a means of simulating a solid foundation, the building's base was designed with fixed supports. Upon completion of the structural model, a continuous load path was defined, beginning with the slabs and ending at the supporting foundation, via the beams and columns.

C. Analysis and Assignment of Loads

According to IS 875 (Parts 1 and 2), gravity loads, which include self-weight, floor finish load, and imposed live load, were allocated. Seismic lateral loads were determined in accordance with IS 1893 (Part 1):2016, whereas wind lateral loads were defined in accordance with IS 875 (Part 3). To provide a realistic distribution of lateral loads among the vertical structural parts, rigid diaphragm action was applied to each floor level. To determine the seismic mass source, we used structural dead load and the required proportion of live load; to account for second-order effects, we used P-Delta analysis. Prior to conducting the finite element analysis, suitable load combinations were constructed in accordance with the idea of limit state design.

D. Use of Finite Element Methods

Before analysis, the analytical model was checked for connectivity and modeling inconsistencies using the model checking tools in ETABS. In order to find the nodal displacements, support responses, member forces, bending moments, shear forces, and axial loads for all the loading situations that were provided, the program solved the governing equilibrium equations and created the global stiffness matrix. In order to assess how the building handled combined gravity and lateral stress, structural responses were extracted, including foundation shear, storey displacement, and inter-storey drift.

E. Achieving Reliable Analytical Findings

A sample ground-floor column was manually calculated using the tributary-area approach to ensure the correctness of the finite element model. Separate calculations were made to determine the gravity loads that were transported from slabs to beams and finally to columns. These calculations were then compared with the relevant ETABS result. The analytical result and the manually estimated column axial load were quite similar, with a discrepancy of around 1.3%; this proved that the study's modeling assumptions were valid and that the finite element model was reliable.

IV. RESULTS AND DISCUSSION

A. Summary of Structural Analysis

The ETABS model and analysis of the reinforced concrete skyscraper under the combined effects of wind, earthquake, and gravity were both successful. In order to replicate the actual behavior of the structure, the finite element model included shear walls, slabs, columns, and beams as well as seismic mass definition, P-Delta effects, and stiff diaphragm action. After making sure there were no geometric inconsistencies or connection issues, the model verification procedure allowed the analysis to be successfully finished. The research yielded a thorough evaluation of the building's performance in terms of structural responses, which included base shear, support reactions, member forces, storey displacements, and axial forces.

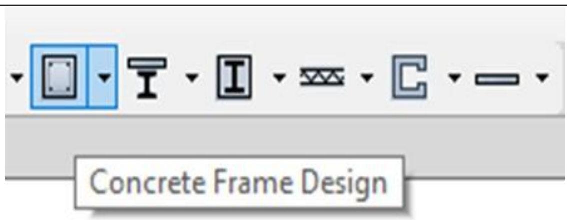

Fig. 2: A synopsis of the findings

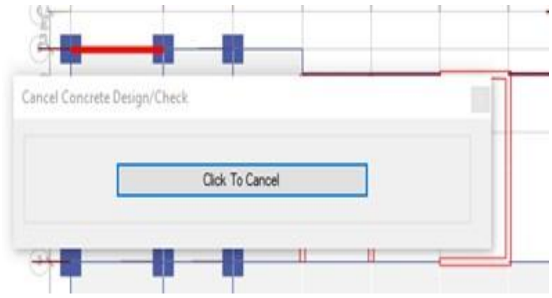

Fig. 2: Conclusions from the design review

B. Stability under Zero Gravity and Lateral Forces

In accordance with the applicable Indian Standard standards, the structural responses to the given combinations of dead load, live load, wind load, and earthquake load were within permissible limits. The lateral rigidity of the building was greatly enhanced by the addition of reinforced concrete shear walls, which controlled inter-level drift and reduced storey displacement. According to their respective stiffness, the vertical load-resisting parts were effectively distributed lateral loads by virtue of the rigid diaphragm assumption. Results were even more reliable once P-Delta analysis was used to account for second-order effects caused by gravity stresses acting via lateralized structural elements.

C. Proof of Concept for ETABS

Manual calculations of gravity load transmission for a typical ground-floor column were used to verify the reliability of the finite element model. The slab loads that were transported to the supporting beams and, eventually, to the chosen column were determined using the tributary-area approach. Both the ETABS result of 1309.87 kN and the manually estimated axial load of 1293.15 kN were compared. The analytical and manual methods were in perfect harmony, with just a 1.3% discrepancy between the two values. This strong association verifies that the study's structural modeling, load assignment, and analytic methods were accurate.

Table 1. Analyzing the Outcomes of ETABS and Manual Calculation

Parameter	Manual Calculation	ETABS Result	Difference
Column Axial Load (kN)	1293.15	1309.87	1.3%

D. Subject under consideration

When it comes to finite element analysis of reinforced concrete skyscrapers, the findings show that ETABS is a fantastic platform. A realistic prediction of the structural response under combined gravity and lateral stress was achieved by the correct specification of material characteristics, member sections, loading circumstances, diaphragm limitations, and boundary conditions. Confidence in the software-generated findings is bolstered and the produced model's suitability for real structural engineering applications is confirmed by the successful validation of the analytical model by independent manual calculations. In order to create building designs that are safe, dependable, and compatible with codes, the research stresses the need of integrating cutting-edge computational tools with engineering judgment.

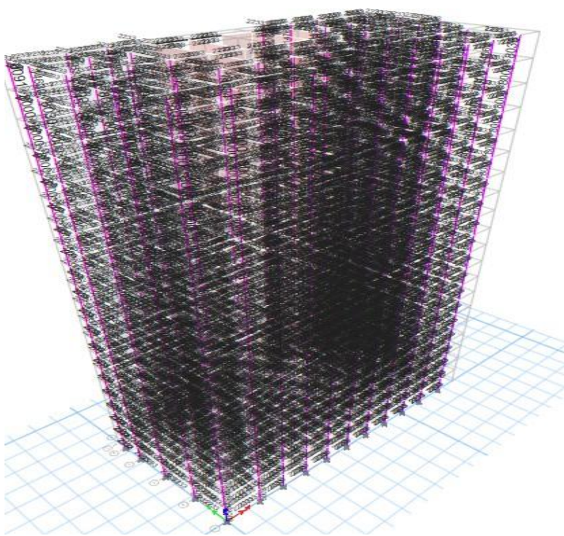

Fig. 3: The towering building's 3D model was examined in its entirety.

V. CONCLUSION

This research proved that ETABS may be used with finite element analysis to assess the structural behavior of a reinforced concrete skyscraper. The structure was studied under wind, earthquake, and gravity loading conditions according to the applicable Indian Standard codes using a complete three-dimensional analytical model that included shear walls, beams, columns, and slabs. With the addition of seismic mass definition, P-Delta effects, and stiff diaphragm action, the reaction of the structure under combined stress may be realistically simulated. Significant structural responses, such as base shear, support reactions, member forces, storey displacement, and column axial loads, were successfully produced by the finite element analysis. A comparison between the ETABS result and the manually estimated column load revealed a tight agreement of roughly 1.3%; this revealed that the analytical model had been verified by separate human calculations based on the tributary-area approach. This verifies that the chosen modeling process was accurate and that the created finite element model is reliable. The research proves that ETABS is a solid program for analyzing reinforced concrete structures with several stories. Accurate assessment of structural performance may be achieved with less effort and time spent compared to traditional manual analysis because to its integrated modeling environment, powerful finite element capabilities, and adherence to Indian Standard design standards. The analytical technique that has been established may be used as a practical foundation for designing tall reinforced concrete structures in a way that is safe, inexpensive, and compatible with codes.

REFERENCES

[1] Bureau of Indian Standards, IS 456:2000 – Plain and Reinforced Concrete – Code of Practice, New Delhi, India, 2000.

[2] Bureau of Indian Standards, IS 875 (Part 1):1987 – Code of Practice for Design Loads (Other than Earthquake) for Buildings and Structures: Dead Loads, New Delhi, India, 1987.

[3] Bureau of Indian Standards, IS 875 (Part 2):1987 – Code of Practice for Design Loads (Other than Earthquake) for Buildings and Structures: Imposed Loads, New Delhi, India, 1987.

[4] Bureau of Indian Standards, IS 875 (Part 3):2015 – Code of Practice for Design Loads (Other than Earthquake) for Buildings and Structures: Wind Loads, New Delhi, India, 2015.

[5] Bureau of Indian Standards, IS 1893 (Part 1):2016 – Criteria for Earthquake Resistant Design of Structures, New Delhi, India, 2016.

[6] Bureau of Indian Standards, IS 13920:2016 – Ductile Design and Detailing of Reinforced Concrete Structures Subjected to Seismic Forces, New Delhi, India, 2016.

[7] Computers and Structures Inc. (CSI), ETABS – Integrated Building Design Software User's Guide, Berkeley, California, USA.

[8] P. Agrawal and M. Shrikhande, Earthquake Resistant Design of Structures, PHI Learning Pvt. Ltd., New Delhi, India.

[9] A. K. Chopra, Dynamics of Structures: Theory and Applications to Earthquake Engineering, 5th ed., Pearson Education.

[10] S. K. Duggal, Earthquake Resistant Design of Structures, Oxford University Press, New Delhi, India.

[11] E. Wilson, Three-Dimensional Static and Dynamic Analysis of Structures, Computers and Structures Inc., Berkeley, California.

[12] R. C. Hibbeler, Structural Analysis, 10th ed., Pearson Education.