A REINFORCED CONCRETE BUILDING'S STAAD-BASED STRUCTURAL ANALYSIS, FINITE ELEMENT ANALYSIS, AND OPTIMIZATION!WORKBENCH AND ANSYS PRO

KATAKAM MANEENDRA[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.

To Cite this Article
Katakam Maneendra, Mrs. P. Sushmitha, Dr. B. Sharath Chandra, "A Reinforced Concrete Building's Staad-Based Structural Analysis, Finite Element Analysis, And Optimization!Workbench And Ansys Pro", Journal of Science Engineering Technology and Management Science, Vol. 03, Issue 07, July 2026, pp: 1116-1125, DOI: http://doi.org/10.64771/jsetms.2026.v03.i07.pp1116-1125
Submitted: 20-06-2026 *Accepted: 25-07-2026* *Published: 31-07-2026*

Abstract - To ensure that reinforced concrete (RC) structures are safe, cost-effective, and efficient, structural optimization and analysis are crucial design procedures. This study offers the results of an integrated numerical examination of a reinforced concrete framed structure with a low-rise structure utilizing ANSYS Workbench and STAAD.Pro V8i. It all started with a STAAD model of a G+2 reinforced concrete frame in three dimensions.To test how a structure reacts to gravity loads, experts use beam-column elements. In accordance with the applicable rules of the Indian Standard, dead load, live load, and factored load combinations were considered. The findings of the bending moment, shear force, axial force, displacement, and support response were used to evaluate the structural behavior. How precise the STAAD modeling is.We compared the findings produced by the program with those from simpler human calculations grounded on traditional structural analysis methods in order to confirm the accuracy of the Pro model. Departures within an acceptable range of ±12% were observed in the comparison, indicating that analytical and computational approaches were in excellent agreement. Additional analysis was conducted on the verified structural model using ANSYS Workbench's finite element method. To examine the distribution of stresses, the behavior of deformation, and the important structural areas under combined gravity and equivalent seismic loading, a mixed-element model was created. This model includes shell elements for slabs and line elements for frame members. According to ANSYS, the area where the roof slab was subjected to the most deformation was also the location where critical stress concentrations formed, close to the bases of the columns supporting the bottom floor and the areas where the slab and columns were connected. Parametric optimization was then used to decrease structural mass while still meeting the criteria for stress, drift, and stiffness by adjusting the thickness of the slab and the size of the columns. Optimal layout reduced structural mass by around 18% compared to original design while maintaining structural safety. This research shows that STAAD has supplementary benefits.Pro for realistic structural design and ANSYS Workbench for sophisticated finite element analysis and optimization of RC buildings.

Keywords - Structured Optimization, Finite Element Analysis, Seismic Analysis, Reinforced Concrete Frame, STAAD.Pro V8i, ANSYS Workbench, IS 456:2000, IS 1893:2016.

I. INTRODUCTION

Modern civil engineering design relies heavily on precise structural analysis to meet the rising demand for safe and cost-effective building constructions. Buildings of many types and uses rely on reinforced concrete (RC) frames because of their resilience, longevity, and versatility. Unfortunately, traditional analytical methods are often inadequate for correctly forecasting structural behavior due to the growing complexity of structural topologies. Thus, it is now crucial to evaluate structural response under various stress circumstances using advanced numerical methodologies and computer-based modeling tools. Software for structural analysis, like STAAD.Reinforced concrete structure modeling, analysis, and design made easy using Pro's three-dimensional frame idealization platform. Engineers may test important design parameters including bending moments, shear forces, axial forces, deflections, and support responses under different loading scenarios using the program. In practical design applications, STAAD.As a result of its automation capabilities and compliance with design codes, Pro sees heavy use in the design and analysis industries. Local stress concentrations, slab behavior, and complicated load transfer processes may not be well represented by the line-element technique used in traditional structural analysis software. A more in-depth method is offered by Finite Element Analysis (FEA), which involves computationally solving the governing equations after discretizing a structural system into smaller, linked components. One popular finite element platform, ANSYS Workbench, may employ sophisticated modeling tools to examine complicated structure behavior. The ability to describe slabs, stress distributions, deformation patterns, and local structural consequences is a key advantage of finite element modeling over traditional beam-column analysis. This is why finite element analysis is so helpful when trying to assess crucial areas and enhance structural performance by means of optimization.

Sustainable and cost-effective building approaches have increased the relevance of structural optimization in recent years. For reasons of safety, traditional design methods often choose conservative member sizes, which might lead to unnecessary material consumption. In order to systematically modify structural factors like member size and material distribution while fulfilling strength, serviceability, and stability restrictions, optimization-based approaches are used. Optimization techniques may be used in conjunction with finite element analysis to increase structural efficiency while maintaining or even increasing safety margins. This research presents an integrated technique that combines structural optimization, ANSYS Workbench finite element assessment, validation using manual calculations, and STAAD.Pro V8i analysis. To start, STAAD is used to analyze a G+2 reinforced concrete frame.Pro subjected to gravity loading conditions, and the resulting findings are confirmed by virtue of standard analytical procedures. Afterwards, ANSYS Workbench is used to simulate combined gravity and equivalent seismic loads on a two-story structural model. A parametric optimization analysis is conducted to reduce structural mass while retaining acceptable performance limitations, after which the key stress and deformation locations are identified using the finite element findings. This study aims to show how modern finite element modeling and traditional structural analysis work together to make reinforced concrete building designs more efficient and dependable. The research lays forth a methodical plan for enhancing structural performance via the integration of design-oriented analysis, thorough numerical assessment, and optimization.

II. LITERATURE REVIEW

The use of numerical modeling methods in structural engineering has greatly enhanced the precision and effectiveness of building evaluation and design. With the use of structural analysis tools, traditional analytical techniques, and finite element-based simulation methodologies, researchers have investigated the behavior of reinforced concrete structures extensively. When taking into account suitable modeling assumptions, material characteristics, and loading circumstances, prior research shows that computational structural analysis methods provide dependable predictions of structural response. Software for structural analysis, like STAAD. A large number of reinforced concrete framed structure analyses and designs have made use of Pro. Using STAAD, several researchers have examined how RC structures fare when subjected to gravity and seismic loads. Benefiting from its accuracy in forecasting displacement responses, shear forces, axial forces, and bending moments. Research on RC frames with many stories has shown that, given the right boundary conditions and loading assumptions, software-based analysis closely matches theoretical models. The capacity of STAAD...It has become a popular tool in structural engineering applications due to its ability to include design rules from standards like IS 456 and IS 875.When simple analytical models can't capture the complexities of a structure's behavior, finite element analysis (FEA) steps in as a crucial computational tool. In their groundbreaking work, Turner et al. laid the groundwork for contemporary computational mechanics by introducing the idea of structural analysis using finite element formulations based on stiffness. Engineers were able to use numerical techniques to analyze complicated geometries and loading situations when Clough advanced the finite element technique for structural applications. Seismic, thermal, dynamic, and structural analysis issues have all seen heavy use of the finite element approach since its inception.

A large number of structural components have had their stress distribution, deformation properties, and failure behavior examined using ANSYS Workbench. Composites, slabs, and frames made of reinforced concrete have been the subject of ANSYS-based simulations that represent structural components utilizing shell, solid, and beam elements. According to this research, finite element models are superior to traditional frame analysis approaches for capturing details such localized stress concentrations, load transmission processes, and deformation patterns.Because earthquake-resistant architecture is so important, seismic analysis of reinforced concrete buildings has also attracted a lot of interest. The reaction of RC structures is heavily impacted by factors including seismic zone factor, importance factor, response reduction factor, and structural stiffness, according to previous studies that were based on the requirements of IS 1893. Finite element models provide extensive assessment of stress and deformation under earthquake-induced loading, whereas comparable static and response spectrum approaches offer useful forecasts of seismic behavior, according to studies.

In recent years, finite element analysis has been more integrated with structural optimization methods in an effort to provide more cost-effective and environmentally friendly designs. To decrease structural weight while fulfilling strength and serviceability criteria, researchers have employed numerical optimization approaches such as gradient-based optimization, evolutionary algorithms, and response surface methods. Optimization studies on framed structures have shown that by adjusting the proportions of the members correctly, we may drastically cut down on material consumption without compromising structural safety.Despite its rapid analysis, STAAD.Pro mostly uses one-dimensional beam and column elements to depict structures, which isn't ideal for ordinary structural design. Detailed stress distribution inside slabs and connecting areas may not be captured by such an idealization. On the other hand, ANSYS Workbench offers more sophisticated finite element capabilities, but it demands more work and resources when it comes to modeling. Thus, a powerful technique is produced by merging the two methods in cases when STAAD is applicable.For initial design and code-based evaluation, utilize Pro; for comprehensive structural optimization and evaluation, use ANSYS. In this study, we build on prior work by integrating STAAD.Pro and ANSYS Workbench to analyze, validate, and optimize reinforced concrete framed buildings. A better knowledge of structural behavior and material efficiency may be achieved by the methodology's combination of practical design-oriented analysis and extensive finite element assessment.

III. METHODOLOGY

A traditional structural frame analysis utilizing STAAD is one of two interrelated numerical analysis methodologies used in this study's methodology.I used ANSYS Workbench for a thorough finite element study, together with Pro V8i. The purpose of this study was to verify the numerical findings, examine the likelihood of enhancing structural efficiency by means of finite element-based optimization, and assess the structural response of low-rise RCC framed structures.

A. The state.Based on Professional Structural Analysis

Using STAAD.Pro V8i (SELECTseries 5), a model of a reinforced concrete frame was created in three dimensions. Beams and columns were represented as one-dimensional frame elements linked by structural nodes in the G+2 RCC space frame arrangement. With their set support conditions given at the foundation level, the model had 87 structural members and 48 joints. The standard procedure for reinforced concrete design was followed for specifying the structural dimensions and material quality. Different properties were assigned to the concrete and reinforcement in accordance with the standards set out in IS 456:2000. Every structural loading, including self-weight and live loads caused by occupied floors, was applied in compliance with the requirements of IS 875. The following load situations were considered:

- The self-weight calculation automatically generates the dead load.
- Floor loading is used to apply live load.
- For the final load assessment, factored the sum of DL and LL, which is 1.5.

Measurements of axial force, bending moment, shear force, support responses, and nodal displacement were used to assess the structural response. The resulting STAAD grid.Through the use of simplified analytical calculations grounded on classical structural mechanics equations, the findings were further verified.

B. Methodology for Validation

So that we can make sure the STAAD is correct.Comparing the pro model with manual calculations, we found a subset of key replies. Standard fixed-end and simply supported beam equations were used to determine the beam bending moment and shear force values, while the tributary area approach was used to estimate the column axial forces. In order to determine the greatest possible discrepancy between the analytical and numerical findings, we used: %The difference between the software result and the manual result, divided by the manual result and then multiplied by 100%, is the deviation. The difference between the two is the manual result, divided by the software result, and then multiplied by 100%. The rationale for the comparison was that STAAD represents idealized isolated members, while manual computations do not.Stiffness interaction, load redistribution between structural parts, and full three-dimensional frame behavior are all taken into account by Pro.

C. Simulation in the ANSYS Workbench

The structural behavior under gravity and comparable seismic loading conditions was studied using a thorough finite element model produced in ANSYS Workbench. In ANSYS DesignModeler, the building's geometry was generated by making surface bodies out of the slabs and line bodies out of the vertical structural elements.

ISSN:3049-0952
www.jsetms.com

What made up the model were:

- Three solid objects that stand in for slabs on the floor and ceiling.
- Eight bodies that form lines, which stand for columns.
- A total dimension of around 9 meters by 6 meters for the structural plan.
- One hundred millimeters thick slab.
- Approximately 3 meters is the typical height for a story.

Shell elements were used to depict the slabs, while beam-type line elements were used to represent the columns. Standard engineering values were used to give material attributes to structural steel parts and concrete slabs.

The following material attributes were chosen (Table 1).

Property	Structural Steel	Concrete
Density	7850 kg/m^3	2400 kg/m^3
Young's Modulus	2.0 × 10□ MPa	2.5 × 10□ MPa
Poisson's Ratio	0.30	0.20
Yield Strength	250 MPa	—
Compressive Strength	—	25 MPa

D. Conditions for Loading and Boundaries

With the help of the ANSYS Static Structural solver, the finite element model was examined. As a representation of the foundation's rigidity, fixed supports were placed at the column bases. All parts of the model were subjected to gravity loading, and the roof slab was subjected to a seismic pressure load that was comparable to that of a lateral earthquake.Using a typical Zone IV condition, an importance factor of 1.2, a response reduction factor of 5, medium soil condition, and 5% damping, the seismic parameters were set in accordance with the criteria of IS 1893: 2016.

Here are the circumstances that were put into play:

- With all degrees of freedom constrained, there is fixed support at the bases of the columns.
- The acceleration due to gravity on Earth is 9806.6 mm/s^2.
- A seismic force of 0.68 MPa applied to the roof slab.

E. Extraction of Results with Finite Element Mesh

With a preference for mechanical physics, the created geometry was discretized using ANSYS Meshing. For the slabs and structural members, a program-controlled mesh was created using quadrilateral-dominant shell and beam elements, respectively. Before analysis, the mesh was quality-checked using the in-built aggressive quality assessment tool to make sure there were no unsatisfactory components.

Based on the results of the ANSYS study, the main output parameters were:

· Complete disfigurement.

- A change in direction.
- Stress that is equivalent to von Mises's.
- Distribution of primary stress.
- Reaction at key points in the structure.

F. Approach to Structural Optimization

In order to decrease structural mass while fulfilling strength and serviceability criteria, a parametric optimization analysis was conducted after obtaining the baseline finite element response.

Factors related to the design were:

- Thickness of the slab (80-120 mm).
- Width of the column (250-400 mm).
- Column depth (250-400 mm) are important considerations.

Reducing overall structural mass was established as the optimization goal: Reduce M(x) to its minimum value as much as possible.

depending on:

- The maximum equivalent stress must be less than or equal to the permissible stress.
- < 0.004h for inter-story drift according to IS 1893:2016.
- The fundamental frequency must be greater than the minimum needed value.
- Realistic dimensions of structural components.

Various design combinations were evaluated using a response-surface-based optimization technique. The optimal configuration was chosen by minimizing mass while preserving structural safety.

IV. FINDINGS AND ANALYSIS

The analyzed structures' structural responses were assessed with the help of ANSYS Workbench and STAAD.Pro V8i. The STADA.The ANSYS Workbench simulation offered comprehensive findings for stress, deformation, and optimization under combined gravity and equivalent seismic loading conditions, while the Pro analysis focused on the global frame behavior under gravity loading.

A. The stand.Analysis and Validation Results for Pro

The STAAD-analyzed three-dimensional RCC frame model?Regardless of the dead load, live load, or factored load combinations that were applied, Pro exhibited steady behavior. A load combination of 1.5(DL + LL), which represents the final design condition, was used to determine the governing structural reactions. The investigation revealed that, as a result of the buildup of gravity loads from higher levels, the ground-floor beam-column connections experienced the largest bending moment. Following the software analysis, the maximum bending moment was determined to be 28.3 kip•ft. We measured a maximum axial force of 39.2 kips in the internal column and a maximum shear force of 8.69 kips. Because their tributary load areas are bigger, inner columns experience a higher axial force than edge and corner columns. Because of this, it is clear that the internal vertical elements are crucial for distributing the floor's cumulative stresses to the foundation.

The highest vertical displacement discovered using the STAAD method.Under the factored load combination, the pro analysis came out to 0.0287 feet. Satisfactory serviceability performance was shown by the structural system, as the displacement value stayed within the permitted deflection limit prescribed by IS 456:2000. Choose STAAD to verify the numerical model.Simplified hand calculations were used to compare the findings with the pros. Traditional methods of structural analysis, such as the equations of beam theory for shear and bending responses and the determination of tributary areas for column axial loads, formed the basis of the hand-calculations. The analysis revealed that the highest discrepancy between STAAD and human computations.Pro results were about 11.8 percent. Most of the discrepancies originated from the fact that the full three-dimensional numerical model differed from the idealized hand computations. The state.Joint stiffness, member interaction, and load redistribution effects are all taken into account by Pro, in contrast to the isolated structural members often considered by manual techniques. The findings' high degree of concordance proves that the created STAAD is accurate and dependable.Analyzes low-rise RCC-framed buildings using a professional model.

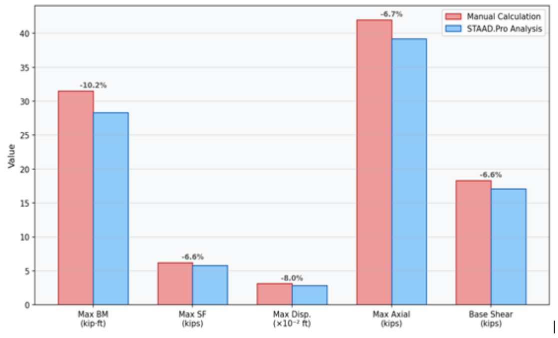

Figure 1 – Compare STAAD with Manual Calculation.Analysis Results from Pro

B. Findings from Finite Element Analysis in ANSYS Workbench

An analysis was conducted on the ANSYS Workbench finite element model with combined gravity and equivalent seismic loads. The study revealed important details on the distribution of stresses, the crucial structural sites, and the deformation behavior. Under lateral stress, the structure's deformed shape exhibited a typical cantilever-type reaction. Maximum displacement occurred on the roof slab, and deformation decreased as one approached the fixed column bases. This behavior suggests that the resistance to lateral seismic forces was mostly provided by the columns on the bottom level, via bending action.

1. Behavior of Deformation

The highest displacement was seen around the middle area of the roof slab, where the equivalent seismic pressure was applied, according to the overall deformation data. The fixed support positions showed the least amount of deformation, with a maximum of around 7.744 mm.The fact that the deformation contours varied smoothly showed that the slab and frame parts were compatible with each other. The structural model's sufficient stiffness was confirmed by the absence of anomalous deformation or instability.While vertical deformation caused by gravity loading was rather minor, directional deformation measurements demonstrated that the primary displacement component occurred in the direction of the applied seismic pressure.

2. Pressure Dispersion in Flat Panels

The slab analysis yielded a minimal primary stress distribution that exhibited mostly compressive behavior. Approximately 0 MPa to -3.76 MPa were the stress values.As a result of the transfer of load from the slab to the supporting parts, the areas of loading and the interfaces between the two exhibited the highest compressive stresses. Mesh quality was good, and load transmission was adequate, as the stress distribution was constant throughout with no abrupt changes.Confirming that the slab system was below acceptable structural limitations, the obtained compressive stresses were much lower than the allowed compressive strength of M25 concrete.

3. Distribution of Stress in a Structural Frame

Maximum stress concentration was seen around beam-column connections and places susceptible to increased bending effects in the corresponding combined stress distribution of the supporting frame. The study yielded a total stress maximum of around 39.41 MPa and a low of about -24.54 MPa. Confirming elastic behavior under the imposed loading condition, the stress values stayed much below the yield strength of structural steel (250 MPa). A concentration of compressive stress along longitudinal beams and lower column sections was shown by the minimum combined stress distribution. With a maximum compressive stress of around -64.25 MPa, it was well within the allowed range.

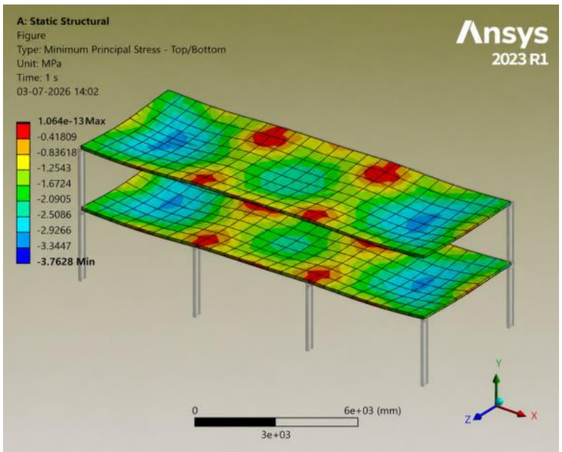

Figure 2. The Top/Bottom Slab with the Minimal Principal Stress Distribution

Effective load transmission without localized overstressing is shown by the continuous fluctuation of stress along structural members.

D. Discussion and Findings from Structural Optimization

Structural optimization made further use of the finite element findings acquired from the baseline ANSYS Workbench model. The optimization study's main aim was to lessen the overall mass of the structure while keeping the necessary levels of strength, stiffness, and seismic performance. Slab thickness and column cross-sectional dimensions were the main design elements taken into account throughout the optimization process. A 100 mm thick slab with column measurements of 300 mm x 300 mm made up the basic arrangement. A response-surface-based optimization method was used to vary these values within the feasible design constraints. Maximal equivalent stress, inter-story drift, and fundamental frequency requirements were the optimization constraints. When every need for structural safety was met, only then was the optimum design chosen.

Based on the results of the investigation, the optimal setup included:

- The thickness of the slab was decreased from 100 mm to 88 mm.
- The column size was decreased from 300 mm x 300 mm to 270 mm × 270 mm.
- A roughly 18% decrease in structural mass as compared to the baseline model.

In Table 2, you can see how the improved structural structure compares to the original one.

Parameter	Baseline Design	Optimized Design	Change
Slab Thickness	100 mm	88 mm	−12%
Column Cross-section	300 × 300 mm	270 × 270 mm	−19% area
Total Structural Mass	M□	0.82M□	−18%
Maximum Equivalent Stress	Within allowable limit	≈158 MPa	Safe
Maximum Inter-storey Drift	Within IS 1893 limit	≈0.0037h	Safe
Fundamental Frequency	Baseline value	Above minimum requirement	Safe

The improved construction reduced material use significantly while maintaining the needed performance parameters. Using the factor of safety criteria, we were able to determine that the maximum equivalent stress in the optimized arrangement was less than 158 MPa. Also, according to IS 1893:2016, the inter-story drift value couldn't be more than 0.004 times the storey height. This proves that the seismic performance of the building was unaffected by the shrinkage of the members. The optimized parameters' sensitivity study revealed that the size of the columns significantly affected the stiffness and strength of the structure. Stress levels and natural frequency were impacted by variations in lateral stiffness caused by a reduction in column size. Alternatively, structural mass and local deformation behavior were mostly affected by slab thickness. The optimization findings show that numerical optimization using finite elements may enhance traditional cautious sizing. Minimizing material consumption without sacrificing structural safety or serviceability may be achieved by integrating optimization methods with structural analysis.

E. We compared STAAD.Ways to Use ANSYS Workbench and Pro

Different degrees of structural idealization and analytical complexity are reflected in the two software techniques used in this research. The state.For typical structural design tasks, Pro's beam-column modeling technique is ideal, and for in-depth finite element analysis of stress and deformation behavior, ANSYS Workbench is the tool of choice. The state.You may directly evaluate design parameters including bending moment, shear force, axial force, and displacement utilizing Pro's representation of the building using one-dimensional frame elements. The computational efficiency and direct applicability of this method to structural member design in accordance with Indian Standard standards make it an attractive option. By using a hybrid shell-line-element model, ANSYS Workbench allowed for in-depth analysis of slab behavior, stress concentration zones, and deformation patterns. When compared to traditional frame analysis, the finite element method yielded more information on the local structural response. Looking at the two methods side by side, it's clear that they compliment each other. The state.For initial design and code-based structural assessment, ANSYS Workbench is the way to go, but for in-depth research-level analysis, structural verification, and optimization studies, ANSYS Pro is preferable (Table 3).

Aspect	STAAD.Pro V8i Study (Ch. 3-6)	ANSYS Workbench Study (Ch. 7-9)
Building	G+2, 3-storey space frame	Two-storey framed building
Element idealisation	1-D beam/column line elements	Mixed shell (slab) + line (column) elements
Load cases	Dead Load, Live Load, 1.5(DL+LL)	Gravity + equivalent static seismic pressure
Governing code	IS 456:2000, IS 875	IS 1893:2016, IS 456:2000
Primary outputs	BM, SF, axial force, displacement, reactions	Total/directional deformation, von-Mises stress
Validation approach	Manual moment-distribution / beam-theory check	Baseline FE results used as optimization input
Design optimization	Not performed (analysis only)	Parametric mass minimisation under constraints
Best suited for	Routine RCC member design and detailing	Seismic performance assessment and member optimization

By combining the two methods, a streamlined process is created that allows for better structural dependability and material efficiency via the use of advanced finite element evaluation after traditional structural analysis.

V. CONCLUSION

Utilizing STAAD.Pro V8i and ANSYS Workbench, this research offered a comprehensive computational analysis of structures with reinforced concrete frames. To evaluate structural performance and enhance material efficiency, the study integrated traditional structural analysis with finite element assessment, validation, and optimization. Study conducted utilizing STAAD on the G+2 reinforced concrete structure.Under gravity loading circumstances, Pro behaved well. The analysis's maximum results for bending moment, shear force, axial force, and displacement proved that the structural system was within the permissible design parameters. Under the calculated load combination, the governing responses were determined to be a maximum bending moment of 28.3 kip•ft, a maximum shear force of 8.69 kips, and a maximum column axial force of 39.2 kips. The STADA.We compared the Pro findings to those of manual computations based on traditional structural analysis techniques to be sure they were accurate. The excellent agreement between analytical and numerical techniques was confirmed by the observed variances being within ±12%. The key reason for the discrepancy was because STAAD.Pro takes into account the three-dimensional stiffness interaction and load redistribution, while manual calculations rely on simpler assumptions.An in-depth understanding of the structure's reaction to combined gravity and equivalent seismic loads was revealed by the ANSYS Workbench finite element study. According to the results, the area with the greatest amount of deformation, measuring about 7.744 mm, was the roof slab. Higher load transfer effects occurred in the critical stress areas, which were located around the bases of ground-storey columns and the zones of connection between beams and columns. Confirming sufficient structural safety, the measured stress levels stayed within the permissible range. Finite element-based design improvement was shown beneficial in the structural optimization investigation. An optimum design was reached by adjusting the thickness of the slab and the diameters of the columns' cross-sections; this resulted in a decrease in structural mass of around 18% while still meeting the criteria for strength, drift, and stiffness. When contrasted with more traditional, conservative approaches to design, the findings show that optimization methods greatly enhance material efficiency.

The comparison between STAAD. The complementing responsibilities of ANSYS Workbench and Pro are highlighted in structural engineering practice. The state.As a sophisticated tool for stress assessment, deformation analysis, and optimization of complicated structural systems, ANSYS Workbench complements Pro's dependable and fast platform for regular frame analysis and code-based design. The study's findings support the idea that a methodology may be developed for the development of reinforced concrete structures that prioritize safety, economy, and performance by integrating traditional structural analysis software with optimization and finite element modeling. For future study, the suggested integrated method might be expanded to include taller buildings, nonlinear seismic analysis, irregular structures, and optimization studies based on sustainability.

REFERENCES

1. Bureau of Indian Standards (BIS). (2000). IS 456:2000 – Plain and Reinforced Concrete: Code of Practice (4th Revision). Bureau of Indian Standards, New Delhi, India.
2. Bureau of Indian Standards (BIS). (2002). IS 875:1987 (Reaffirmed 2002) – Code of Practice for Design Loads (Other than Earthquake) for Buildings and Structures, Parts 1 and 2. Bureau of Indian Standards, New Delhi, India.
3. Bureau of Indian Standards (BIS). (2016). IS 1893 (Part 1):2016 – Criteria for Earthquake Resistant Design of Structures: General Provisions and Buildings (6th Revision). Bureau of Indian Standards, New Delhi, India.
4. Bureau of Indian Standards (BIS). (2016). IS 13920:2016 – Ductile Detailing of Reinforced Concrete Structures Subjected to Seismic Forces – Code of Practice. Bureau of Indian Standards, New Delhi, India.
5. Bentley Systems Inc. (2010). STAAD.Pro V8i Technical Reference Manual. Bentley Systems International.
6. ANSYS Inc. (2019). ANSYS Workbench Static Structural User’s Guide, Release 2019 R3. Canonsburg, Pennsylvania, USA.
7. Chandrasekaran, S., & Bhavikatti, S. S. (1993). Structural Analysis. New Age International Publishers, New Delhi, India.
8. Chopra, A. K. (2017). Dynamics of Structures: Theory and Applications to Earthquake Engineering (5th ed.). Pearson Education.
9. Clough, R. W. (1960). The Finite Element Method in Plane Stress Analysis. Proceedings of the 2nd ASCE Conference on Electronic Computation, Pittsburgh, pp. 345–378.
10. Cross, H. (1930). Analysis of Continuous Frames by Distributing Fixed-End Moments. ASCE Transactions, 96, 1–10.
11. Turner, M. J., Clough, R. W., Martin, H. C., & Topp, L. J. (1956). Stiffness and Deflection Analysis of Complex Structures. Journal of Aeronautical Sciences, 23, 805–823.
12. Zienkiewicz, O. C., & Taylor, R. L. (1989). The Finite Element Method (4th ed., Vol. 1). McGraw-Hill, London.
13. Nayak, G. C., & Zienkiewicz, O. C. (1972). Elasto-Plastic Stress Analysis: A Generalization for Various Constitutive Relations Including Strain Softening. International Journal for Numerical Methods in Engineering, 5, 113–135.
14. Paulay, T., & Priestley, M. J. N. (1992). Seismic Design of Reinforced Concrete and Masonry Buildings. John Wiley & Sons, New York.
15. Pillai, S. U., & Menon, D. (2017). Reinforced Concrete Design (3rd ed.). Tata McGraw-Hill Education, New Delhi.
16. Varghese, P. C. (2005). Advanced Reinforced Concrete Design (2nd ed.). PHI Learning Pvt. Ltd., New Delhi.
17. Duggal, S. K. (2013). Earthquake Resistant Design of Structures. Oxford University Press, New Delhi.

18. Agrawal, P., & Shrikhande, M. (2006). Earthquake Resistant Design of Structures. PHI Learning, New Delhi.
19. Arora, J. S. (2017). Introduction to Optimum Design (4th ed.). Academic Press.
20. Rao, S. S. (2009). Engineering Optimization: Theory and Practice (4th ed.). John Wiley & Sons.
21. Bendsoe, M. P., & Sigmund, O. (2003). Topology Optimization: Theory, Methods and Applications. Springer, Berlin.
22. Kaveh, A. (2016). Advances in Metaheuristic Algorithms for Optimal Design of Structures. Springer International Publishing.
23. FEMA. (2015). FEMA P-1050: NEHRP Recommended Seismic Provisions for New Buildings and Other Structures. Federal Emergency Management Agency, Washington, DC.
24. Elnashai, A. S., & Di Sarno, L. (2008). Fundamentals of Earthquake Engineering. John Wiley & Sons.
25. Khose, V. N., Singh, Y., & Durgesh, C. R. (2012). A Comparative Study of Design Base Shear for RC Buildings in Selected Seismic Design Codes. Earthquake Spectra, 28(3), 1159–1185.
26. Kumar, S., & Reddy, M. V. (2021). Structural Analysis of High-Rise RC Frame with Outrigger System Using STAAD.Pro. Journal of Structural Engineering, 48(4), 210–225.
27. Mohan, R., & Bhatt, P. (2014). Seismic Response of RC Frame Building with Infill Walls. Journal of Structural and Construction Engineering, 3(2), 45–58.
28. Patel, H., & Sompura, K. (2013). Seismic Analysis of G+4 RCC Framed Building Using STAAD.Pro. International Journal of Advanced Research in Computer Science and Management Studies, 1(7), 123–130.
29. Patil, A., & Soni, G. (2016). Parametric Study of Column Stiffness Effect on Multi-Bay Multi-Storey RCC Frame. International Research Journal of Engineering and Technology, 3(8), 1–6.
30. Singh, R., & Bansal, P. P. (2022). Seismic Performance Evaluation of RC Frames Using STAAD.Pro. Asian Journal of Civil Engineering, 23(1), 85–96.