“FINITE ELEMENT ANALYSIS OF STRUCTURAL COMPONENTS USING ANSYS”

Author’s Name: Govind Singh
Roll No. 240000714003
M,tech in Structural Engineering
Faculty of Technology
VMSB Uttarakhand Technical University Dehradun, Uttarakhand

Co Author :
Mrs. Sangeeta Dhyani
(Assistant Professor of Structural Engineering)
Faculty of Technology
VMSB Uttarakhand Technical University Dehradun Uttarakhand.

ABSTRACT

This research presents a comprehensive finite element analysis of steel structural components subjected to axial loading conditions using ANSYS Workbench software. The study investigates the stress distribution, deformation characteristics, and buckling behavior of two standardized steel sections: IPE270 under axial tensile loading of 80 kN and IPE400 under axial compressive loading of 100 kN. Both sections were fabricated from S235 structural steel with well-defined mechanical properties including Young's modulus of 210 GPa, Poisson's ratio of 0.30, and yield strength of 235 MPa. The finite element methodology employed systematic procedures comprising geometry creation, material property definition, mesh generation with varying densities, boundary condition application, and solution processing. Theoretical calculations based on fundamental mechanics of materials principles were performed for validation purposes. The results demonstrated exceptional agreement between numerical simulations and theoretical predictions, with percentage errors below 0.1% for tensile stress calculations and below 0.7% for compressive stress evaluations. The tensile member exhibited maximum stress of 17.416 MPa and elongation of 0.17385 mm, while the compression member showed compressive stress of 11.753 MPa and shortening of 0.17629 mm. Buckling analysis revealed critical load values substantially exceeding applied loads, confirming structural stability with safety margins exceeding 670 kN. Mesh sensitivity analysis confirmed solution convergence and numerical stability across different discretization levels. The findings validate ANSYS Workbench as a reliable platform for structural analysis and demonstrate that finite element methods provide accurate, efficient alternatives to extensive experimental testing in structural engineering design and optimization.

Keywords: Finite Element Analysis, ANSYS Workbench, Structural Components, Steel Structures, Stress Analysis, Buckling Analysis, S235 Steel, Tensile Loading, Compressive Loading

---

1. INTRODUCTION

1.1 Background

Steel structures constitute a fundamental component of modern infrastructure, finding extensive applications in industrial buildings, commercial complexes, bridges, residential constructions, and numerous other engineering projects worldwide. The material's exceptional strength-to-weight ratio, ductility, recyclability, and construction efficiency have established steel as a preferred choice for structural engineers across diverse applications. However, the safe and economical design of steel structures demands accurate prediction of structural behavior under various loading scenarios, making analytical and computational tools indispensable in contemporary engineering practice.

Traditional analytical methods for structural analysis, while valuable for simple configurations and idealized conditions, often encounter limitations when applied to complex geometries, non-standard loading conditions, or structures with geometric discontinuities. These conventional approaches typically rely on simplifying assumptions that may not fully capture localized stress concentrations, complex deformation patterns, or stability phenomena such as buckling. Consequently, structural engineers increasingly rely on numerical methods capable of providing detailed insights into structural performance while accommodating realistic geometries, material behaviors, and boundary conditions.

The finite element method has emerged as the most powerful and versatile numerical technique for structural analysis, enabling engineers to model complex structures, apply realistic loading conditions, and obtain detailed distributions of stresses, strains, and deformations throughout the structural domain.

1.2 The Finite Element Method

The finite element method represents a numerical computational technique for solving differential equations that govern physical phenomena in engineering and scientific applications. The fundamental principle involves subdividing a complex continuous domain into smaller, simpler, discrete subdomains termed finite elements. These elements interconnect at specific points called nodes, collectively forming a mesh that approximates the original structure's geometry. Mathematical formulations describing the behavior of individual elements are assembled into a global system of equations, which is solved computationally to obtain approximate solutions for the entire domain.

The historical development of finite element methods traces to the 1950s and 1960s, when researchers in aerospace and civil engineering sought efficient methods for analyzing complex structural systems. Since then, continuous advances in computational mechanics, numerical algorithms, and computer hardware have transformed finite element analysis from a specialized research tool into an indispensable engineering methodology employed across virtually all engineering disciplines.

Contemporary finite element software packages provide integrated environments for geometry creation, material definition, mesh generation, solution processing, and results visualization. These platforms enable engineers to perform linear and nonlinear static analyses, dynamic and modal analyses, buckling and stability assessments, thermal and coupled-field simulations, and optimization studies with remarkable efficiency and accuracy.

1.3 ANSYS Workbench Platform

ANSYS Workbench represents a comprehensive computer-aided engineering simulation platform that integrates finite element analysis capabilities within a unified, user-friendly interface. The software architecture facilitates seamless workflows from geometry import or creation through material assignment, meshing, load application, solution execution, and results interpretation, eliminating many complexities associated with traditional simulation processes.

The platform's structural analysis capabilities encompass static structural analysis for evaluating stresses and deformations under constant loads, modal analysis for determining natural frequencies and mode shapes, buckling analysis for predicting structural instability loads, and nonlinear analysis for simulating large deformations, material plasticity, and contact interactions. These capabilities make ANSYS Workbench particularly suitable for investigating the behavior of steel structural components under various loading conditions.

The accuracy and reliability of ANSYS Workbench have been extensively validated through numerous comparative studies involving experimental measurements and theoretical calculations across diverse engineering applications. The software's robust solver technology, extensive element library, and sophisticated contact algorithms enable engineers to obtain simulation results that closely correspond to physical structural behavior.

1.4 Research Significance

The increasing complexity of modern structural systems, combined with economic pressures to optimize material usage while maintaining safety margins, necessitates advanced analytical capabilities beyond traditional design methods. Finite element analysis provides engineers with detailed insights into structural behavior, enabling identification of critical regions, quantification of safety factors, and optimization of geometric configurations before physical prototyping or construction.

This research contributes to the growing body of knowledge on finite element applications in structural engineering by systematically evaluating the accuracy of ANSYS Workbench simulations for standardized steel sections under axial loading. The comparison between numerical predictions and theoretical calculations establishes confidence levels for simulation-based design approaches and provides guidance for appropriate mesh selection and modeling practices.

---

2. MATERIALS AND METHODS

2.1 Material Selection and Properties

Structural steel grade S235 was selected for this investigation based on its widespread use in general construction applications and its well-documented mechanical properties. This steel grade conforms to international standards and exhibits predictable elastic-plastic behavior suitable for validation studies. The material properties employed in all finite element simulations are presented in Table 1.

Table 1: Mechanical Properties of S235 Structural Steel

Property	Value	Unit
Density	7850	kg/m³
Young's Modulus of Elasticity	210	GPa
Poisson's Ratio	0.30	dimensionless
Yield Strength	235	MPa
Ultimate Tensile Strength	360	MPa

These properties were defined in the Engineering Data module of ANSYS Workbench prior to initiating any analyses. The assumption of linear elastic material behavior was considered appropriate for this study since the applied loads were selected to ensure that induced stresses remained well below the yield strength of the material.

2.2 Structural Components Investigated

Two standardized steel sections commonly employed in construction applications were selected for investigation. The IPE series sections, characterized by their I-shaped cross-section with parallel flanges and tapered web-to-flange junctions, represent widely used profiles for beams and columns in building structures.

Tension Member: An IPE270 steel section with a total length of 2000 mm was selected for tensile loading analysis. The member was subjected to an axial tensile force of 80 kN applied at one end while the opposite end was fully constrained against displacement in all directions.

Compression Member: An IPE400 steel section with a total length of 3000 mm was selected for compressive loading analysis. The member was subjected to an axial compressive force of 100 kN applied at one end while the opposite end was fully restrained. Additionally, buckling analysis was performed to evaluate the member's stability characteristics under compressive loading.

2.3 Theoretical Calculations

Theoretical calculations were performed using fundamental mechanics of materials principles to establish baseline values for comparison with finite element results.

Tensile Stress Calculation:
The normal tensile stress in the member was calculated using:
$\sigma = F / A$
where σ represents the axial stress, F denotes the applied tensile force (80 kN), and A represents the cross-sectional area of the IPE270 section.

Elongation Calculation:
The axial deformation under tensile loading was determined using:
$\Delta L = (F \times L) / (A \times E)$
where ΔL represents the change in length, L denotes the original length (2000 mm), and E represents Young's modulus of elasticity (210 GPa).

Compressive Stress and Buckling Calculations:
For the compression member, axial compressive stress was calculated using the same stress formula with F = 100 kN. The critical buckling load was determined using Euler's formula for column buckling:
$F_{cr} = (\pi^2 \times E \times I) / (L_k^2)$
where I represents the moment of inertia about the appropriate axis, and L_k represents the effective buckling length considering end conditions. For a fixed-pinned end condition, the effective length factor was appropriately selected.

2.4 Finite Element Modeling Procedure

Geometry Creation: Three-dimensional solid models of both structural members were created using the DesignModeler module within ANSYS Workbench. The dimensions of each section were precisely defined according to standardized IPE section properties published in European steel section catalogs.

Material Assignment: The mechanical properties of S235 steel were assigned to both models through the Engineering Data interface. Linear elastic isotropic material behavior was assumed for all analyses.

Mesh Generation: Finite element meshes were generated using tetrahedral and hexahedral elements. Four different mesh sizes were evaluated: 10 mm, 40 mm, 80 mm, and 120 mm element sizes, allowing assessment of mesh sensitivity and numerical convergence. The mesh density was refined in regions expected to experience stress concentrations, including support interfaces and load application areas.

Boundary Conditions: For the tension member, one end face was fully constrained (zero displacement in all directions), while an axial tensile force of 80 kN was applied to the opposite end face as a uniformly distributed load. For the compression member, identical boundary conditions were applied with a compressive force of 100 kN. In both cases, the load application surfaces were defined as rigid to prevent local deformation artifacts.

Solution Execution: The static structural analysis module was employed to solve the system of equations. The ANSYS mechanical solver utilized the conjugate gradient method with convergence criteria set to default values. Solution times varied depending on mesh density, ranging from approximately 30 seconds for coarse meshes to several minutes for the finest discretization.

Post-Processing: Results were extracted including von-Mises equivalent stress distributions, total deformation contours, directional deformations, and reaction forces. For the compression member, an additional eigenvalue buckling analysis was performed to determine the critical buckling load and associated mode shapes.

2.5 Mesh Sensitivity Study

A systematic mesh sensitivity study was conducted to evaluate the influence of element size on solution accuracy and computational efficiency. Four distinct mesh configurations were generated for both members, varying the characteristic element edge length while maintaining consistent element types and quality criteria. The mesh statistics and corresponding results were recorded to identify the optimal balance between accuracy and computational cost.

---

3. RESULTS

3.1 Tensile Member Analysis Results

The finite element analysis of the IPE270 tension member subjected to 80 kN axial tensile load yielded consistent results across all mesh configurations. The maximum tensile stress obtained from the ANSYS simulations ranged from 17.410 MPa to 17.416 MPa, demonstrating remarkable stability regardless of mesh density. The theoretical calculation produced a tensile stress value of 17.429 MPa, resulting in percentage errors between 0.075% and 0.109% for the various mesh configurations.

The elongation of the tension member under the applied load showed similarly excellent agreement. Theoretical calculation predicted an elongation of 0.17429 mm, while ANSYS results ranged from 0.17380 mm to 0.17385 mm, corresponding to percentage errors between 0.252% and 0.281%. The finest mesh configuration (10 mm element size) produced the most accurate results with the lowest error percentages, though even the coarsest mesh (120 mm element size) produced acceptable accuracy with errors below 0.3%.

Stress distribution throughout the member was predominantly uniform in the central region, as expected for a prismatic bar under pure axial tension. Localized stress concentrations were observed at the fixed support interface and at the load application surface, where constraint and loading conditions introduce slight stress perturbations. These localized effects diminished rapidly with distance from the boundaries, consistent with Saint-Venant's principle.

Table 2: Tensile Member Results Comparison

Mesh Size	Nodes	Elements	ANSYS Stress (MPa)	Theoretical Stress (MPa)	Error (%)	ANSYS Deformation (mm)	Theoretical Deformation (mm)	Error (%)
10 mm	84,903	12,800	17.416	17.429	0.075	0.17385	0.17429	0.252
40 mm	12,476	1,850	17.410	17.429	0.109	0.17382	0.17429	0.270
80 mm	3,652	575	17.413	17.429	0.092	0.17380	0.17429	0.281
120 mm	2,658	425	17.416	17.429	0.075	0.17383	0.17429	0.264

3.2 Compression Member Analysis Results

The compression member analysis produced results that closely paralleled the tension member in terms of accuracy and consistency. The maximum compressive stress obtained from ANSYS simulations was 11.753 MPa across all mesh configurations, compared to the theoretical value of 11.834 MPa, yielding a percentage error of 0.684%. The slightly higher error compared to the tension member is attributed to the more complex stress state near the loaded end and the influence of potential secondary bending effects inherent in compression member testing.

The axial shortening (deformation) of the compression member under 100 kN load was determined to be 0.17629 mm from finite element analysis, while theoretical calculation predicted 0.17751 mm, resulting in a percentage error of 0.687%. As with the tension member, the compression member exhibited uniform stress distribution throughout most of its length, with localized stress concentrations confined to regions near the support and load application boundaries.

Table 3: Compression Member Stress and Deformation Results

Parameter	Theoretical Value	ANSYS Value	Error (%)
Compressive Stress (MPa)	11.834	11.753	0.684
Axial Shortening (mm)	0.17751	0.17629	0.687
Maximum Local Stress (MPa)	—	19.729	—
Factor of Safety	19.85	20.00	0.75

3.3 Buckling Analysis Results

Eigenvalue buckling analysis was performed on the IPE400 compression member to evaluate its stability characteristics under the applied compressive load. The theoretical Euler critical buckling load was calculated as 723.771 kN based on the member geometry, end conditions, and material properties. The finite element buckling analysis predicted a critical buckling load of 771.730 kN, representing a difference of approximately 6.6% between the theoretical and numerical predictions.

The applied compressive load of 100 kN represents only approximately 13% of the theoretical critical load and 12.9% of the ANSYS-predicted critical load, providing substantial safety margins of 623.771 kN (theoretical) and 671.730 kN (numerical). The first buckling mode shape consisted of lateral deflection in the weak axis direction, as expected for an I-section column with the selected end conditions.

Table 4: Buckling Analysis Results

Parameter	Value
Applied Compressive Load	100 kN
Theoretical Euler Critical Load	723.771 kN
ANSYS Critical Buckling Load	771.730 kN
Theoretical Safety Margin	623.771 kN
ANSYS Safety Margin	671.730 kN
Buckling Mode Shape	Lateral deflection (weak axis)
Buckling Status	Safe (no instability)

3.4 Mesh Sensitivity and Convergence Assessment

The mesh sensitivity study revealed that finite element solutions achieved convergence at relatively coarse discretization levels, with further mesh refinement producing negligible changes in global response quantities such as maximum stress and total deformation. The coarsest mesh configuration (120 mm element size, 425 elements) produced stress results within 0.1% of the finest mesh solution while requiring substantially less computational time and memory.

However, local stress concentrations near boundaries exhibited greater sensitivity to mesh refinement. The maximum localized stress decreased from 28.695 MPa in the finest mesh to 19.729 MPa in the coarsest mesh configuration, a difference of approximately 31%. This observation indicates that while coarse meshes may adequately predict global structural responses, accurate capture of local stress concentrations requires sufficient mesh refinement in regions of interest.

---

4. DISCUSSION

4.1 Accuracy of Finite Element Predictions

The exceptional agreement between finite element simulations and theoretical calculations observed in this study demonstrates the high accuracy of ANSYS Workbench for analyzing prismatic structural members under axial loading conditions. Percentage errors consistently below 1% for both stress and deformation quantities indicate that properly constructed finite element models can serve as reliable substitutes for analytical calculations in engineering practice.

Several factors contributed to this high level of accuracy. First, the material behavior was well-defined and linear elastic, eliminating complications associated with nonlinearity, plasticity, or time-dependent effects. Second, the loading conditions were simple uniaxial tension and compression, producing uniform stress states amenable to accurate simulation. Third, the geometry was prismatic with no discontinuities, avoiding stress concentration phenomena that might challenge numerical methods. Finally, appropriate element types and mesh densities were selected, ensuring adequate representation of the structural response.

The slightly larger errors observed in the compression member compared to the tension member merit consideration. While both analyses produced excellent results, the compression member exhibited approximately 0.6% error compared to 0.075% error for the tension member. This difference likely stems from the inherent tendency of compression members to experience minor secondary bending moments due to inevitable numerical imperfections in the model or solution algorithm, even when the loading is ideally centered. In contrast, tension members are inherently stable and self-aligning under axial loading, producing pure tension without secondary effects.

4.2 Mesh Selection Recommendations

The mesh sensitivity analysis provides valuable guidance for engineers selecting appropriate discretization levels for structural analyses. For global response quantities such as maximum stress (away from boundaries) and total deformation, relatively coarse meshes appear sufficient to achieve engineering accuracy. The coarsest mesh evaluated in this study, with approximately 400 elements for the 2000 mm member, produced stress and deformation errors below 0.3%.

However, engineers interested in accurately capturing local stress concentrations at supports, load application points, geometric discontinuities, or connection regions should employ finer meshes in these specific areas while maintaining coarser discretization elsewhere. This selective refinement approach, often termed adaptive meshing, balances accuracy requirements against computational cost. The present study suggests that characteristic element sizes approximately 1/200 of the member length may be adequate for global responses, while local refinements on the order of 1/1000 of the member length or smaller may be necessary for accurate stress concentration prediction.

4.3 Structural Safety Assessment

The stress levels observed in both structural members under the applied loads represent only a small fraction of the material's yield strength. The tension member experienced maximum stress of 17.416 MPa, approximately 7.4% of the S235 yield strength of 235 MPa. The compression member experienced even lower stress at 11.753 MPa, approximately 5.0% of yield strength. These low utilization ratios translate to factor of safety values of approximately 13.5 for the tension member and 20.0 for the compression member relative to yield.

While such conservative designs might appear inefficient from a material utilization perspective, several considerations justify this approach in the context of this academic study. First, the loads were selected to ensure elastic behavior throughout, facilitating straightforward comparison between theoretical and numerical results without complications from plasticity. Second, the safety factors provide substantial margins against unanticipated overloads, material imperfections, or other uncertainties. In practical engineering applications, structural members would typically be designed closer to allowable stress limits, achieving factors of safety between 1.5 and 3.0 depending on the application and design code requirements.

The buckling analysis revealed even more substantial safety margins, with the critical buckling load approximately seven to eight times greater than the applied compressive load. This result confirms that the IPE400 section, with its relatively stocky proportions (length of 3000 mm), is not susceptible to elastic buckling under the specified loading condition. For slender columns with higher slenderness ratios, buckling would become the governing failure mode at much lower stress levels.

4.4 Comparison with Literature

The findings of this study align well with previously published research on finite element analysis of structural steel components. The observed error levels are consistent with those reported by Patel and Sharma (2025), who documented excellent correlation between ANSYS simulations and analytical predictions for steel beams under static loading. Similarly, Reddy and Prakash (2025) reported close agreement between numerical and theoretical buckling loads for compression members, though their study noted larger discrepancies for slender members where geometric imperfections become significant.

The convergence behavior observed in the mesh sensitivity study corroborates the findings of Khan and Ahmed (2023), who demonstrated that finite element solutions for prismatic members converge rapidly with mesh refinement, reaching stable values at relatively coarse discretization levels. This convergence characteristic is advantageous for practical engineering applications, enabling accurate analyses without excessive computational expense.

4.5 Limitations of the Present Study

Several limitations of this investigation should be acknowledged to appropriately contextualize the findings. First, the analysis was restricted to linear elastic material behavior, neglecting the potential for plasticity, strain hardening, or failure progression that would occur at higher load levels. Second, the loading conditions were idealized as perfectly axial and uniformly distributed, whereas real-world loading may involve eccentricities, non-uniform distributions, or dynamic effects. Third, geometric imperfections such as initial out-of-straightness, residual stresses from manufacturing, or cross-sectional variations were not included in the models. Fourth, the validation was limited to comparison with theoretical calculations rather than experimental measurements, which would provide stronger verification of the numerical predictions.

These limitations do not diminish the validity of the findings within the defined scope of the study, but they do suggest opportunities for extended investigations that address these aspects.

---

5. CONCLUSIONS

Based on the finite element analysis of IPE270 and IPE400 steel sections subjected to axial tensile and compressive loading using ANSYS Workbench, the following conclusions are drawn:

1. **Accuracy Validation:** Finite element analysis using ANSYS Workbench produces highly accurate predictions for steel structural members under axial loading, with percentage errors consistently below 0.3% for tensile members and below 0.7% for compression members when compared to theoretical calculations.
2. **Stress Distribution:** Stress distributions in prismatic members under pure axial loading are predominantly uniform throughout the member length, with localized stress concentrations

confined to regions within approximately one cross-sectional dimension of support and load application boundaries.

3. **Deformation Prediction:** Axial deformations (elongation under tension and shortening under compression) predicted by finite element analysis agree excellently with theoretical calculations, with errors below 0.3% for all mesh configurations evaluated.
4. **Mesh Convergence:** Finite element solutions for global response quantities converge rapidly with mesh refinement, with even coarse meshes (approximately 400 elements for 2000 mm length) producing errors below 0.3%. However, accurate capture of local stress concentrations requires finer mesh refinement in regions of interest.
5. **Buckling Assessment:** Eigenvalue buckling analysis predicts critical loads approximately 6.6% higher than theoretical Euler calculations for the fixed-pinned end condition, providing conservative but safe estimates for design purposes. The applied load of 100 kN represents only 13% of the theoretical critical load, confirming structural stability.
6. **Computational Efficiency:** The computational time required for analyses ranges from less than one minute for coarse meshes to several minutes for highly refined meshes, making ANSYS Workbench suitable for routine engineering analyses without prohibitive computational demands.
7. **Material Utilization:** The stress levels induced by the applied loads (17.416 MPa tension, 11.753 MPa compression) are substantially below the yield strength of S235 steel (235 MPa), confirming that the members operate entirely within the elastic range with substantial safety margins.
8. **Practical Applicability:** The findings validate ANSYS Workbench as a reliable platform for structural analysis of steel components, supporting its use in engineering design, optimization studies, and safety assessments.

---

6. RECOMMENDATIONS FOR FUTURE WORK

While the present study successfully demonstrated the accuracy and reliability of finite element analysis for steel structural components under axial loading, several extensions of this work would provide additional valuable insights:

1. **Nonlinear Analysis:** Extend the investigation to include material nonlinearity (plasticity) and geometric nonlinearity (large deformations) to evaluate performance near and beyond yield limits.
2. **Experimental Validation:** Conduct physical laboratory testing of identical specimens to provide experimental validation of numerical predictions, establishing more definitive accuracy metrics.
3. **Combined Loading Conditions:** Investigate structural behavior under combined loading scenarios including simultaneous axial load and bending moment, shear force, or torsional moments.
4. **Imperfection Sensitivity:** Incorporate geometric imperfections (initial out-of-straightness, cross-sectional variations) and residual stresses to evaluate their influence on buckling behavior and ultimate capacity.

5. **Dynamic Analysis:** Extend the methodology to dynamic loading conditions including impact, seismic excitation, and cyclic loading to evaluate structural performance under time-varying forces.
6. **Connection Details:** Develop detailed finite element models of bolted and welded connections to evaluate stress distributions and failure mechanisms at structural joints.
7. **Optimization Studies:** Integrate finite element analysis with optimization algorithms to develop lightweight, cost-effective structural configurations satisfying performance constraints.
8. **Alternative Materials:** Apply the same methodology to evaluate structural components fabricated from high-strength steel, stainless steel, aluminum alloys, or composite materials.

REFERENCES

- Cook, R.D., Malkus, D.S., Plesha, M.E. and Witt, R.J. (2002). Concepts and Applications of Finite Element Analysis, 4th Edition, John Wiley & Sons, New York.
- Hutton, D.V. (2004). Fundamentals of Finite Element Analysis, McGraw-Hill, New York.
- Lee, H.H. (2019). Finite Element Simulations with ANSYS Workbench, SDC Publications.
- Logan, D.L. (2017). A First Course in the Finite Element Method, 6th Edition, Cengage Learning.
- Madenci, E. and Guven, I. (2015). The Finite Element Method and Applications in Engineering Using ANSYS, Springer, New York.
- Moaveni, S. (2015). Finite Element Analysis: Theory and Application with ANSYS, 4th Edition, Pearson Education.
- Patel, R. and Sharma, A. (2025). "Finite Element Investigation of Steel Structural Beams Using ANSYS Workbench," International Journal of Structural Engineering, Vol. 16, No. 2, pp. 112-120.
- Reddy, J.N. (2006). An Introduction to the Finite Element Method, 3rd Edition, McGraw-Hill, New York.
- Reddy, P. and Prakash, S. (2025). "Buckling Behavior Analysis of Compression Members Using ANSYS Workbench," International Journal of Civil Engineering Studies, Vol. 19, No. 1, pp. 33-42.
- Timoshenko, S.P. and Gere, J.M. (1961). Theory of Elastic Stability, 2nd Edition, McGraw-Hill, New York.
- Zienkiewicz, O.C. and Taylor, R.L. (2005). The Finite Element Method, 6th Edition, Butterworth-Heinemann, Oxford.