ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Volume 13 Issue 02 February 2026

Structural Behaviour of Building under Various Loading Conditions: An ANSYS-Based Study

[1] Lokesh Marathe, [2] Dipak Thakur, [3] Anjali Dhotre, [4] Pooja Ritewad, [5] Sharmishta Chakraborty

[1][2][3][4][5] Civil Engineering, D.Y.Patil College of Engineering, Akurdi, Pune, India
Email: [1] patillokesh1805@gmail.com, [2] dipaksthakur9721@gmail.com, [3] anjalidhotre2004@gmail.com, [4] pritewad04@gmail.com, [5] sdchakraborty@dypcoeakurdi.ac.in

Abstract— This study investigates the structural behaviour of buildings under various loading conditions using ANSYS. Different load cases, including dead load, live load, wind load, and seismic forces, are applied to evaluate stress distribution, deformation, and overall stability. The analysis helps identify critical structural responses and potential failure zones. Results demonstrate the effectiveness of numerical simulation in predicting building performance. The findings contribute to safer and more efficient structural design practices.

Index Terms— Structural Behaviour, Ansys Simulation, Load Cases, Stress Distribution, Deformation, Structural Stability, Seismic and Wind Loads, Numerical Analysis, Failure Zones, Building Performance

I. INTRODUCTION

The safety, durability, and performance of buildings under various loading conditions are fundamental concerns in the field of structural engineering. Every structure is subjected to a combination of loads throughout its lifespan, including dead loads (self-weight of structural elements), live loads (occupancy and usage-related loads), wind loads, and seismic forces. These loads can act individually or simultaneously, producing complex stress patterns and deformation behaviours within the structure. An accurate assessment of how buildings respond to such loading conditions is essential to ensure structural integrity, serviceability, and long-term reliability.Traditionally, structural analysis was carried out using simplified analytical methods and assumptions, which often limited the ability to capture real-world behaviour, especially for complex geometries and loading scenarios. With the rapid advancement of computational technology, numerical methods such as the Finite Element Method (FEM) have revolutionized structural analysis. These methods allow engineers to model complex structures with greater precision and to simulate realistic loading conditions.

As a result, it has become possible to predict structural responses more accurately, identify critical stress regions, and evaluate potential failure mechanisms before actual construction.ANSYS, a widely used finite element analysis (FEA) software, provides a powerful platform for conducting such simulations. It enables detailed modelling of structural components, application of various load combinations, and analysis of resulting stresses, strains, and displacements. By using ANSYS, engineers can perform comprehensive investigations into how structures behave under different loading scenarios, thereby reducing reliance on costly and time-consuming experimental testing. Moreover, it facilitates optimization of design by allowing iterative modifications and performance evaluation under controlled virtual environments.

This research focuses on investigating the structural behaviour of a building under various loading conditions using ANSYS. The study involves modelling a building structure and subjecting it to different load cases, including dead load, live load, wind load, and seismic load. The analysis aims to evaluate key parameters such as stress distribution, deformation patterns, and overall stability of the structure. Special attention is given to identifying critical regions that are more susceptible to high stress concentrations or excessive displacement, which could indicate potential failure zones.The outcomes of this study are expected to demonstrate the effectiveness of ANSYS-based numerical simulation in predicting building performance under realistic loading conditions. The insights gained from the analysis can help engineers make informed decisions during the design phase, improve structural safety, and optimize material usage. Ultimately, this research contributes to the advancement of modern structural engineering practices by promoting the use of simulation tools for safer, more efficient, and cost-effective building design

II. OBJECTIVES

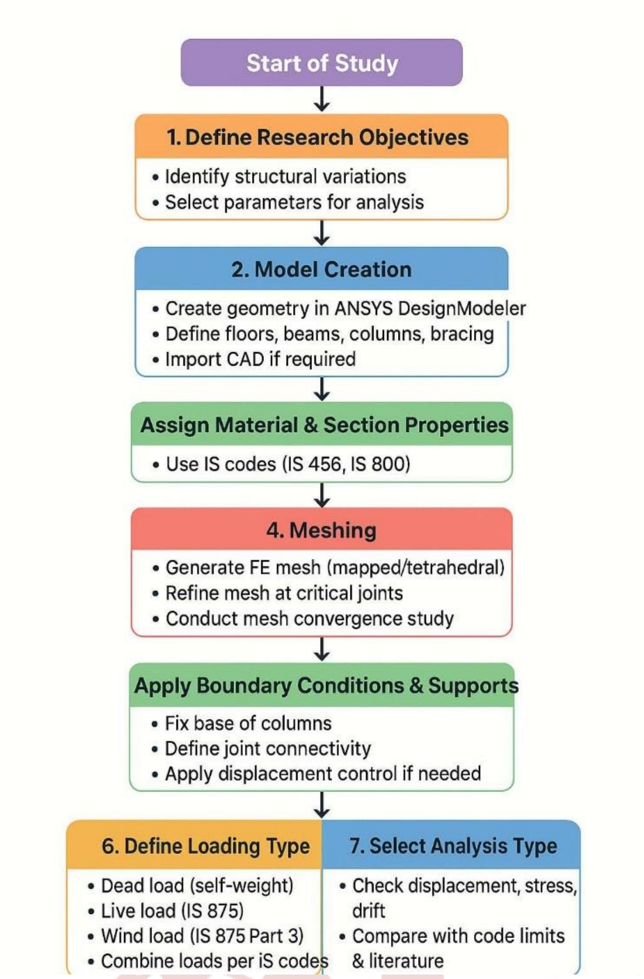
To analyze the structural behaviour of a building under various loading conditions using ANSYS.

- To apply different load cases such as dead load, live load, wind load, and seismic load on the building model.
- To evaluate stress distribution and identify critical stress regions within the structure.
- To study deformation and displacement patterns under different loading scenarios.
- To assess the overall stability and performance of the building.
- To identify potential failure zones and structural

weaknesses.

- To validate the effectiveness of ANSYS as a numerical simulation tool for structural analysis.
- To contribute to safer, more efficient, and optimized structural design practices

III. METHODOLOGY



1. Model Cration

- The structure geometry is defined in ANSYS Workbenchs DesignModeler or imported CAD model, capturing the multistorey frame, columns, beams, slabs, and bracing elements.
- The model reflects the actual architectural layout including storey levels, spans, and floor- to-floor heights, and may include details such as shear walls or bracing systems where applicable.
- Each structural component (e.g., beam, column, slab) is created as a separate body or part, enabling appropriate assignment of cross-sectional properties and material behaviour.

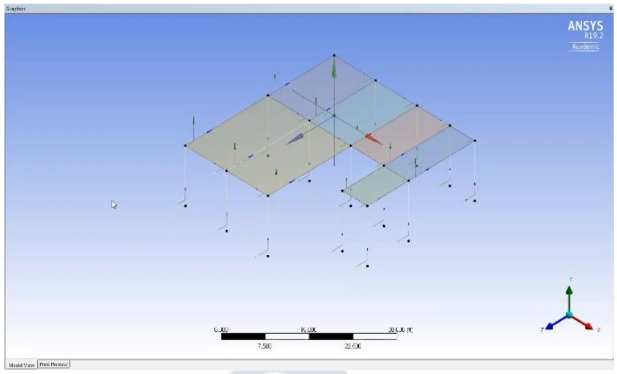


Fig.1 Model Creation

1.1 Material and Section Properties

- Assign material properties consistent with Indian standard codes (e.g., IS 800:2007 for steel, IS 456:2000 for concrete) —modulus of elasticity, Poisson' s ratio, yield strength,
- Define steel sections (e.g., I-section, channel, cold-formed section) or concrete sections according to the literature review insights.
- In the video, the presenter assigns structural steel with linear isotropic behaviour, which is adopted in this study along with nonlinear behaviour where required.
- Composite sections (if any) are modelled by combining steel and concrete properties or by using equivalent homogenised proper

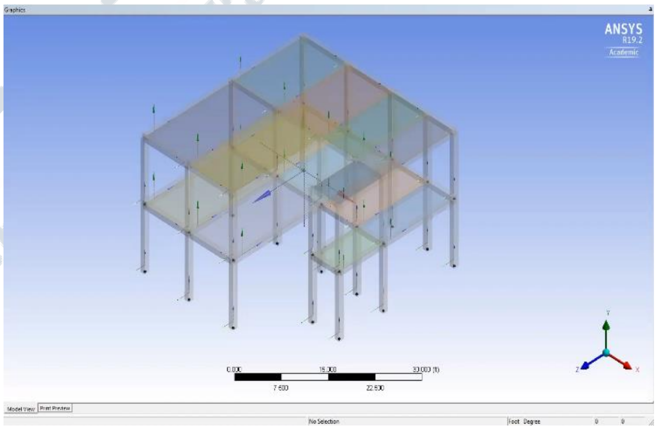


Fig.2 Material and Section Properties

1.2 Meshing

- Generate a finite element mesh with appropriate mesh size and element quality, ensuring convergence of results. In the video, a mapping mesh is used for regular geometry and tetrahedral or hexahedral elements for complex parts.
- Conduct mesh refinement at critical zones (connections, bracing nodes, slab– beam junctions) to capture local stress concentrations and deformations accurately.
- Perform a mesh sensitivity study by reducing element size until key result parameters (maximum displacement, stress peak) converge within acceptable variation (< 5%).

ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Volume 13 Issue 02 February 2026

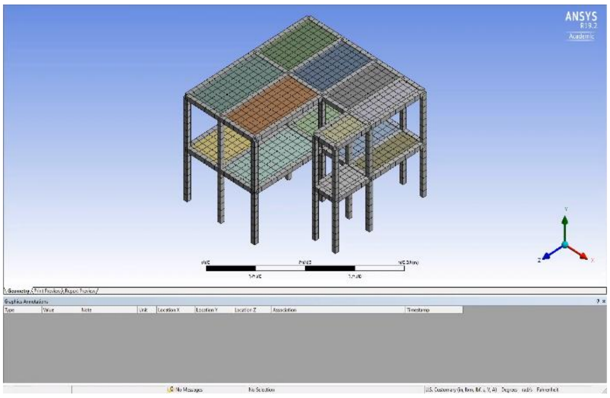


Fig.3 Meshing

1.3 Solution

1. Total Deformation

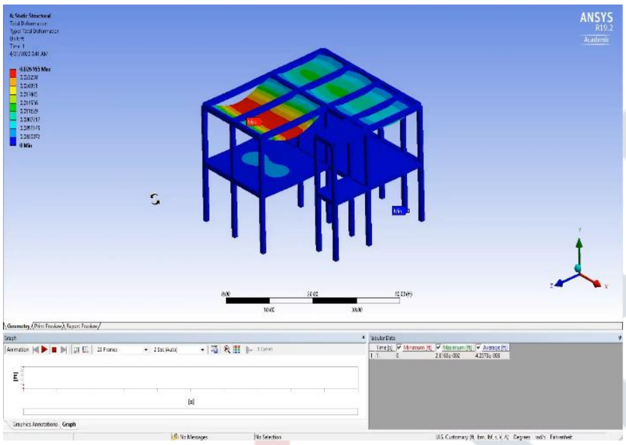


Fig.4 Total Deformation

- The total deformation of a framed building structure. The model consists of beams, columns, and a roof slab subjected to loads such as gravity. The deformation is represented using a color contour, where blue indicates minimum deformation (near zero at fixed supports) and red indicates maximum deformation (about 0.026 units at the roof slab).
- The results show that the maximum displacement occurs at the top slab, as it is the least constrained part of the structure, while the columns and base remain stable due to fixed supports. The gradual color variation from bottom to top indicates increasing deformation, confirming realistic structural behavior.
- This analysis helps in evaluating strength, stiffness, and safety, and suggests that if deformation is too high, design improvements like increasing member size or adding supports may be required.

2. Directional Deformation

- The analysis type selected is Static Structural, focusing on directional deformation along the Y-axis. The structure consists of columns, beams, and slab elements, forming a small building frame.
- Material properties and boundary conditions were applied, where the base supports are fixed to restrict all degrees of freedom. External loads such as gravity and pressure are applied to simulate real-life conditions. The mesh is generated automatically to discretize the model for numerical analysis.
- Maximum deformation: 0 ft
- Minimum deformation: -3.875×10^{-3} ft
- Average deformation: -1.0291×10^{-3} ft
- The deformation pattern indicates that the structure behaves safely under given loads, with maximum displacement within acceptable limits. Stress results such as principal stress and normal stress are also evaluated to ensure structural safety.
- Overall, the analysis confirms that the structure is stable, and deformation is within permissible limits, making the design safe and efficient under applied loading conditions.

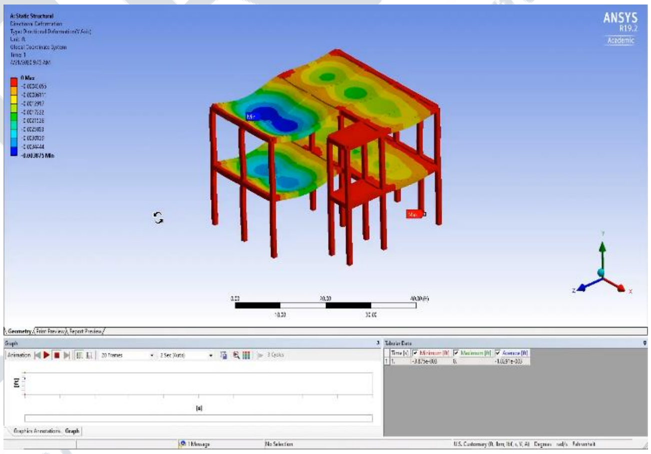


Fig.5 Directional Deformation

3. Maximum Principal Stress

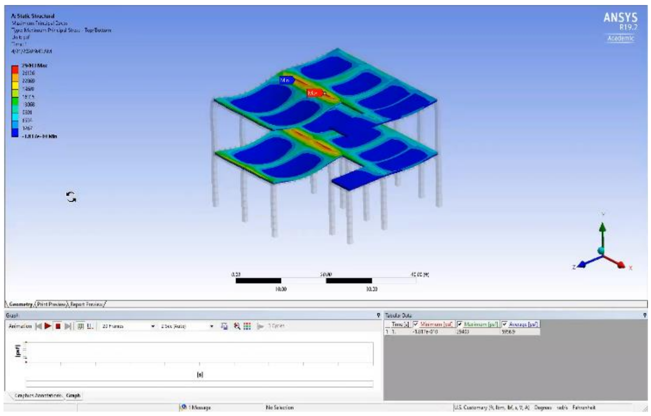


Fig 6 Maximum Principal Stress

- The result obtained represents the Maximum Principal Stress distribution on the structure at the top and bottom surfaces.

- The stress values range from a minimum of approximately -1.817×10^{-10} psf (negligible) to a maximum of 29,403 psf, with an average stress of about 5,956.9 psf. The stress contour indicates that the maximum stress is concentrated near the central beam/slab junction, highlighted in red color, whereas the minimum stress regions are located in the slab areas, shown in blue.
- The model consists of a multi-level slab supported by vertical columns, and the applied loading causes stress concentration mainly at the connections and load transfer regions. The deformation and stress distribution suggest that the structure is experiencing higher stress at the mid-span and joint regions, which are critical points for design consideration.
- Overall, the results help in identifying critical stress zones, ensuring structural safety, and optimizing the design to prevent failure.

4. Minimum Principal Stress

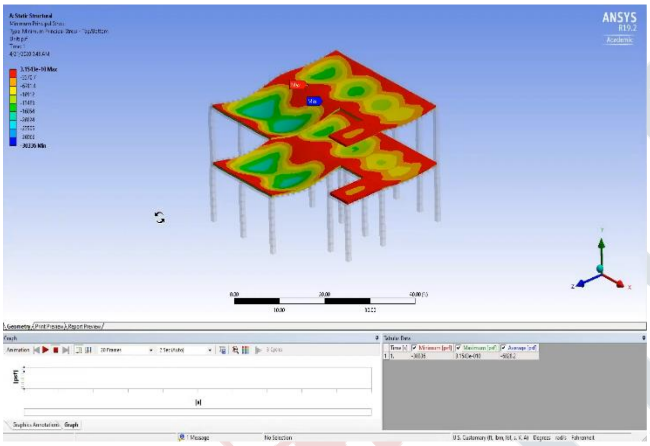


Fig.7. Minimum Principal Stress

- The distribution of minimum principal stress in a multi-storey slab system. The results indicate a maximum stress value close to zero and a minimum compressive stress of -30336 psf. The stress is unevenly distributed, with higher compression observed near the central regions of the slab and lower values at the edges.
- The color contour clearly represents stress variation, where red indicates maximum and blue indicates minimum stress. This analysis helps in understanding load transfer and identifying critical stress zones in the structure.

5. Normal Stress

- The distribution of normal stress (X-axis) across a multi-level structural system.
- The model represents a multi-storey slab supported by columns, where loads such as gravity and applied pressure are considered to evaluate structural behavior.
- The contour plot uses a color scale to indicate stress variation, where red/yellow regions represent maximum tensile stress (~18617 psf) and blue regions indicate maximum compressive stress (~ -18740 psf).
- The results show that stress concentrations occur mainly near slab edges and connection regions, while relatively uniform stress distribution is observed in central areas.
- The analysis helps in identifying critical zones prone to failure or excessive deformation, which is essential for improving design safety and material optimization.

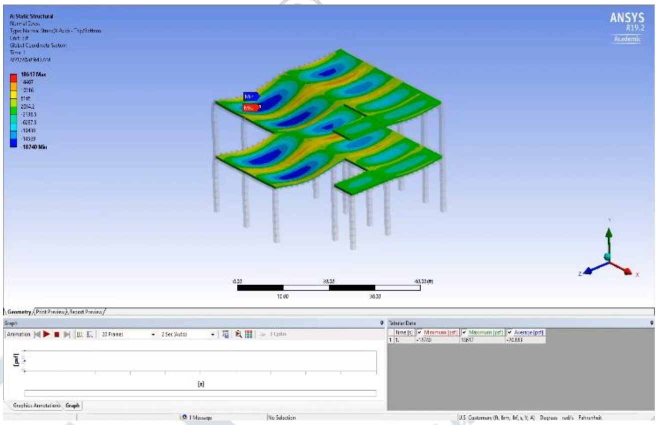


Fig.8 Normal Stress

6. Shear Stress

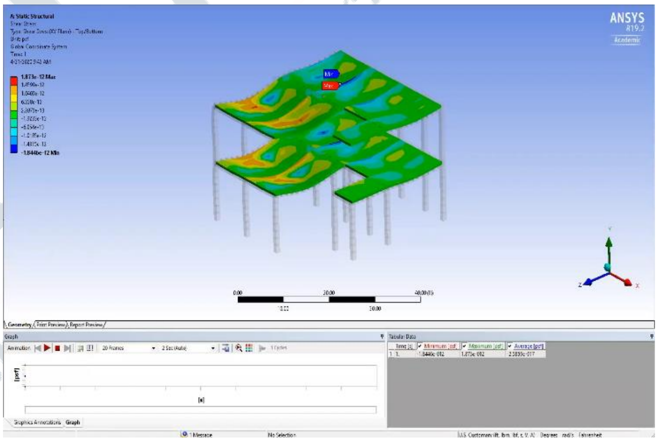


Fig.9 Shear Stresses

The results indicate that **shear stress concentrations are relatively low and uniformly distributed**, with slightly higher values observed near **edges, slab intersections, and support regions**

- Compared to normal stress, the **magnitude of shear stress is very small**, suggesting that shear effects are less critical for this structure under the given loading conditions.
- This analysis is important for verifying **structural stability and resistance to sliding or shear failure**, ensuring that the design meets safety and performance requirements.

IFERP
Explore Your Research Journey.
ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Volume 13 Issue 02 February 2026

7. Force Reaction

- The reaction forces are visualized using **vector arrows at the supports**, indicating both the **magnitude and direction of forces** transferred to the foundation.
- The results indicate that the **maximum reaction force occurs in the vertical (Y-axis) direction (~7.70 × 10^5 lbf)**, which corresponds to the applied gravity and structural loads.
- The **horizontal reaction forces (X and Z directions) are negligible**, showing that the structure is primarily subjected to vertical loading with minimal lateral effects.
- This analysis is essential for determining **foundation design requirements**, ensuring that supports can safely resist the loads and maintain overall structural stability.

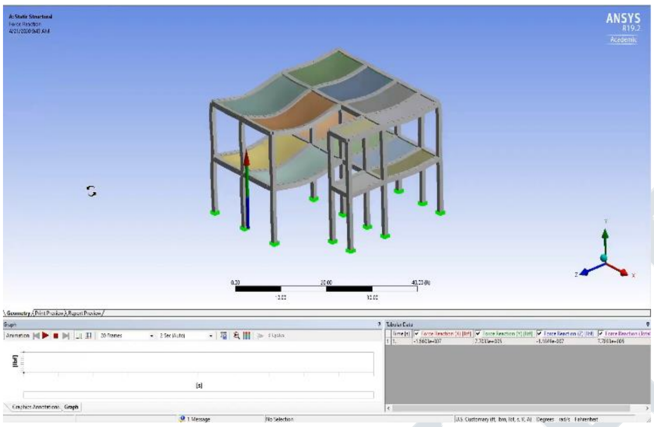


Fig.10 Force Reaction

8. Moment Reaction

Fig.11. Moment Reaction

- The results included deformation, stress distribution, and reaction forces, with particular focus on moment reactions at the supports. Significant moment values were observed along the Y and Z axes, indicating notable bending effects in the structure.
- Overall, the analysis demonstrates that the structure can sustain the applied loads, though areas with high stress and moment concentrations may require further design consideration for improved safety and performance.

9. Beam tools Direct Stress

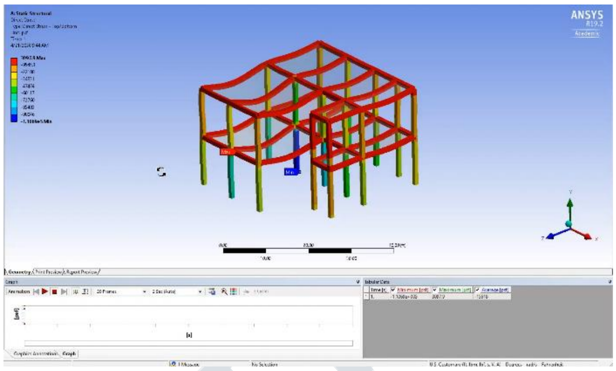


Fig.12 Direct Stress

- The displayed result is Direct Stress, visualized using a color contour where red indicates maximum stress and blue indicates minimum stress. The structure consists of beam and column elements subjected to loads such as gravity and possibly additional pressures, with fixed supports at the base.
- The results indicate a Maximum stress of approximately 3097.9 psf and a minimum of about −1.1069 × 10^5 psf, highlighting areas of tension and compression within the frame. This type of analysis helps engineers evaluate structural performance and identify critical regions that may require design optimization or reinforcement.

10. Beam Tool Minimum Combine Stress

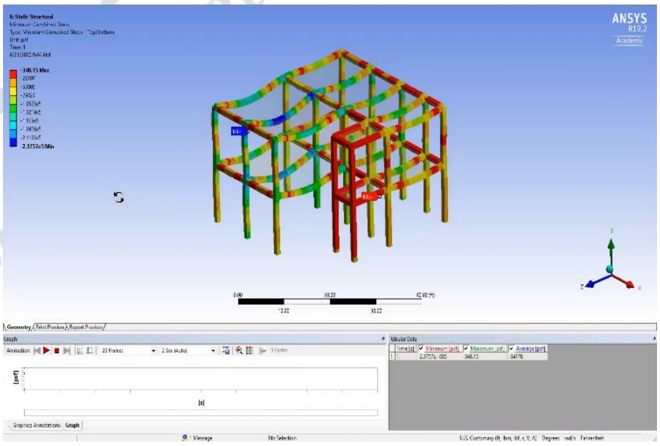


Fig.13 Minimum Combine Stress

- The static structural analysis shows the distribution of minimum combined stress across the frame structure under applied loads. The results indicate that thestructure is primarily subjected to compressive stresses, with a minimum value of −2.3757 × 10^5 psf, a maximum of −348.15 psf, and an average stress of −64,176 psf.
- Stress concentrations are observed mainly in the vertical members and joint regions, while the curved top beams exhibit lower stress levels, indicating better load distribution. Overall, the structure remains under compression, and critical regions may require

IFERP Explore Your Research Journey.

ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)

Volume 13 Issue 02 February 2026

reinforcement to prevent potential failure such as buckling.

11. Beam tool Maximum combine stress

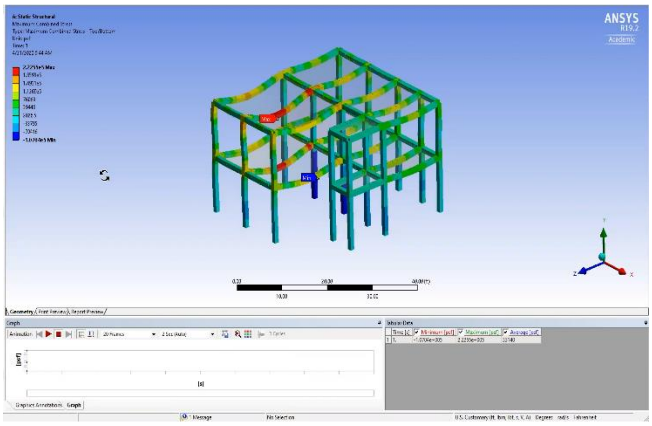


Fig.14. Maximum Combine Stress

- The result displayed is the maximum combined stress distribution, with colors indicating stress levels across the structured representing the highest stress and blue the lowest.
- The model consists of interconnected beams forming a multi-level frame, likely representing a building or industrial support structure. Fixed supports and loads (such as gravity or pressure) have been applied, and the simulation calculates how the structure responds.
- The highlighted Max and Min points indicate the locations of peak and minimum stress, helping engineers identify critical areas that may require design improvements for safety and stability.

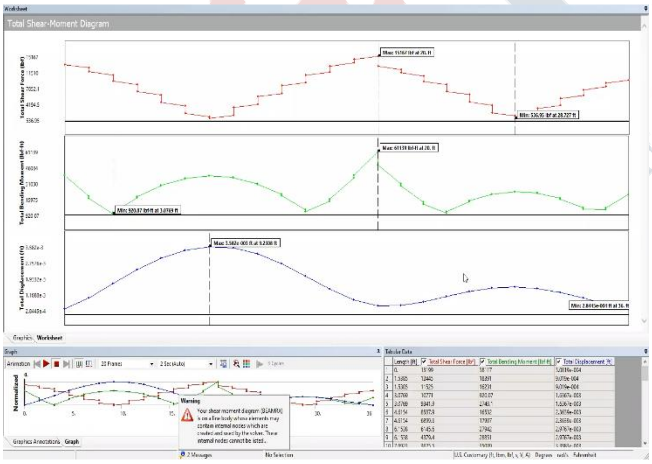


Fig.15 Total Shear Moment Diagram

IV. RESULT

Combined Structural Analysis Results Table

Category	Parameter	Maximum Value	Location (Max)	Minimum Value	Location (Min)	Interpretation
Shear-Moment	Total Shear Force	15167	20 ft	536.95	28.727 ft	Shear along selected beam/path
Stress	Maximum Combined Stress	2.2255×10^5	–	-1.0704 10^5	–	Highest overall combined stress in frame
Stress	Minimum Combined Stress	-348.15	–	-2.3775×10^5	28.727 ft	High compressive stress zones
Stress	Direct Stress	30979	–	-1.1069×10^5	–	Axial/direct stress in members
Stress	Maximum Principal Stress	29403	–	~0	–	Tensile-dominant zones (slab/beam top/bottom)
Stress	Maximum Principal Stress	29403	–	–	–	Tensile-dominant zones (slab/beam top/bottom)

V. CONCLUSION

The present study investigates the structural behaviour of a building under various loading conditions using ANSYS. The results indicate that shear force varies significantly along the beam, with a maximum value of **15167 lbf**, highlighting critical sections. Stress analysis shows high combined stresses in the structural frame, especially at beam-column joints. Both compressive and tensile stresses are observed, indicating the need for balanced structural design. Maximum principal stress values suggest that beams and slabs are more prone to tensile cracking. The study also reveals that load combinations greatly influence structural performance. Critical regions are mainly located at joints and support zones. Proper reinforcement detailing is essential to ensure safety and durability. Overall, ANSYS proves to be a reliable tool for accurate structural analysis and design optimization.

REFERENCES

[1] [1] N. J. Jain and S. S. Mishra, Comprehensive analysis of structural beams: Theoretical calculations and simulation analysis using ANSYS," *Materials Research Proceedings*, 2025.

[2] [2] S. G. Morkhade and L. M. Gupta, Experimental and parametric study on steel beams with web openings," *International Journal of Advanced Structural Engineering*, vol. 7, no. 3, pp. 249 261, 2015.

[3] [3] Finite element analysis of RC buildings subjected to blast loading, *Ain Shams Engineering Journal*, 2022.

[4] [4] Finite element analysis of beam with different loading condition, *Materials Today: Proceedings*, 2020.

[5] [5] B. Venkatesan et al., Finite Element Analysis for Beam-Column Joint subjected to cyclic loading using ANSYS, *Circuits and Systems Journal*, 2016.

[6] [6] Evaluation of structural performance of T-section beams using numerical analysis, *Frontiers in Built Environment*, 2024.

[7] [7] Bureau of Indian Standards, *IS 456: Plain and Reinforced Concrete – Code of Practice*, New Delhi, India.

[8] [8] Bureau of Indian Standards, *IS 875 (Part 1-5): Code of Practice for Design Loads (Other than Earthquake) for Buildings and Structures*, New Delhi, India.IS

[9]