Elastic-Plastic Analysis of Steel Frame Using ANSYS: Finite Element Method

Ashish Raj
Indian Institute of Technology, Dhanbad, Jharkhand, India

Prof. Prosanta Khan
Professor, Indian Institute of Technology, Dhanbad, Jharkhand, India

Abstract

This study investigates the elastic–plastic behaviour of a G+7, three-bay steel moment-resisting frame to evaluate how bending moments, lateral stiffness, and hinge mechanisms evolve under gravity and lateral loads. The structure, with a total height of 24 m and width of 18 m (three bays of 6 m each), is modelled in ANSYS Workbench using BEAM188 elements to capture both axial and flexural responses. ISMB 350 sections are assigned to all beams and columns, with material properties defined by an elastic modulus of 2×10ˆ5MPa, yield strength of 250 MPa, tangent modulus of 3000 MPa, and Poisson's ratio of 0.3. The steel material follows a bilinear isotropic hardening model to simulate post-yield ductility, while geometric nonlinearity is incorporated to account for second-order (P–Δ) effects and large deflections. Linear static analysis under gravity and lateral loads of 100 N in both global X and Z directions results in a maximum roof displacement of 1.78×10^{-3} m, which is far below the 48 mm serviceability limit, confirming elastic performance under service-level loading. The deformation contours reveal symmetric stiffness distribution, and the bending moment diagram shows a gradient from 23.46 kN-m at the top-storey joints to zero at the base, indicating efficient load transfer and base anchorage. This distribution suggests that, with increased lateral demand, plastic hinges will initiate at upper joints and propagate downward, leading to stable energy dissipation. The anticipated nonlinear pushover analysis is expected to yield ductility factors of 3–4 and roof drift ratios of 1.5–2.0%, demonstrating that elastic–plastic finite element analysis provides a more realistic and reliable prediction of stiffness degradation, ductility, and collapse behaviour in multi-storey steel frames.

Keywords

Elastic–plastic analysis, finite element analysis, Pushover analysis, Plastic hinge formation, ANSYS Workbench.