Parametric Analysis of Prestressed Concrete Beams with the Finite Element Method

B. Bebani *

Institute of Building Engineering, Poznan University of Technology, Poznan, Poland

M. Kuczma

Institute of Building Engineering, Poznan University of Technology, Poznan, Poland

K. Ziopaja

Institute of Building Engineering, Poznan University of Technology, Poznan, Poland

Abstract:

This study presents a numerical analysis of a prefabricated prestressed concrete beam with a span length of 4600 mm, cross-section of 240 x 400 mm, reinforced with a single 7-wire strand with a diameter of 15.2 mm, subjected to a three-point bending test. In order to ensure the propriety and reliability of the numerical model of the studied beam, a parametric analysis of the influence of selected model components on the determined results was carried out. Using the finite element method (FEM) software Abaqus/CAE by Simulia, the following parameters were analyzed: a) the effect of mesh size and meshing method, b) the type and shape of elements, c) the method of transferring the prestressing force, d) the effect of passive reinforcement. To ensure the high precision and reliability of the model, numerical results were compared and contrasted with analytical solutions, assessing the sensitivity of these factors. The final results presented in the paper demonstrate a strong agreement between numerical simulations and analytical calculations for the prestressed concrete beam, encompassing the deflection and stresses in both the elastic and inelastic ranges.

Keywords:

Prestressed concrete beams, finite element analysis, parametric and numerical analysis, analytical solution, meshing strategy.