



ABSTRACT

Massive concrete structures are integral to infrastructure development but their intricate thermo-mechanical behavior presents significant challenges. This research focuses on, conducting a comprehensive thermo-mechanical analysis of a pier head, using ABAQUS software. The study is organized into three phases to understand the temperature variations during cement hydration and the subsequent impact on structural integrity.

The methodology incorporates thermal analysis to replicate temperature fluctuations during cement hydration, validated through a mesh convergence study and comparison with field data. Additionally, mechanical analysis is employed, considering concrete properties, temperature changes, and construction phases. Nonlinear material behavior and contact interactions between layers are incorporated for a more accurate simulation, aiming to address the complexities of real conditions.

The results confirm the effectiveness of employing a multilayer system to regulate temperatures and minimize thermal stress-induced cracking. By conducting thermal and mechanical analysis, specific points within the pier head are specified as potential locations for existing internal cracks, as evidenced by the maximum difference in temperature and, the correlation between thermal stresses and concrete tensile strength. This research provides valuable contributions to mass concrete assessment, highlighting critical stress zones, and providing predictive capabilities regarding crack formation within specific temporal and spatial parameters.

Keywords: ABAQUS, Mass Concrete, Finite Element Analysis, Thermo-Mechanical Analysis, Cement Hydration Effect.