

Numerical Simulation of the Bond Behavior Between Concrete and Steel Reinforcing Bars in Specialty Concrete

Authors : Camille A. Issa, Omar Masri

Abstract : In the study, the commercial finite element software Abaqus was used to develop a three-dimensional nonlinear finite element model capable of simulating the pull-out test of reinforcing bars from underwater concrete. The results of thirty-two pull-out tests that have different parameters were implemented in the software to study the effect of the concrete cover, the bar size, the use of stirrups, and the compressive strength of concrete. The interaction properties used in the model provided accurate results in comparison with the experimental bond-slip results, thus the model has successfully simulated the pull-out test. The results of the finite element model are used to better understand and visualize the distribution of stresses in each component of the model, and to study the effect of the various parameters used in this study including the role of the stirrups in preventing the stress from reaching to the sides of the specimens.

Keywords : pull-out test, bond strength, underwater concrete, nonlinear finite element analysis, abaqus

Conference Title : ICBSE 2015 : International Conference on Building Science and Engineering

Conference Location : New York, USA

Conference Dates : June 04-05, 2015