

Modelling crack propagation in RC beam-column joints

ABSTRACT

Accurate modelling is required to estimate crack propagation in a beam–column joint. In this study, a numerical method is developed to model crack propagation and failure loading in a beam–column joint under static load. To realize this objective, a four-node, thin-layer interface element is produced to model the fracture process zone and crack propagation. Moreover, the fracture criterion for determining the growth of a crack based on the release rate of strain energy is established. To validate the present model, ABAQUS software is used to simulate crack propagation by conventional cohesive elements. The numerical results obtained are extremely close to the experimental results within an accuracy level ranging from 4.3% to 6.7%. Meanwhile, the ABAQUS software data and the experimental data are predicted at a margin of error ranging from 12.4% to 16%.

Keyword: ABAQUS; Crack propagation; Interface element; Joint