عنوان مقاله:

3D Modeling and Evaluation of Cracked Specimen by Applying ABAQUS Software

محل انتشار:

چهارمین کنگره بین المللی مهندسی برق، کامپیوتر و مکانیک (سال: 1399)

تعداد صفحات اصل مقاله: 6

نویسندگان:

Zohreh Shirazi - *PhD student, Faculty of Mechanical Engineering, Iran University of Science and Technology, Tehran, IRAN*

Ehsan Anbarzadeh - *PhD student, Faculty of Mechanical Engineering, Iran University of Science and Technology, Tehran, IRAN*

خلاصه مقاله:

The important issue in solving partial differential equations is to achieve a simple equation that is numerically stable. Due to complicated geometry, abstruse behavior of matter, boundary condition, and various loading in problems it is arduous to get a precise solution. The approximate solution with acceptable accuracy to solving these problems is an enormous evolvement. There are various methods with advantages and disadvantages for this matter that the finite element method is one of the best them. One of the most significant factors in fracture mechanics is the stress intensity coefficient. It is used to anticipate crack growth and the remaining life of the cracked member. The stress intensity factor expresses the stress close to the crack tip. One of the effective items on the stress intensity coefficient is the crack length that is examined by researchers in various methods. In this investigation, a cracked sample was studied and simulated by using ABAQUS software. Finally, the results of analytical and numerical solutions were .compared and an admissible convergence was observed

کلمات کلیدی:

Crack growth, Stress intensity factor, 3D modeling, ABAQUS software