

Abstract

In this study, two programs are developed. one is the finite element 2-D program with extrapolation methods (displacement and stress) and the other is a graphical user interface in matlab to calculate the stress intensity factor (K_I) with the effect of domain loading (rotational loading, thermal loading, pressure loading and combined loading) on fracture mechanism.

The new idea in this work is implemented by using a graphical user interface in matlab (GUI). A GUI was successfully developed using MATLAB programming language to solve the stress intensity factor(K_I) for five cases. These cases have been selected for disc, tube and cylinder with different positions of crack to evaluate opening mode (K_I). As a part of analysis of linear elastic fracture mechanics (LEFM).

The results revealed that agreement with closed-form solutions and this proves that the finite element method can be used as an effective tool for the analysis of domain loading and combined loading conditions. The stress and displacement method give accurate results; however it requires preparation of a suitable finite element mesh in the crack front, worth mention is that the mesh a near the crack is worked refinement to obtain more accuracy in results. The thermal loading have more effects on the calculations than another loading (pressure and rotational loading).