PAPER • OPEN ACCESS

Computational two-dimensional modeling of the stress intensity factor in a cracked metallic material

To cite this article: J E Rolón *et al* 2016 *J. Phys.: Conf. Ser.* **687** 012070

View the article online for updates and enhancements.

Computational two-dimensional modeling of the stress intensity factor in a cracked metallic material

J E Rolón[1], **E D Cendales**[1] **and I M Cruz**[2]

[1] Fundación Universitaria Los Libertadores, Bogotá, Colombia.
[2] Universidad ECCI, Bogotá, Colombia.

E-mail: `jerolonr@libertadores.edu.co` , `icruzb@ecci.edu.co`

Abstract. Cracking of metallic engineering materials is of great importance due cost of replacing mechanical elements cracked and the danger of sudden structural failure of these elements. One of the most important parameters during consideration of the mechanical behavior of machine elements having cracking and that are subject to various stress conditions is the stress intensity factor near the crack tip called factor Kic. In this paper a computational model is developed for the direct assessment of stress concentration factor near to the crack tip and compared with the results obtained in the literature in which other models have been established, which consider continuity of the displacement of the crack tip (XBEM). Based on this numerical approximation can be establish that computational XBEM method has greater accuracy in Kic values obtained than the model implemented by the method of finite elements for the virtual nodal displacement through plateau function.

1. Introduction
Geometrical discontinuities such as cracks or notches are found usually in different structures and machine elements [1]. These imperfections in the material acts as stress concentrators so that the material is subject to lower values of stress than the yield stress can increase the size of the crack to reach the critical size at which the failure of the material occurs suddenly [2,3]. Because this condition, fracture study in materials represents a fundamental area the enginnering analysis and design of structures [4,5].

2. Computational implementation
In this article for the construction of the mesh tetrahedral elements were used, because this type of elements coupled better to the geometry near to the tip of the crack, compared with hexahedral elements of similar sizes. On the tip of the crack mesh refinement was performed, this refinement has the shape of circular rings, which permits to find more realistic values around the tip of the crack. All algorithms used in the mesh are available in the software used to resolve the state of the stress in each node. According to this computational implementation developed in this research, it was not required the second mesh having a size of crack increased. It was enough to recalculate the stiffness matrix considering new nodal coordinates.

3. Results
During the development of finite element analysis for the case illustrated in this paper, it was established the comparison with results from other investigations in order to validate the results

obtained by implementing performed. Figure 1 shows the results obtained for the simulations in which number of degrees of freedom of the nodes of the mesh in the tip of the crack change and the simulation was established for different values of this parameter. The same figure illustrates the expected behavior of parameter K_I according to a study by Rooke et al. [6]. Also the results obtained by Alatawi et al. [7], are presented in order to make comparison due they developed computer simulations to evaluate the relative stress intensity factor K_I/K_o, for a similar geometry that was used in the present study.

Based on the results shown, it is observed that the results obtained by Alatawi et al. [7], present convergence to increase the number of degrees of freedom in the nearby area the crack tip when a number of degrees of freedom greater than 150 was used, so that this implementation allows direct evaluation of parameter K_I. Moreover, Figure 1 shows that the results obtained in present study for the direct evaluation the stress intensity factor through computational implementation of FEM have relative convergence regarding the results established by implementing of XBEM. However considerable discrepancy is observed when the number of degrees of freedom is less than 200, approximately.
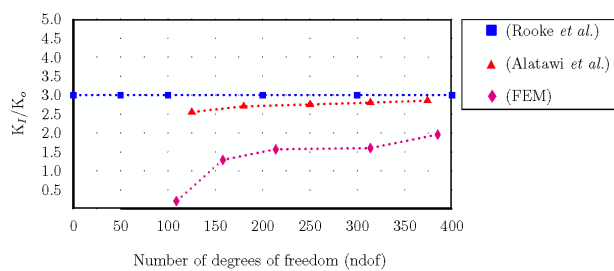
Figure 1. Variation of standardized stress intensity factor as a function of dnof.
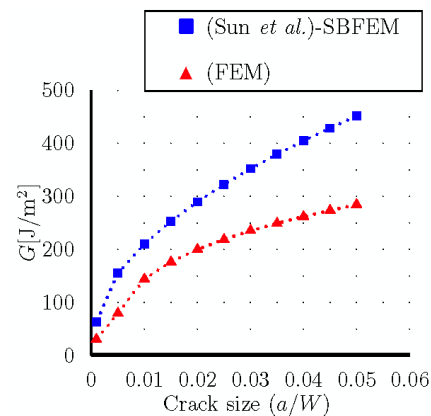

Figure 2. Variation of the driving force for crack growth as a function of the relative crack size.

Figure 2 shows the variation of the driving force (G) for the increase of the crack size obtained by Sun et al. [8], through the implementation of the method SBFEM for relative stress intensity factor K_I/K_o, for the same geometry. Sun et al., report that there is convergence between the values for G parameter obtained by computational simulation which was calculated by the Equation 1 in order to validate the simulation. From Figure 2 can be established that the FEM computational implementation developed in the present work shows subestrimacion for the driving force during advancing of the crack respect to the results obtained by computational implementation based SBFEM.

$$G = -\frac{1}{l} \int_0^l \Delta\mathbf{u}(\varepsilon_{crack}(r,\theta),\eta(\theta))^T \sigma(\varepsilon_{initial}(r,\theta),\eta(\theta)) \tag{1}$$

The behavior of stress was evaluated by FEM simulation and were subsequently compared with the results reported by Sun et al. [8]. As Figure 3 illustrates, the values for the same parameter obtained in this study through FEM simulation present overestimation of the normal stresses on the front of crack $(\theta = 0)$ for crack sizes smaller than 0.01mm. However, for higher values there is a considerable difference between the two computational estimates, presenting underestimation of stresses at points near the crack tip.
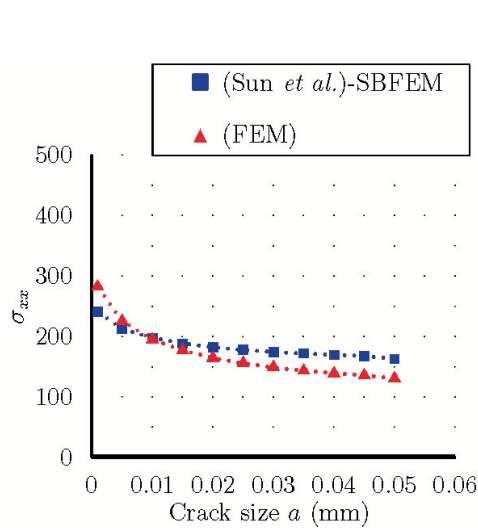
Figure 3. σ_{xx} in front of the crack ($\theta = 0$) depending on the crack size.
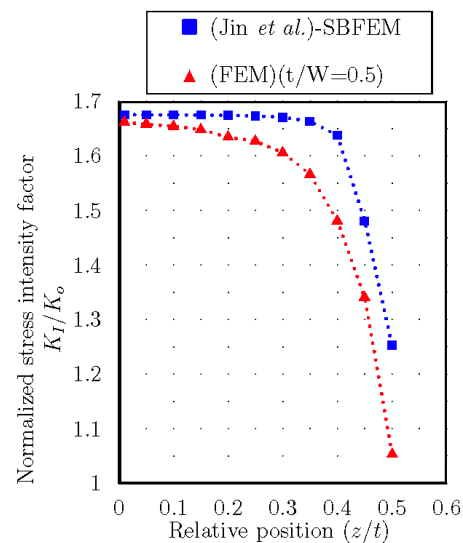
Figure 4. Variation of standardized stress intensity factor as a function of the relative position at the crack tip.

In order to evaluate the stress triaxiality in the crack tip the results of computational simulations obtained by Jin et al. [9] and the results obtained by simulation implemented in this study are presented in Figure 4. The results in both simulations illustrate a purely triaxial behavior at the center of the crack ($z/t = 0$) in which the parameter value K_I has the highest value. Moreover, in distant symmetrical positions at the crack tip ($z/t > 0$) this parameter decreases sharply to a minimum value as expected at the free surface of the plate in which the stress state has a distinctly a plane stress condition.

4. Conclusions

Finite element method has been implemented based closely on the stress intensity factors in linear elastic fracture mechanics theory. The implementation is able to evaluate SIFs directly without any requirement for aditional calculations for 2D simulations. Results shows convergency between results established using XBEM method and FEM method. However is evident that XBEM implementation present more accurancy than FEM method on the evaluation of SIFs and G (driving force). Nevertheles XBEM present higher complexity in the computational implementation for that reason FEM could be used as a first estimation for these parameters used in dimensioning and desing of mechanical systems.

References

[1] Williams M and Ellinger G 1953 *Welding Journal* **32** 498
[2] Sih G 1966 *International Journal of Fracture Mechanics* **2** 628
[3] Hull D and Bacon D 1953 *Introduction to dislocactions* (Woburn: Betterworth-Heinemann)
[4] Eshelby J 1975 *The Physics of Metals* **2** 10
[5] Westergaard H 1939 *Journal of Applied Mechanics* **6** 49
[6] Rooke D and Cartwright D 1976 *Compendium of Stress Intensity Factors* (London: Her Majesty's Stationery Office)
[7] Alatawi I and Trevelyan J 2015 *Engineering Analysis with Boundary Elements* **52** 56
[8] Sun Z, Ooi E and Song C 2015 *Engineering Fracture Mechanics* **134** 330
[9] Jin Z and Wang X 2013 *Theoretical and Applied Fracture Mechanics* **132** 110