International Journal of Soft Computing and Engineering (IJSCE)
ISSN: 2231-2307, Volume-5 Issue-5, November 2015

Finite Element Modelling of Mixed Mode Crack Propagation

Abdulnaser M. Alshoaibi

Abstract*— *This paper illustrates an algorithm for automatic simulations of crack propagations in 2D linear elastic finite element representation. The crack tip singularity and stress intensity factors around the crack tip are obtained by using the displacement extrapolation method. The crack propagation direction can be predicted by using the maximum circumferential stress theory criterion. The developed program has been examined with two different types of geometries namely: single edge cracked plate with one hole and double edge cracked plate with two holes with different types of loading. The results obtained by the current program have been assessed by comparing with the relevant works.

***Index Terms*— *Finite element method, crack trajectories, holes, stress intensity factors, adaptive mesh*.**

I. INTRODUCTION

The importance of fracture mechanics techniques for the analysis of structures containing cracks has increased considerably. The analysis of crack propagation as well as the failure prediction of structural components in engineering applications is important research subjects. In the last decade numerical analysis of fracture problems have become an effective way of approaching this problem due to the development of the computing capacity. Several methods for the numerical analysis of fracture problems have been developed. The finite element based methods are the more recurrent in the literature. The ranges of applicable fracture mechanics to study the mechanical behaviour of cracked materials subjected to an applied load are linear elastic fracture mechanics (LEFM) and elastic plastic fracture mechanics (EPFM). LEFM was originally developed to describe crack growth and fracture under essentially elastic conditions. It deals with only limited crack tip plasticity and can be used in damage tolerance analyses to describe the behaviour of cracks. The fundamental postulate of LEFM is that the crack behaviour is determined solely by the values of the stress intensity factors (SIFs) which are functions of the applied load and the geometry of the crack structure [1]. The factors define the stress field close to the crack tip of a new crack and provide fundamental information on how the crack will propagate. For isotropic materials, the near crack tip singular stresses and displacements near the crack tip can be determined [2]. Several methods have been proposed to numerically estimate the stress intensity factors using finite element method such as the displacement extrapolation technique [3], the *J*-integral [4] and the energy domain integral [5]. Among these methods, the displacement extrapolation technique is simplest and highly accurate. It is used when the singular elements are present at the crack tip. In the finite element fracture analysis, these special elements known as quarter-point elements are used to provide the polynomial elements representing stress and strain singularities near the crack tip. Nodal relaxation is frequently used to release nodes, one by one, in order to enable the crack tip to propagate through the mesh. In contrast, methods based on near-tip field fitting procedures require finer meshes to produce a good numerical representation of crack-tip fields. The most accurate methods being those based on nodal displacements, which are a primary output of the finite element program [6]. In adaptive mesh refinement, most analysts favour either the Delaunay technique or the advancing front method over other techniques when generating meshes due to the quality of the unstructured meshes generated [7]. The main advantage of the advancing front method is that it tends to produce nicely graded meshes and high quality triangles that are usually very close in shape to equilaterals. The boundary integrity is also preserved, since the discretisation of the domain boundary constitutes the initial front. Phongthanapanich and Dechaumphai [8] used a finite element method, with the adaptive Delaunay triangulation as mesh generator to analyze two-dimensional crack propagation problems. They described the Delaunay triangulation procedure consisting of mesh generation, node creation, mesh smoothing, and adaptive remeshing, all with object-oriented programming. They also used the displacement extrapolation method to determine the values of stress intensity factors for compact tension specimens, central cracked plates and single edge cracked plates for certain geometries only. Rashid [9] developed the arbitrary local mesh replacement method based on two distinct meshes. One that surrounded the propagating crack front and moved with it, and another that filled the rest of the domain. Bittencourt et al. [10] developed a strategy for quasi-automatic simulation of the propagation of arbitrary cracks in 2D using FRANC2D. Bouchard et al. [11] introduced an interesting remeshing technique to model crack propagation using the discrete crack approach. The main objective of this work is to determine the effect of holes to crack propagation trajectory. The computational code is written in FORTRAN programming language for finite element analysis calculation processes, which is based on load and displacement control for linear-elastic crack propagation modeling. The mesh for the finite elements is the unstructured type; generated using the advancing front method. The global h-type adaptive mesh is adopted based on the norm stress error estimator. The quarter-point singular elements are uniformly generated around the crack tip in the form of a rosette. The displacement extrapolation technique used in the calculation is explained. The advantage of this method is that, it is well suited for multiple cracks and it can be performed faster.

Revised Version Manuscript Received on November 09, 2015.

Dr. Author Abdulnaser M. Alshoaibi, Department of Mechanical Engineering, Jazan University, Jazan, P.O. Box 706, Kingdom of Saudi Arabia.

The present software code has been developed to enable the user to determine 2D-cracks under mixed mode loading and, with the aid of automatic adaptive mesh finite element, to analyst fatigue crack path lifetimes. This program is written in FORTRAN language. The finite element calculations provided by this software produce results comparable to the current available commercial software.

II. CRACK TRAJECTORY SIMULATION METHOD

There are four important components in a crack trajectory simulation using the finite element analysis with adaptive mesh, i.e. the mesh optimization algorithm, the crack criterion, the direction criterion and the crack propagation technique. In this part, the technique used to determine each component of the crack propagation trajectory is briefly described.

2.1 Adaptive Mesh Refinement

The mesh refinement can be controlled by the characteristic size of each element, predicted according to the error estimator. This initial model is solved by an incremental theory using von Mises yield criterion. After the solution has converged at the end of each load step, the solution errors are estimated. If the error at some point in the model exceeds a specified maximum error, the incremental analysis is interrupted and a new finite element model is constructed. The system decides automatically where to refine the mesh. If it is necessary, the system refines the mesh considering the initial boundary conditions. After the new mesh is generated, the solution variables (displacements, stresses, strains, etc.) are mapped from the old mesh to the new mesh. The analysis is then restarted from the current step and it is continued until the errors again become larger than the specified limit. In the final analysis or in each step, the user can visualize the responses using a graphics post-processor. The details description of the procedure can be referred to Alshoaibi [12]. The strategy used to refine the mesh during the analysis process is adopted from Zienkiewicz [13] and Alshoaibi [14].

2.2 The Crack Criterion

The crack criterion is used to determine when the cracks start to initiate. In LEFM, the stress intensity factor (SIF), *K* is usually used as a fracture criterion. In this paper, the displacement extrapolation method [8] is used to calculate the stress intensity factors as follow:

$$K_I = \frac{E}{3(1+\upsilon)(1+\kappa)}\sqrt{\frac{2\pi}{L}}\left[4(v_b - v_d) - \frac{(v_c - v_e)}{2}\right] \quad (1)$$

$$K_{II} = \frac{E}{3(1+\upsilon)(1+\kappa)}\sqrt{\frac{2\pi}{L}}\left[4(u_b - u_d) - \frac{(u_c - u_e)}{2}\right] \quad (2)$$

where *E* is the modulus of elasticity, υ is the Poisson's ratio, κ is the elastic parameter defined by (3-4υ) for plane strain and (3-υ)/(1+υ) for a plane stress problem and *L* is the element. The *u* and *v* are the displacement components in the *x* and *y* directions, respectively; the subscripts indicate their position as shown in Fig. 1.

2.3 The Crack Direction

The third component of a crack propagation simulation using the finite element analysis with adaptive mesh is the direction criterion. The direction criterion is used to decide where the crack propagates. There are several methods used to predict the direction of a crack trajectory such as the maximum circumferential stress theory, the maximum energy release rate theory and the minimum strain energy density theory. In the maximum circumferential stress theory, the direction of the crack propagation θ is computed from:

$$K_I \sin\theta + K_{II}(3\cos\theta - 1) = 0 \quad (3)$$

Analyzing Eq. (3) for the two pure modes, it is found that for pure mode I, K_{II} =0, $K_I \sin\theta = 0$ and $\theta = 0^\circ$, and for pure mode II, K_I=0 and $\theta = \pm 70.5^\circ$.These values of θ are the extreme values of the crack propagation angles. The intermediary values are found by solving Eq. (4) for θ considering the mixed mode, resulting in:

$$\theta_0 = \pm\cos^{-1}\left\{\frac{3K_{II}^2 + K_I\sqrt{K_I^2 + 8K_{II}^2}}{K_I^2 + 9K_{II}^2}\right\} \quad (4)$$

The maximum circumferential stress criterion determines that the crack extension should occur in the direction that maximizes the circumferential stress in the region close to the crack tip [15]. In order to ensure that the opening stress associated with the crack direction of the crack extension is maximum, the sign of θ_0 should be opposite to the sign of K_{II} . The two possibilities are illustrated in Fig. 2. The criterion for the crack to propagate from the crack tip is based on the material toughness, K_C . If the calculated stress intensity factor, $K_I \geq K_C$ then the crack will propagate to the direction θ_0 expressed by Eq. (8). The crack increment length Δa is taken as 20%-50% of the initial crack length, depending on the ratio of K_{II}/K_I ratio [8].

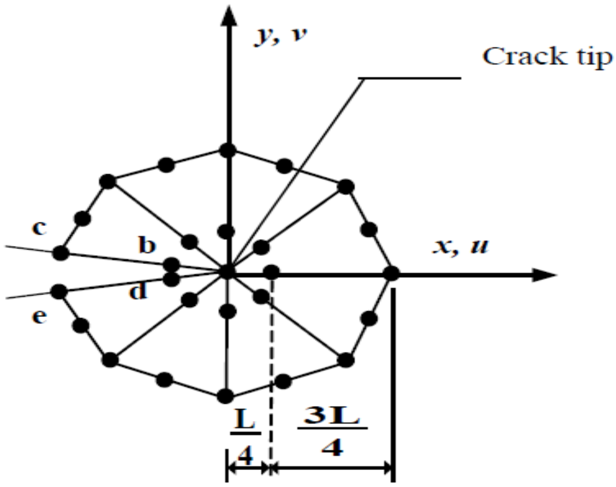


Fig. 1. Quarter-point triangular elements around the crack tip

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

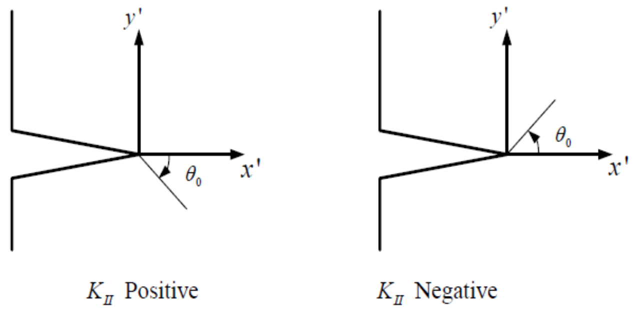


Fig. 2. Sign of the propagation angle

2.4. Crack propagation mechanism with mesh generation

Initial mesh generation is a key step in fracture mechanics problems. 2D cracked geometries are complex to mesh because a very fine mesh is needed near the crack tip while a coarse mesh up to two orders of magnitude larger suffices far from crack tip. To this end, linear elastic problems are considered, but the methodology can be extended to more complex models such as elastic–plastic or dynamic fracture mechanics. Crack growth is computed discretely by using a finite element model for each new crack length step.

At each step of the propagation, a finite element model is defined. In the first step, the model is provided by the analysis as input for the simulation. Then, in the following steps, the model is provided by the output of the algorithm itself in the previous steps. In each step, when the crack propagates, the elements inside the geometry gets deleted and rebuilt again with adaptive strategy and updated for the next cycle of propagation.

The remeshing sequence is as follows. The crack geometry is first identified by the user in the initial geometrical configuration (Fig. 3a). The crack is then extended to a new tip location by the addition of a new segment. All affected elements are then removed by inserting a new crack (Fig. 3b) and the crack geometry is updated. Then, the quarter-point singular elements are placed around the tip in a uniform rosette pattern (Fig. 3c). Finally, the whole domain is remeshed (Fig. 3d). After the creation of the finite element mesh for the new configuration, the model is ready for a new cycle of propagation.

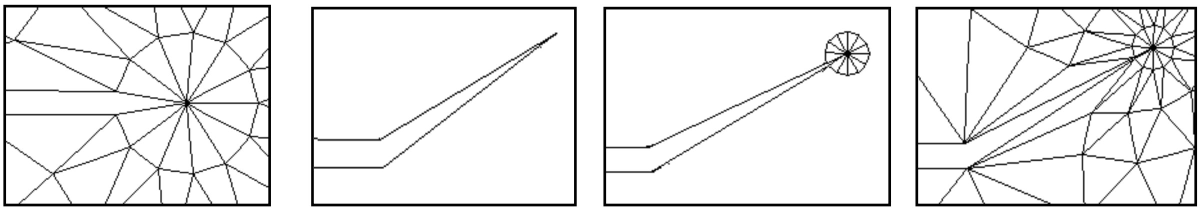


Fig. 3. Sequential procedure for geometry updating and remeshing

III. NUMERICAL ANALYSIS AND VALIDATION

Two well-known plate geometries, namely the single edge cracked plate with one hole and the double edge cracked plate with two holes. The modulus of elasticity and Poisson's ratio of the specimens were taken as 70 GPa and 0.3, respectively. All dimensions of the geometry were in mm.

3.1 Single edge cracked plate with one hole

Fig. 4 shows the geometry of the single edge cracked plate with one hole and its final adaptive mesh.

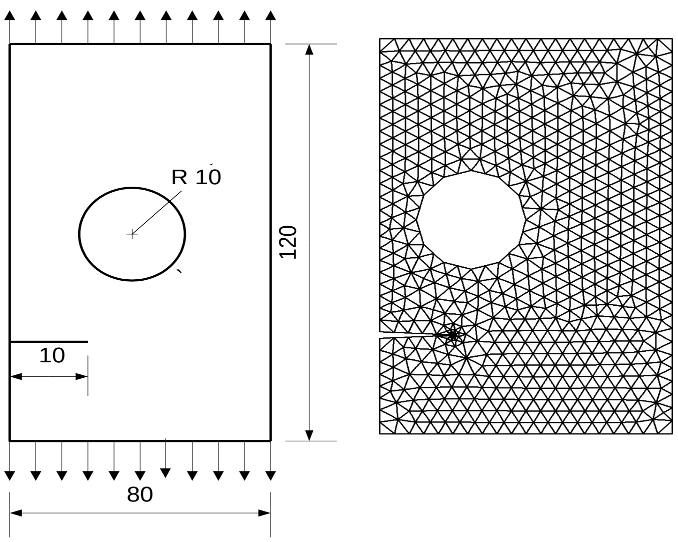
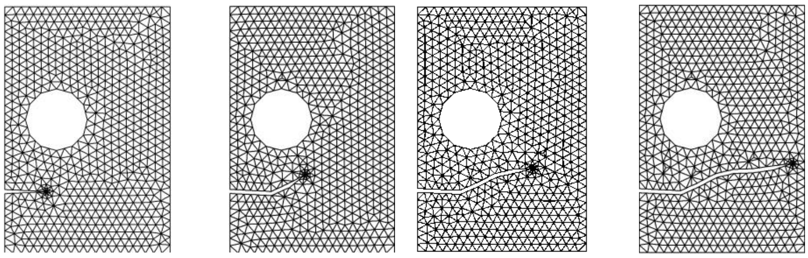


Fig. 4. Problem statement and the final mesh of the initial crack for the single edge cracked plate with one hole

The plate was simply fixed at the bottom edge and the load distributed uniformly on the top edge. The crack would move in a straight path if there was no hole at the plate for mode I loading. However, due to the presence of the hole, the crack did not follow a straight line path, but curved towards the hole as shown in Fig. 5.

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

This was due to the stress concentration effect; cracks are likely to initiate at a hole boundary. Once the crack tip has moved beyond the hole, the crack reoriented horizontally in the mode I loading as shown in Fig. 5d. Overall, the presence of holes in the plate disturbed the stress or strain fields and provided interesting curvilinear crack trajectories on each specimen. The behavior of the crack propagation in plane stress was almost the same as obtained by [9].



(a) (b) (c) (d)

Fig. 5. Crack Growth Trajectory for Single Edge Cracked Plate with One Hole

Fig. 6 shows the configuration corresponding to the final step of crack propagation. The results proof that for a horizontal crack, the crack direction is dependent on the initial crack location. An initial location close to the hole could cause the crack path to intersect the hole, whereas the crack path is more nearly straight for initial locations that are more remote from the hole.

3.2 Double edge cracked plate with two holes

The geometry and the final adaptive mesh of this specimen are shown in Fig. 7. The plate was simply fixed at the bottom edge and subjected to a uniform distributed load at the upper edge. Each crack was found to grow towards the nearest hole as seen in, Fig. 8a. Then, the crack reoriented horizontally since the cracks have modified the stress distribution at each other's tip as seen in, Fig. 8c. Eventually, the cracks were attracted again by the opposite holes and curved towards the holes, (Fig. 8d). The result illustrated that the crack propagated with the same trajectory. This proofed Bouchard et al.'s [16] statement which said that, two cracks can propagate with the same length if they are symmetric. However, for multiple cracks, some authors suggested that the cracks propagate one after the other according to the values of the stress intensity factors [15]. As can be seen in Fig. 8d, a slight convergence of the crack paths could be detected at the areas close to the holes. The present work simulated the crack propagation for the double edge crack plate simultaneously. Each crack tip had its own refinement with its own concentric mesh that did not depend on each other. The stress distribution is also presented in terms of the Von Mises equivalent stress field, as in Fig. 9. The Figure clearly shows the effects of each other's stresses. Comparison between the present results and those obtained by Bouchard et al. [11] show that there are good agreement for crack trajectory and stress distribution. Experimental work has been also conducted for this geometry to get a further validation for the computational method as shown in Fig. 10.

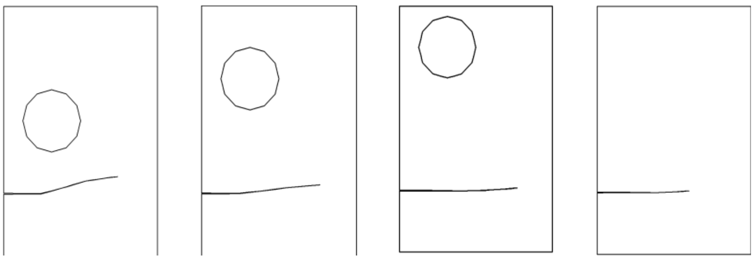


Fig. 6. Final configuration of crack propagation for different locations of hole

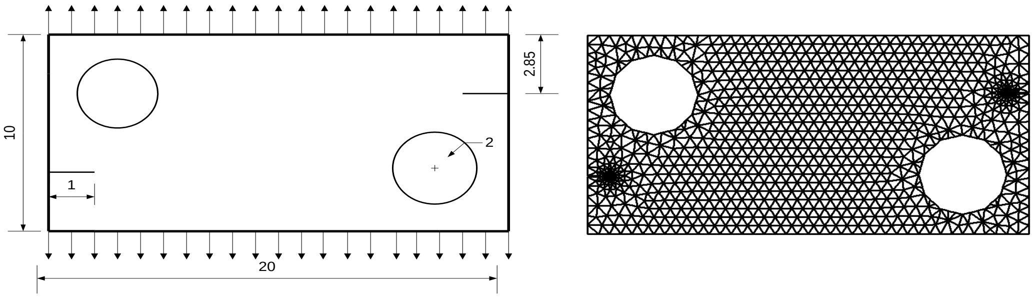
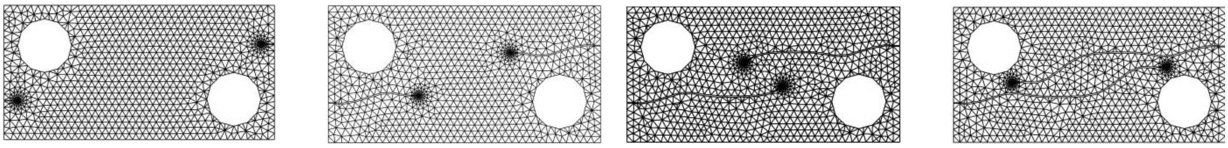


Fig. 7. Problem statement and the final mesh of the initial crack for the double edge cracked plate with two holes

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication



(a) (b) (c) (d)

Fig. 8. Crack growth trajectory for the double edge cracked plate with two holes

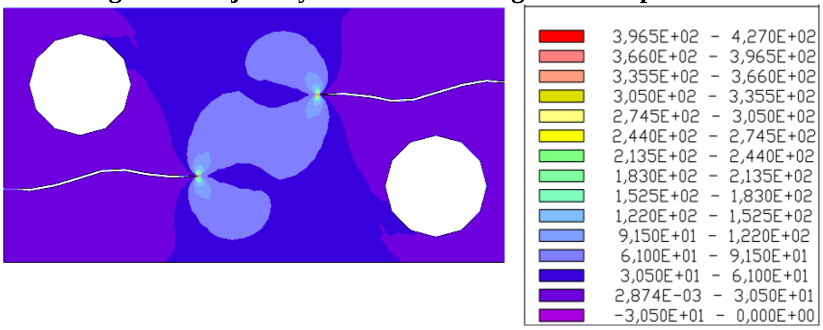


Fig. 9. Von Mises Stress field distribution at 8th step of crack propagation

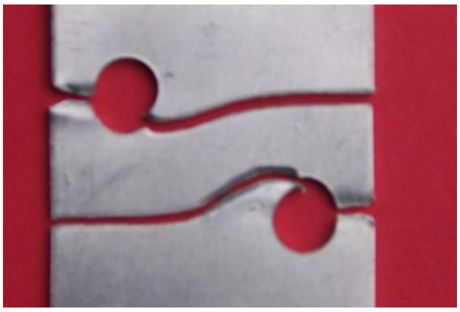
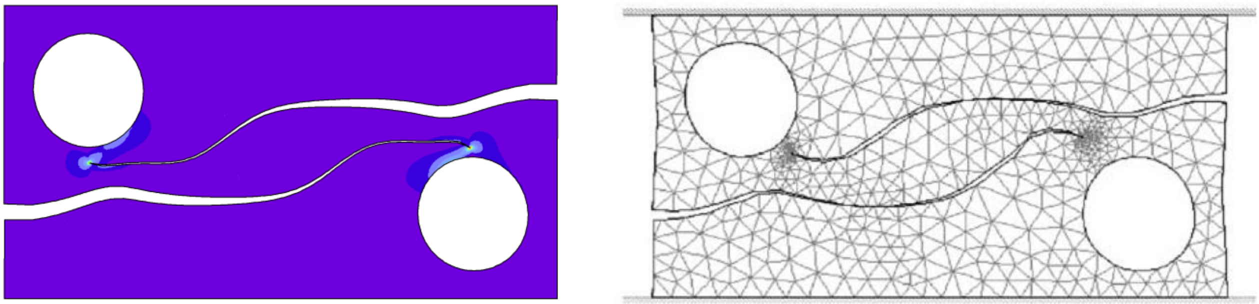


Fig. 10 Experimental cracks propagation trajectories



(a) (b)

Fig. 11. Crack propagation simulation for the double edge cracked with two holes specimen (a) present and (b) Bouchard *et al.* [11].

Figure (11.a) shows the contour of final step of crack propagation with the maximum principal stress distribution with deformation which is reassemble to the numerical work obtained by Bouchard et al. [11] (Figure 11.b).

IV. CONCLUSION

The adaptive finite element method using advancing front method for crack propagation analysis and stress intensity factors prediction have been presented. The norm stress error was taken as a posterior estimator for the *h*-type adaptive refinement. The strategy has been used successfully to simulate the propagation of cracks in plate specimens with holes. The presence of holes in the plates disturbed the stress and strain fields providing interesting crack trajectories. The crack simulations for mode I and mixed mode cases showed the acceptable crack path predictions. The results of the assessments strongly indicated that the finite element simulation for two-dimensional linear elastic fracture mechanics problems has been successfully employed.

REFERENCES

1. Zhu, W.X. and Smith, D.J, On The Use of Displacement Extrapolation to Obtain Crack Tip Singular Stresses and Stress Intensity Factors, Engineering Fracture Mechanics, vol. 5, pp. 391-400, 1995.
2. Isida, M, Effect Of Width And Length On Stress Intensity Factors Of Internally Cracked Plates Under Various Boundary Conditions, International Journal of Fracture, vol. 7, pp. 301-316, 1971.
3. Chan, S.K, Tuba, I.S and Wilson, W.K, On the Finite Element Method in Linear Fracture Mechanics, Engineering Fracture Mechanics, Vol. 2, pp.1–17, 1970.

4. Parks, D.M, A Stiffness Derivative Finite Element Technique for Determination of Crack Tip Stress Intensity Factors, International Journal of Fracture, vol. 10, pp. 487–502, 1974.
5. Moran, B and Shih, C.F, A General Treatment of Crack Tip Contour Integrals, Engineering Fracture Mechanics, vol. 35, pp. 295–310, 1987.
6. Guinea, G.V, Planan, J and Elices, M, K_I Evaluation By The Displacement Extrapolation Technique, Engineering Fracture Mechanics, vol. 66, pp. 243-255, 2000.
7. El-Hamalawi, A 2D Combined Advancing Front Delaunay Mesh Generation Scheme, Finite Element in Analysis and Design, vol. 40, pp. 967-989, 2004.
8. Phongthanapanich, S and Dechaumphai, P, Adaptive Delaunay Triangulation With Oject-Oriented Programming For Crack Propagation Analysis, Finite Element in Analysis and Design, vol. 40, pp. 1753-1771, 2004.
9. Rashid M.M, The arbitrary Local Mesh Replacement Method: An Alternative to Remeshing for Crack Propagation Analysis, Comput. Methods Appl. Mech. Engrg. Vol. 154 , pp. 133-150, 1998.
10. Bittencourt, T.N, Wawrzynek, P.A, A.R. Ingraffea and J.L. Sousa, Quasi-Automatic Simulation Of Crack Propagation For 2D LEFM Problems, Eng. Fract. Mech. Vol. 55, pp. 321–334, 1996.
11. Bouchard, P.O, Bay F and Chastel Y, Numerical Modeling Of Crack Propagation: Automatic Remeshing And Comparison Of Diffeent Criteria, Comput.Methods Appl. Mech. Engrg. Vol 192, pp. 3887–3908, 2003.
12. Alshoaibi M. Abdulnaser, Finite Element Procedures for the Numerical Simulation of Fatigue Crack Propagation under Mixed Mode Loading. Structural Engineering and Mechanics, vol. 35, No.3, pp.283-299, 2010.
13. Zienkiewicz, O.C., Taylor, R.L. and Zhu, J.Z. 2005. The finite element method: its basis and fundamentals. 6th Edition, Elsevier Butterworth-Heinemann.
14. Alshoaibi M. Abdulnaser, A. K. Ariffin and M.N. Almaghribi, 2009. Development of efficient Finite Element software of Crack Propagation simulation Using Adaptive Mesh Strategy. American Journal of Applied Science. Vol. 6, No. 4, pp: 661-666.
15. Erdogan F and Sih GC, On The Crack Extension In Plates Under Plane Loading And Transverse Shear, J Basic Eng. Vol. 85, pp. 519–27, 1963.
16. Bouchard, P.O, Bay, F, Chastel, Y and Tovena I. Crack Propagation Modeling Using an Advanced Remeshing Technique, Computer Methods Applied Mechanics Engineering, pp. 723-742, 2000.

AUTHORS PROFILE

Dr. Abdulnaser M. Alshoaibi, PhD Mechanical and Materials Engineering. Research work: Computational and Experimental Fracture Mechanics.