Key Engineering Materials Vols. 261-263 (2004) pp 687-692
Online available since 2004/Apr/15 at www.scientific.net

doi:10.4028/www.scientific.net/KEM.261-263.687

The Implementation of Inter-Element Model for Crack Growth Simulation

Ahmad Kamal Ariffin, Syifaul Huzni,
Nik Abdullah Nik Mohamed & Mohd. Jailani Mohd. Nor

Computational Mechanics Research Group
Department of Mechanical & Materials, Faculty of Engineering
Universiti Kebangsaan Malaysia
43600 Bangi, Selangor D.E., Malaysia

Keywords: adaptive mesh, inter-element crack model, crack propagation, CTOA

ABSTRACT

The implementation of inter-element model to simulate crack propagation by using finite element analysis with adaptive mesh is presented. An adaptive finite element mesh is applied to analyze two-dimension elastoplastic fracture during crack propagation. Displacement control approach and updated Lagrangean strategy are used to solve the non-linearity in geometry, material and boundary for plane stress crack problem. In the finite element analysis, remeshing process is based on stress error norm coupled with *h*-version mesh refinement to find an optimal mesh. The crack is modeled by splitting crack tip node and automatic remeshing calculated for each step of crack growth. Crack has been modeled to propagate through the inter-element in the mesh. The crack is free to propagates without predetermine path direction. Maximum principal normal stress criterion is used as the direction criteria. Several examples are presented to show the results of the implementation.

1. INTRODUCTION

The implementation of automatic adaptive mesh in finite element analysis is receiving much attention as such program allow the user to obtain finite element solution for different engineering problem within some prescribed accuracy. In fracture mechanics field of study, adaptive finite element has proved to be very well suited for the study of crack propagation [1-3].

There are two different strategies that widely used to model fracture phenomena using finite element analysis [4]. The first one called "damage model". This strategy describes the fracture phenomena using micro-mechanical models in order to represent the evolution of damage through material. The fully coupled models, damage and material behavior, the damage zone is represented by the stress fall in this region. By using this fracture model, the crack propagation is not represented in the mesh since the crack is taken into account through a sharp drop in the normal stresses in the damaged area. In the second strategy, called "splitting node", cracking is represented as a displacement discontinuity induced by the separation of the edges of a crack. This process will produce mesh distortion around the crack tip due to element splitting. To maintain the accuracy of finite element calculation around the crack tip, remeshing is therefore required for each step of crack propagates.

In this paper, the splitting node strategy through adaptive mesh of finite element analysis applied to simulate the crack propagation for two-dimensional elastoplastic condition. A computer codes have been developed using Fortran programming language for finite element analysis calculation process, which is based on displacement control. The program that developed consists of three processes i.e. non-linearity in geometry, material and boundary conditions. Finite element calculation also includes with adaptive mesh process, calculation of crack growth criterion process and crack propagation process.

2. ADAPTIVE MESH REFINEMENT

Generally, the smaller the mesh sizes in a finite element mesh, the more accurate the finite element approximate solution. However, reduction in the mesh size leads to greater computational effort. It is therefore more attractive to selectively refine the mesh, in areas where the error in the approximate solution is largest. This is referred to as adaptive mesh refinement, and requires the estimation of the error in the finite element solution.

Several author have been proposed a different error estimators that could be used in finite element analysis with adaptive mesh, see [1] and [5]. In this paper error estimator is used proposes by Zienkiewich and Zhu [6] which is based on stress error norm. The central idea of the estimator is to obtain a better approximation of the exact stresses by means of a projection process. The strategy used to refine the mesh during analysis process is adopted from [7].

3. CRACK GROWTH CRITERIA AND CRACK DIRECTION CRITERIA

Crack growth criteria and crack direction criteria are two important components in crack propagation simulation. Crack growth criteria is used to determine when the crack will start, and the direction criteria decide where the crack will propagates.

The aim of this paper is not to present the most relevant fracture criteria or direction criteria but to show how to numerically initiate and propagate a crack through a mesh. Therefore, CTOA (crack tip opening angle) criterion is used to characterize crack initiation. This criterion is widely used in thin elastoplastic plate and rather simple to implement in the mesh of finite element analysis. This criterion criteria states that crack will starting to growth when a critical angle of crack mouth opening is exceed.

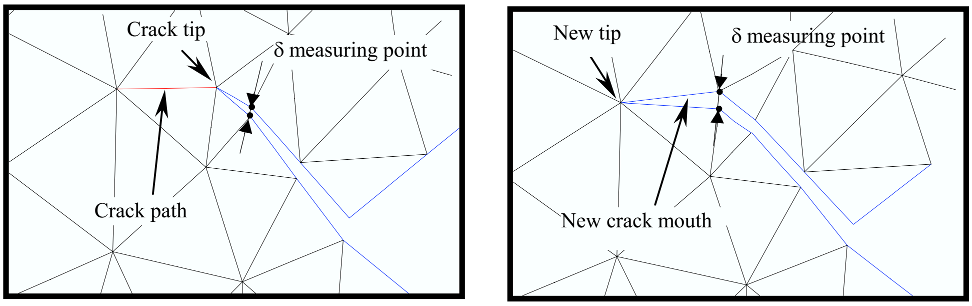
Under the assumption that the crack is growing, there are several criteria for the direction of crack growth such as maximum principal stress, energy release rates criterion and criterion of local symmetry [8]. The criterion used in this paper is maximum principal stress.

The maximum principal stress criterion states that crack growth will occur in a direction perpendicular to the maximum principal stress. This criterion is considerably easy to use in approximation method. Because of its simplicity, the maximum principal stress criterion is often used in practice. In this criterion, the crack is extended by linear segment.

4. CRACK INITIATION AND PROPAGATION

In the crack propagation modeling, there are two most widely used crack initiation model i.e. release node and element failure [9]. In this paper split node strategy, which similar to release node technique, is used to model the crack initiation. In split node strategy, the crack tip motion is modeled by changing the location of crack tip from one node to the next along the crack axis. The nodal will be gradually split after certain criterion exceeds.

Figure 1 shows the splitting mechanism of crack tip node. Here the crack tip node is split into two new nodes by adding a very small gap after a certain condition i.e. maximum crack tip opening angle is exceeded.



(a) (b)

Fig.1. Crack propagation mechanism (a) before crack advance (b) crack advance

The crack tip opening angle criterion is measured at the edge of the closest segment that connects to the crack tip (as seen in Figure 1). The measuring point is not fixed at a certain distance from the crack tip. This method is applied to overcome the measuring problem that usually occurs when the crack is not going straight.

To simulate the crack propagation, the inter-element crack propagation model is used in this paper. In the inter element modeling, crack is modeled to propagate through inter element in the mesh. The node of the element that choose as a direction is the node which have the closest path to the direction path which calculated using maximum principal stress criterion, as shown in Figure 2. Determination of crack growth direction is starting right after CTOA obtain a minimum angle for crack to growth.

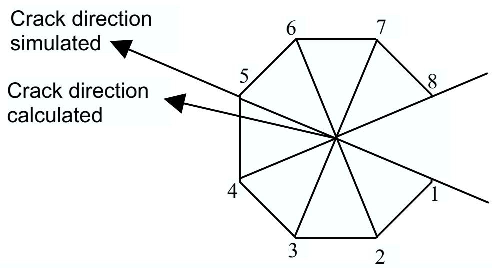


Fig.2. Determination of crack propagation direction

By using this inter element crack propagation model, the path taken by crack propagation is depending on the element size of the mesh. To obtain a smooth path of crack propagation, the size of the element mesh around the crack tip must be small enough.

5. THE SIMULATION OF CRACK PROPAGATION

Several numerical examples are presented below to show the ability of technique that implemented in this paper. In the first example of crack propagation simulation, a simple shape of rectangular plate with single crack subjected to vertical loading and uniformly distribute on the top boundary is used as shown in Figure 3. For each step of crack advance, global remeshing procedure is applied to keep a good precision around the crack tip.

As seen from simulation result, the crack is propagates in cleavage mode (mode I) where the direction of crack propagation perpendicular to the maximal stress, which is vertical due to applied load. Although the crack path geometry is not smooth, the simulation result is able to illustrate a crack mouth opening phenomena for each time of load increased.

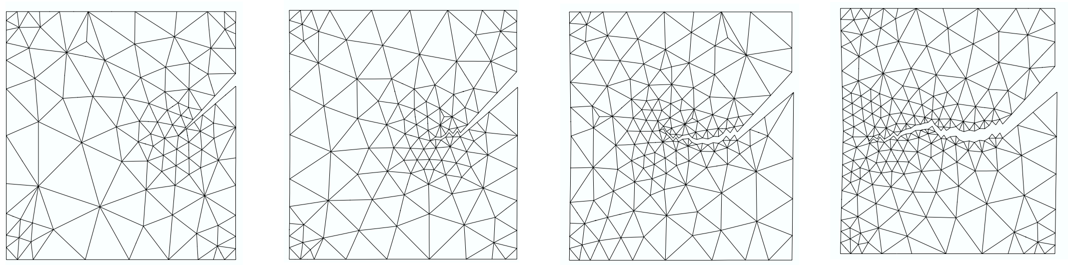
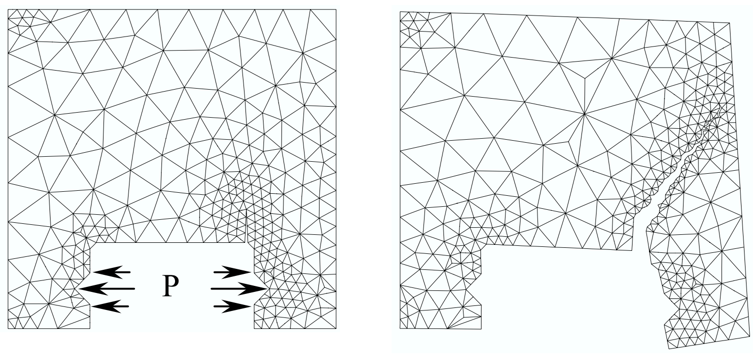
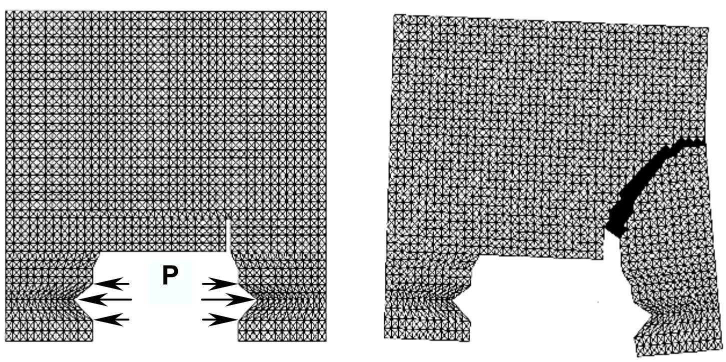


Fig.3. Some of the crack propagation sequence under mode I loading condition

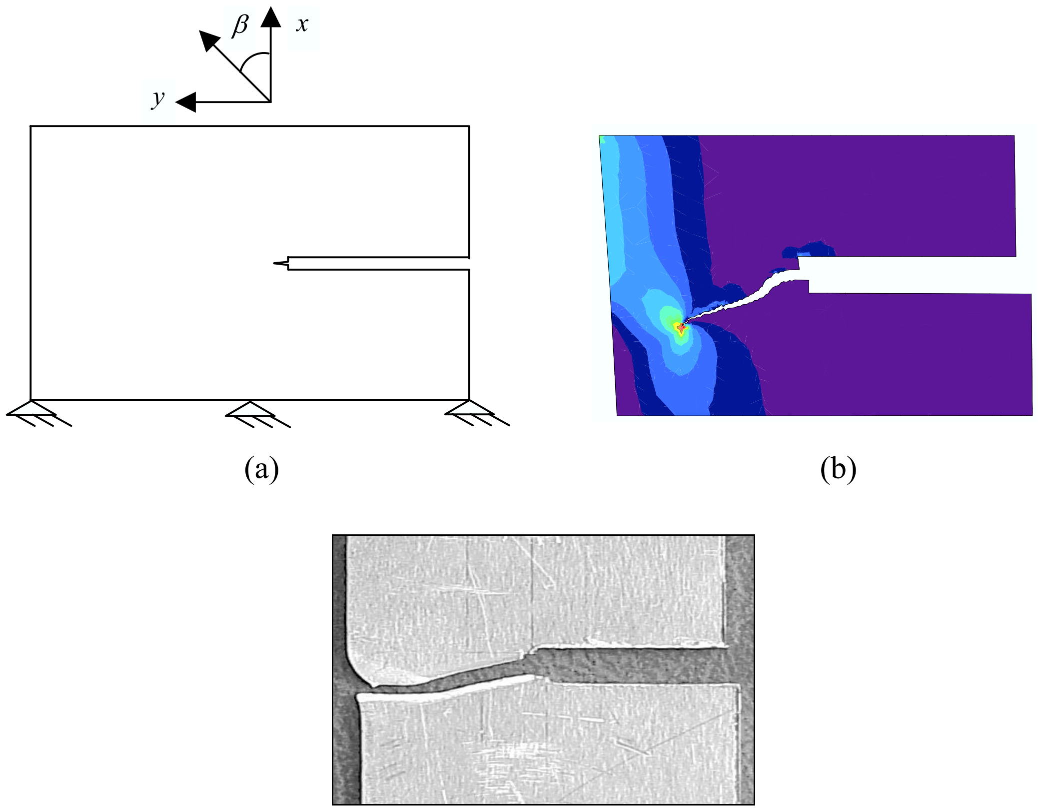


(a)



(b)

Fig.4. Simulation result of the coupled pressure bar specimen, (a) simulate using presented technique (b) simulated by [9]



(a) (b)

(c)

Fig.5. Crack propagates simulation for mixed mode I-II with loading angle 60^{o}.
(a) Loading and boundary condition of model, (b) Simulation result and
(c) experimental result

Figure 4 shows the crack propagation of coupled pressure bar specimen. The pre crack is placed near to one side of specimen grips. Unsymmetrical loading produce cracks propagation that is started from crack-tip and propagates to the right side of specimen. In the range of stable fracture, the result obtained from presented technique is similar to simulation result from [9] as seen in Figure 4(b).

Comparison between simulation and experimental result is shown in Figure 5. A rectangular plate with a notch is subjected to mixed mode I-II loading condition with $\beta = 60^{o}$, as seen in Figure 5(a). Load is uniformly distributes at the top of boundary segment. A small size of pre crack is introduced in the middle of notch as initial crack. This initial crack is needed as a point to measure crack propagation criteria during simulation process. In the experimental work, the specimen is made from aluminum with 1 mm thickness. As seen in Figure 5(b) and 5(c), the direction of crack propagation obtained from numerical simulation is similar to experimental result.

6. CONCLUSION

The inter element crack model has been successfully applied to simulate crack propagation using finite element method with adaptive mesh. The maximum principle normal stress criterion is able to predict the crack direction in the simulation. Several numerical examples presented above shows the ability of the technique that implemented. In these examples, every step in the crack evolution is completely automatic.

REFERENCES

1. Sienz, J. and Hinton, E, Computer and Structures Vol. **3** No.1-4(1997)p.31-63.
2. Lee, C.K. and Lo, S.H., International Journal for Numerical Methods In Engineering, **35**(1992)p.1967-1989.
3. Murthy, K.S.R.K. and Mukhopadhyay, International Journal of Fracture, **108**(2001)p.251-274.
4. Bouchard, P.O, Bay, F., Chastel, Y. & Tovena, I., Comput. Methods Appl. Mech. Engrg, **189**(2000)p.723-724.
5. Antonio Rodriguez-Ferran and Antonio Huerta, International Journal of Solid and Structures, **37**(2000)p.7501-7528.
6. Zienkiewicz, O.C. and Zhu, J.Z., International Journal For Numerical Methods In Engineering, **28**(1989)p.2839-2853.
7. Ahmad Kamal Arifin, Syifaul Huzni, Nik Abdullah Nik Mohamed, Proceedings ETM 2002 Experimental and Theoretical Mechanics, Indonesia, Sanur-Bali(2002)p.21-25.
8. Stone, T. J. and Babuska, I., Comput. Methods Appl. Mech. Engrg., **160**(1998)p.245-271.
9. Beisel, S.R., Johnson, G.R., Popelar, C.H., Engineering Fracture Mechanics **61,**(1998)p.407-425

Advances in Fracture and Failure Prevention
doi:10.4028/www.scientific.net/KEM.261-263

The Implementation of Inter-Element Model for Crack Growth Simulation
doi:10.4028/www.scientific.net/KEM.261-263.687