Finite Element Modeling of Fatigue Crack Propagation Using a Self Adaptive Mesh Strategy

Abdulnaser M. Alshoaibi* and A. K. Ariffin
Department of Mechanical & Materials Engineering, Universiti Kebangsaan Malaysia,
43600 Bangi, Selangor Darul Ehsan, Malaysia
E-Mail: alshoaibi@gmail.com

Abstract – A new finite element model has been developed to predict fatigue crack growth in arbitrary 2D geometries under constant amplitude loading. The purpose of this model is on the determination of 2D crack paths and surfaces as well as on the evaluation of components Lifetimes as a part of the damage tolerant assessment. Throughout the simulation of crack propagation an automatic adaptive mesh is carried out in the vicinity of the crack front nodes and in the elements which represent the higher stresses distribution. The fatigue crack direction and the corresponding stress-intensity factors are estimated at each small crack increment by employing the displacement correlation technique under facilitation of singular crack tip elements. The propagation is modeled by successive linear extensions, which are determined by the stress intensity factors under linear elastic fracture mechanics (LEFM) assumption. The stress intensity factors range history has to be recorded along the small crack increments. Upon completion of the stress intensity factors range history recording, fatigue crack propagation life of the examined specimen is predicted. Verification of the predicted fatigue life is validated with relevant experimental data and numerical results obtained by other researchers. The comparisons show that this model is capable of demonstrating the fatigue life prediction results as well as the fatigue crack path satisfactorily.

Keywords: Finite element simulation, stress intensity factors, mixed mode fracture, adaptive mesh, Fatigue life prediction

I. Introduction

There are many different mechanical failure exists in simple or complex modes. The deterioration of engineering structures due to fatigue has been a difficult problem facing engineers for many decades. It has been estimated that between 50 and 90 % of these failures are due to fatigue. Fatigue analyses have often employed rather large safety factors, in order to compensate for the large degree of uncertainty involved. Fatigue damage accumulation can be explained in terms of the initiation and growth of small cracks in the metal. The crack propagation progressively reduces the capability of the components to withstand the applied external load and finally break the components. The major source of failure of structural components is fatigue crack growth. In the past, the S–N curves were the only engineering tools, and crack propagation was not considered to predict lifetime. Nevertheless, at present, studying crack growth behavior has been made possible by the LEFM and the prediction of remaining lifetime of components is accessible. This is of great practical importance in order to know if a part can be still used or must be replaced.

A predictive method consists in integrating the crack growth law. Two kinds of data are required: first, the experimental Paris crack growth law of the considered material and second, the stress intensity factors (SIFs) distribution along the crack front. The Paris relation is quite easy to obtain on the contrary to SIFs [1].

Various attempts have been made to develop efficient model to estimate fatigue crack growth and life time. The analytical approach, such as that contained in NASCRAC and NASA/FLAGRO for fatigue crack growth, employs known solutions for $I_K$ as a function of crack length to generate crack growth predictions. Such predictions are therefore limited to self-similar crack growth. Further, the analytical solutions are available for only a limited set of combinations of geometries and boundary conditions. However, several simulation codes have already been developed for the evaluation of cracks within 2D components, e.g. FRANC2D, FASTRAN [2]-[3], AFGROW and NASGRO. In addition, there are many recent development performed by many researchers to find an
efficient method to predict the fatigue crack growth under mixed mode loading in 2D linear elastic structures e.g. [4]-[5]-[6].

The present model 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 analyse fatigue crack path lifetimes. This program is written in FORTRAN language.

II. Mesh Generation and Adaptive Refinement

In this work, the unstructured triangle mesh is automatically generated by employing the advancing front method. This adopted technique however requires generating background mesh in order to accurately control the distribution of the geometrical characteristics such as the element size, element stretching and stretching directions for the new mesh. The background does not have to be precisely representing the geometry; however the accuracy of the distribution depends on this excellence and it must be completely cover the computational domain [7].

The strategy taken to generate the background mesh is to utilize all the initial boundary nodes of geometry and construct the boundary triangles as the background mesh by the dichotomy technique [8]. In order to properly represent the field singularity around the crack tip, the singular elements have to be constructed as well. Since the advancing front method generates the triangle elements starting from the boundary faces, the area around the crack tip for the construction of the singular elements is supposed to be isolated. This area is isolated by first generating nodes around the crack tip in the rosette form and then the crack tip node and the jointed boundary segments are removed. New boundary segments are then introduced linking all the new nodes to temporarily ‘cut out’ the template area from the original domain. Subsequently the advancing front triangulation can be executed. Finally singular elements are ‘patched’ into the rosette template to complete the process [8].

In general, the smaller mesh size gives more accurate finite element approximate solution. However,
reduction in the mesh size leads to greater computational effort. The adaptive mesh refinement is employed as the optimization scheme. This scheme based on a posteriori error estimator which is obtained from the solution from the previous mesh. Here stress error norm is taken as the error estimator. The strategy used to refine the mesh during analysis process is adopted from [9] as follows:

The main idea in the \( h \)-type adaptive mesh refinement is to obtain the ratio of element norm stress error to the average norm stress error of the whole domain which is also known as relative stress norm error and from this ratio the new size of the element for the refinement process can be predicted. In this procedure the mesh size of each element is defined as:

\[
h_e = \sqrt{2A_e}
\]

(1)

where \( A_e \) is the area of the triangle element.

The norm stress error for each element is defined by:

\[
\left\| \mathbf{\sigma} - \mathbf{\sigma}^* \right\| = \int_{\Omega_e} (\mathbf{\sigma} - \mathbf{\sigma}^*)^T (\mathbf{\sigma} - \mathbf{\sigma}^*) \, d\Omega
\]

(2)

while the average norm stress error for the whole domain is:

\[
\left\| \mathbf{\sigma} \right\| = \frac{1}{m} \sum_{e=1}^{m} \int_{\Omega_e} \mathbf{\sigma}^T \mathbf{\sigma} \, d\Omega
\]

(3)

where \( m \) is the number of total elements in the whole domain and \( \mathbf{\sigma}^* \) is the smoothed stress vector which consists of smoothed stresses components.

In the finite element treatment the integration with the isoparametric triangular element will be converted by the summation of quadratics following the Radau rules as follows:

\[
\int_{\Omega_e} \mathbf{\sigma}^T \mathbf{\sigma} \, d\Omega = \frac{1}{m} \sum_{e=1}^{m} \int_{\Gamma_e} \mathbf{\sigma}_e^T \mathbf{\sigma}_e \, d\Omega
\]

(4)

and similarly

\[
\int_{\Omega_e} (\mathbf{\sigma} - \mathbf{\sigma}^*)^T (\mathbf{\sigma} - \mathbf{\sigma}^*) \, d\Omega = \frac{1}{m} \sum_{e=1}^{m} \sum_{p=1}^{p_e} \mathbf{\sigma}_e^T (\mathbf{\sigma}_e - \mathbf{\sigma}_e^*) \, d\Omega
\]

(5)

where \( t^* \) is the thickness of element for a plane stress condition and \( t^* = 1 \) for a plane strain condition. It is obvious that these parameters are evaluated involving the values of stresses and smoothed stresses at the sampling points only.
It is considerable then to make the relative stress norm error for each element \( \zeta_e \) less than some specified value \( \zeta \) (say 5% for many engineering applications [7]). Thus

\[
\zeta_e = \frac{\| \varepsilon \|}{\| \varepsilon \|} \leq \zeta
\]

and the new element relative stress error norm with permissible error of \( \zeta \) is defined as:

\[
\varepsilon_e = \frac{\| \varepsilon \|}{\zeta} \leq 1
\]

This means any element with \( \varepsilon_e > 1 \) need to be refined and subsequently, the new size of mesh refinement need to be predicted. Here used the asymptotic convergence rate a criterion is used whereby it is assumed that:

\[
\| \varepsilon \| \propto h^p_e
\]

where \( p \) is the polynomial order of approximation. In the present study case \( p = 2 \) since the quadratic polynomial is used for the finite element approximation. The approximate size of new element thus

\[
h_{ne} = \frac{1}{\sqrt{\varepsilon_{ne}}} h_e
\]

The current mesh will be regarded as the new background mesh and the advancing front method will be repeated depending on the number of mesh refinement set by the user. The totally new mesh with the adaptive mesh size will be generated in this scheme. During the process, the interpolation of the background mesh parameters needs the parameters to be assigned to each of the background nodes instead of to each of the background elements as by the Equation (9). The new mesh size of each element is then manipulated similar way as the averaging method to obtained the smoothed stress before and the averaged mesh size is therefore assigned to each of the points.

III. Stress Intensity Factor And Fatigue Crack Growth Analysis

In this paper, the displacement extrapolation method [10] is used to calculate the stress intensity factors as follows:

\[
K_e = \frac{E}{3(1+\nu)(1+\kappa)} \sqrt{\frac{2\pi}{L}} \left[ 4(u'_x - u'_e) - \frac{(v'_x - v'_e)}{2} \right]
\]

where \( E \) is the modulus of elasticity, \( \nu \) is the Poisson’s ratio, \( \kappa \) is the elastic parameter defined by:

\[
k = \begin{cases} (3-4\nu) & \text{plane stress} \\ \frac{(3-4\nu)}{\left(1+\nu\right)} & \text{plane strain} \end{cases}
\]

and \( L \) is the quarter point element length. 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.

![Fig. 1 Quarter-point singular elements around the crack tip](image)

In order to simulate fatigue crack propagation under linear elastic condition, the crack path direction must be determined. There are several methods use to predict the direction of crack trajectory such as the maximum circumferential stress theory, the maximum energy release rate theory and the minimum strain energy density theory. Bittencourt et al. (1996) have shown that, if the crack orientation is allowed to change in automatic fracture simulation, the three criteria provide basically the same numerical results, since the maximum circumferential stress criterion is the simplest, presenting a closed form solution, it is briefly described bellow.

The stress at the crack tip for mode I and II are given by assuming up the stress fields generated by each mode as:

\[
\sigma_I = \frac{1}{\sqrt{2\pi r}} \cos(\theta/2) \left[ K_I \left( 1 + \frac{1}{2} K_{II} \sin \theta \right) \right]
\]

\[
\sigma_{II} = \frac{1}{\sqrt{2\pi r}} \cos(\theta/2) \left[ K_{II} \cos^2(\theta/2) - \frac{3}{2} K_{II} \sin \theta \right]
\]

\[
\tau_{\theta} = \frac{1}{\sqrt{2\pi r}} \cos(\theta/2) \left[ K_I \sin \theta + K_{II} (3\cos \theta - 1) \right]
\]
where \( \sigma_r \) is the normal stress component in the radial direction, \( \sigma_\theta \) is the normal stress component in the tangential direction and \( \tau_{r\theta} \) is the shear stress component. The maximum circumferential stress theory asserts that, for isotropic materials under mixed-mode loading, the crack will propagate in a direction normal to maximum tangential tensile stress (thus \( \tau_{r\theta} = 0 \)).

The direction normal to the maximum tangential stress can be obtained by solving \( \frac{d\sigma_\theta}{d\theta} = 0 \) for \( \theta \). The nontrivial solution is given by:

\[
\theta = 2 \arctan \left( \frac{1}{4} \frac{K_r}{K_\theta} \pm \frac{1}{4} \sqrt{\left( \frac{K_r}{K_\theta} \right)^2 + 8} \right)
\]

Since fatigue is a cyclic dissipation of energy in the form of hysteretic loops, it is a cumulative damage process. The elapsed time for damage is expressed in terms of the number of fatigue cycles (\( N \)). This process is defined by crack growth per cycle; \( da/dN \). The rate \( da/dN \) depends on the applied stress intensity factor range. Basically, for fatigue crack to grow, the resulted stress intensity range at each crack tip must exceed the stress intensity threshold defined as

\[
\Delta K_{th} = f \Delta \sigma_a \sqrt{\pi a}
\]

where \( f \) is a function of geometry and loading and \( \Delta \sigma_a \) is the stress range limit. Equation (15) set a criterion whereby fatigue crack does not propagate if \( \Delta \sigma < \Delta \sigma_a \). However, practically a parameter known as equivalent stress intensity factor range, \( \Delta K_{eq} \), is used in such \( \Delta K_{eq} > \Delta K_{th} \) indicates that fatigue crack grows will happen otherwise there is no fatigue crack propagation. This parameter is defined as:

\[
\Delta K_{eq} = \Delta K_f \cos^2(\theta/2) - 3\Delta K_h \cos^2(\theta/2)\sin(\theta/2)
\]

Thus it is easy to model the stable crack propagation using the generalized Paris’ equation as:

\[
\frac{da}{dN} = C \left( \Delta K_{eq} \right)^m
\]

where \( C \) and \( m \) are the material properties. The number of cycles \( N_{eq} \) can be predicted for the crack propagation by integrating from the initial length \( a_i \) to the final crack length \( a_f \) as:

\[
N_{eq} = \int_{a_i}^{a_f} \frac{1}{C \left( \Delta K_{eq} \right)^m} da
\]

For a single-cracked body case, \( \Delta a \) can be substituted directly into the piece wise numerical integration.

Nevertheless, for multi-cracked bodies, the maximum increment length \( \Delta a_{max} \) and the maximum equivalent stress intensity range \( \Delta K_{eq,max} \) are required and the increment of each crack is determined by

\[
\Delta a = \Delta a_{max} \left( \frac{\Delta K_{eq}}{\Delta K_{eq,max}} \right)^m
\]
Fig. 4 shows the comparison of stress intensity factors distribution at tips A and B, along the crack length with the result of [12] obtained by using meshless finite element. The figure exhibits a good agreement with the reference result. The retardation of $K_{I}^{A}$ at crack length of 27 mm to above is due to the interaction with the opposite crack trajectories.

The fatigue life diagram is presented in Fig. 5. The fatigue life of the structure is evaluated as 6840 cycles, which is in good agreement with the results obtained by [12] using a meshless method and also with the numerical results using BEM obtained by [13].

III. 2 Modified Four Point- Bending SEN specimen

The crack pattern for mixed mode fracture is usually a curve. It is of interest to investigate fatigue crack growth behavior for such case. For this purpose, a modified four point bending is tested under constant amplitude fatigue loading with load ratio, $R = \frac{\sigma_{\text{min}}}{\sigma_{\text{max}}} = 0.1$.

Fig. 6 shows the dimensions of the modified four-point bending SEN and the final adaptive mesh for the first step. This specimen is steel with the material properties as follows:

$E = 205 \text{ GPa}$, $\nu = 0.333$, $K_{Ic} = 90 \text{ MPa} \sqrt{\text{m}}$, $m = 2.1$, $C = 4.5 \times 10^{-10}$, and $\Delta K_{\text{th}} = 11.6 \text{ MPa} \sqrt{\text{m}}$.

Fig. 7 shows four different steps of fatigue crack propagation of this specimen. The predicted crack path
behaviors by the developed program is compared with the experimental results observed by [14], in which they also compared their experimental results with the numerical results using Quebra2D software as shown in Fig. 8. One can obviously observe the similarity of crack path pattern predicted in the present study and those obtained in Fig. 11.

The fatigue life diagram of this specimen is presented in Fig. 9. The life of the structure is evaluated as 653604 cycles, which is in agreement with experimental and numerical results using ViDa software obtained by [14].

III. Conclusion

The adaptive finite element model has been developed to analyze fatigue crack path, fatigue life and stress intensity factors along the crack length under constant amplitude loading and LEFM consideration. The developed code is assessed with several test specimens and the outcomes are compared to the similar works either by numerically or experimentally. The program has demonstrated comparably satisfaction predictions of fatigue life and crack propagation paths at least for the tested 2D geometry specimens and therefore it can be highly justified that the adaptive finite element procedure has been successfully employed.

References


Authors’ information

Dr. Abdulnaser Mohammed Alshoabi
Email: alshoaibi@gmail.com
naser@vlsi.eng.ukm.my

Dr. Alshoaibi received his PhD from the Department of Mechanical and Materials Engineering in Universiti Kebangsaan Malaysia in 2007, MSc from the Department of Nuclear Engineering in Baghdad University, Iraq in 2002 and BSc From the Department of Materials Engineering in Babylon University, Iraq, in 1998. His research interests are fracture mechanics, failure analysis, finite element, powder compaction, stochastic fracture, corrosion and materials properties.

Prof. Dr. Ahmad Kamal Ariffin
Email: kamal@eng.ukm.my

Prof. Kamal received his PhD in Computational Mechanics from University of Wales Swansea, UK, in 1995, Bachelor degree in Mechanical Engineering from Universiti Kebangsaan Malaysia in 1990. He is now the Head of Mechanical and Materials Engineering at Universiti Kebangsaan Malaysia. His research interests are fracture mechanics, failure analysis, finite element, powder compaction, stochastic fracture, corrosion, Parallel computation.