Engineering Failure Analysis 59 (2016) 41–56

Contents lists available at ScienceDirect

Engineering Failure Analysis

journal homepage: www.elsevier.com/locate/efa

A simple method based for computing crack shapes

D. Peng [a,*], P. Huang [a], R. Jones [a], A. Bowler [b], D. Edwards [c]

[a] Department of Mechanical and Aerospace Engineering, Monash University, P.O. Box 31, Monash University, Victoria 3800, Australia
[b] Maritime Systems Project Office (MPSPO), RAAF Base Edinburgh, Edinburgh, SA 5111, Australia
[c] Airbus Group Australia Pacific, Buildings 46, Corner of East Avenue and Explosives Road, Edinburgh, SA 5111, Australia

ARTICLE INFO

Article history:
Received 15 July 2015
Received in revised form 27 September 2015
Accepted 6 November 2015
Available online 10 November 2015

Keywords:
Crack shapes
Intergranular cracking
Nastran
Genetic algorithm

ABSTRACT

The growth of cracks from small naturally occurring material discontinuities plays a major role in the operational lifespan of aircraft structures. Calculating the life of an airframe structure requires the determination of the crack path(s) which for complex real life geometries can often be highly complex. This paper presents a simple method based on finite element analysis for estimating the crack growth path. The analysis is based on an element removal approach and uses the major principal stress as the failure criteria. Evolution strategies are derived from the biological process of evolution. Three examples are presented demonstrating the utility of the proposed technique.

1. Introduction

A knowledge of potential crack paths[1]/shapes is important both when calculating the lifespan of an airframe and when determining the appropriate inspection procedures. For many "real life" problems, the development of the shapes of life limiting cracks is often quite complex. A general analytical solution does not yet exist. A number of methods can be used for this class of problems. Of these methods: the finite element [1–14], boundary element [15], mesh free [16,17], extended finite element [18,19], virtual crack closure-integral [20] and s- and p-element [21,22] methods are perhaps the most widely used techniques for computing the stress intensity factors needed to simulate 3D crack growth. However, as explained in Appendix X3 of the ASTM fatigue crack growth standard E-647-13a, a significant proportion of the life of an operational structure is spent in the regime where the crack is small.[2] For cracks growing in primary structural members the crack shape often has a complex three dimensional shape. However, in the absence of fractography data, it is not possible to know the crack shapes prior to the analysis. The analyst is therefore forced to make educated guesses of the shapes or compute new K (stress intensity factor) solutions after each increment in the crack growth. Both approaches have limitations. To determine the stress intensity factors and thereby the subsequent crack shape for small cracks requires a very fine local mesh. Such meshes result in a commensurate large CPU solution time. Furthermore, it is generally necessary to re-mesh and resolve after each increment of crack growth. This usually involves substantial computational effort and thereby often limits such approaches to simple problems. As such the challenge is to develop a simple standalone method whereby the crack shapes can be determined prior to any fatigue analysis. This would enable the stress intensity factors needed to life a structure to be determined prior to the fatigue analysis.

To meet this challenge, the present paper investigates the use of the 'Nibbling Algorithm' approach described in [23,24]. Here attention is focused on the potential for determining the crack path/shapes associated with growth of small three dimensional cracks in realistic structural geometries. This 'Nibbling Algorithm' approach computes the crack path(s)/shapes by sequentially

* Corresponding author.
[1] In this paper the phrase "crack path prediction" is defined as predicting the successive position of the crack front.
[2] Here the definition of "small" is as defined in ASTM E647.

http://dx.doi.org/10.1016/j.engfailanal.2015.11.016

removing those elements that have the highest maximum principal stress. In [23,24] it was shown to successfully simulate both the growth of squats in railway tracks as well as the growth of an inclined crack under constant amplitude loading. An advantage of this method is that it only uses an uncracked finite element mesh and does not require re-meshing every time the crack grows.

To illustrate this approach, three problems are analysed and the computed crack shapes are compared with those seen in test specimens. Two problems involve crack growth from fastener holes. One "fastener hole" problem is related to cracking fastener holes in RAAF (Royal Australian Air Force) AP3C (Orion) aircraft. In this context, when assessing the fatigue life of RAAF AP3C (Orion) aircraft [28,29] attention has recently focused on understanding a new class of multi-site damage problems. This involves the growth of small sub mm cracks in 7075-T6 wing skins at fastener holes containing intergranular cracking. The other "fastener hole" problem involves cracks that grow from etch cracks [31,32] at fastener holes in a specimen test performed in support of RAAF F/A-18 aircraft. The other problem considered examines crack growth in the helicopter round robin test programme described in [25,27].

The results of these studies illustrate the ability of the 'Nibbling Algorithm' to generate complex three dimensional crack shapes that are in reasonable agreement with those seen experimentally.

2. Methodology

This section describes a simple evolutionary procedure for computing crack growth under alternating loads. This technique is termed a 'Nibbling Algorithm'. The 'Nibbling Algorithm' is a heuristic method. It works by removing elements from highly stressed regions. In this study, as in [23,24], a representative maximum principal stress for each element and an ultimate tensile strength are used in the element selection criterion [33]. The average maximum principal stress for each element is derived from the corresponding gauss point stresses and its value is used to determine if the element will be removed from the structure. In a given stage (iteration) the ith element is removed if:

$$\sigma_i \geq (1-SF) * Max\langle\sigma_{1\,max}|\sigma_u\rangle \tag{1}$$

where σ_i is the representative average maximum principal stress for the ith element, $\sigma_{1,\,\max}$ is the peak maximum principal stress for all the elements in the structure and σ_u is the ultimate tensile strength of material and SF is an elimination factor. The elimination factor plays an important role in controlling the iteration process. A high value will lead to a rapid convergence, but may cause instability. The instability may drive the solution away from a correct pathway of crack growth. In contrast, a very low value will require a large number of iterations and can dramatically increase the solution time.

Finite element analysis is generally used for determining the structural response. Once an element has "failed"[3] it is then removed from the structure. As outlined above the maximum principle stress at the centroid of each element is chosen as the element removal criteria and elements with highest maximum principle stress are eliminated at each evaluation/iteration. This is done by changing the Young's modulus of the structure to a very small value, typically 1/1000th of its previous value. This results in a new FE model. The updated (new) structure is then re-analysed. Depending on the response of the new structure the algorithm will again use the removal criteria to identify elements and eliminate them from the structure. This process is continued and the associated crack shapes determined until the resulting structure fails or the user terminates the iterative process.

3. Crack growth in a helicopter lift frame

As the first example considered the problem of crack growth in the helicopter component described in [25]. This component was a flanged plate with a central lightning hole made of the 7010 alloy, see Fig. 1. The finite element model used in [26] to study crack growth in this geometry is shown in Figs. 2 and 3. Due to symmetry considerations only 1/4 of the structure was analysed, see Fig. 3. The resultant mesh had 17,451 elements and 78,608 nodes, see [26]. The material of the round-robin was taken to be an aluminium alloy 7010-T73651 which had a Young's modulus and a Poisson's ratio of E = 70,000 MPa, and $\nu = 0.3$ respectively. The average room temperature tensile strength for this material is 502 MPa and the yield strength is 440 MPa. See Fig. 4

As per [25] it was assumed that there was a small initial corner defect on the inner edge of the large central hole, see Fig. 3. Two different elimination factors were used in this example, viz.: 0.01 (case 1) and 0.05 (case 2).

The predicted crack growth shapes are shown in Figs. 2 and 5 for case 1 and case 2 respectively where we see minimal difference between the results obtained using these two elimination factors. Comparing the computed (Fig. 6) and measured (Fig. 7), there is good agreement between the computed and the experimental crack shapes [27].

4. Crack tunnelling at a fastener hole with intergranular cracking

Having established the potential of this approach to accurately capture crack shapes. Let us investigate a new class of multi-site damage problems, viz.: the interaction between intergranular cracks and cracks that grow from small naturally occurring material discontinuities at a fastener hole [28–31]. The specific example studied involves crack growth associated with a P-3 DNH (Dome Nut Hole) coupon [30] which has IGC (Intergranular Cracking) at the fastener hole, see Fig. 8. The unit used in this specimen

[3] The term 'failure' means that the element is cracked or is not taking part or contributing to the overall performance of the structure.