SIMULATION OF THE CRACK GROWTH OF 3D STRUCTURES UNDER FATIGUE IN TAKING ACCOUNT OF RESIDUAL STRESSES

H. Nguyen-Dang, M. Scibetta and B.Borlee[1]

The technique of fissuring box may be considered as an interesting alternative tool in order to mesh (FEM) 3D crack front. It consists of bounding around the crack a box in such a way that the refinement or the remeshing of the finite elements in the neighbourhood of the crack tip may be done without modification of the mesh of finite element situated outside the box. Using this technique the paper presents also various methods of calculation of the intensity factors: Barsoum's direct method via displacement response, EDI, VCE... A practical example is also presented. The influence of the presence of the residual stresses are considered almost on the life prediction of the cracked 2D structures.

INTRODUCTION

Life time and crack propagation calculation are very important data for design in the aerospace area. It is usually hard to predict how a crack will occur and propagate since geometry and loading of the real structures are complex. Nowadays most of structures can be quite easily analysed by finite element methods even though the structure is cracked, but meshing is still a problem when the crack grows since the geometry of the structure changes. This problem can be partially solved by using a fissuring box. This tool consists in generating automatically a mesh along the 3D crack front. The crack front can be any continuous line such a spline or an ellipse. When the crack grows, the crack front progress. In order to predict this progression, we can use an iterative method. At each step we compute the stress intensity factor along the crack front by a classical elastic analysis. The crack front shape for the next step is determined by the crack propagation law. Then we re-mesh the cracked part of the structure with the fissuring box tool. In this paper we will present an example of the use of this technichs: we attempt to predict a crack propagation in an industrial case.

---

[1]LTAS-Fracture Mechanics, University of Liège, BELGIUM

NUMERICAL METHODS FOR STRESS INTENSITY FACTOR CALCULATION

Ingraffea-Manu's Method

Using the Barsoum's finite element, Ingraffea and Manu [4]established remarkable formulas that directly provide stress intensity factor in function of node displacements on the crack lips. A more complete form has been proposed by Nguyen and de Saxe [10] and implemented in the finite element code SAMCEF.

$$K = \frac{E}{4(1-\nu^2)}\sqrt{\frac{2\pi}{L_1}}\,F(u_i)$$

where L_1 in the radial direction and F is a function of node displacements. This technique is quite easy to implement and has low CPU-time cost.

Méthode VCE [3]

The potential energy of a discretized structure is: $U = \frac{1}{2}\mathbf{q}^T\mathbf{K}\mathbf{q} - \mathbf{q}^T\mathbf{g}$

The variation is $\delta U = \frac{1}{2}\mathbf{q}^T\delta\,\mathbf{K}\mathbf{q} - \mathbf{q}^T\delta\,\mathbf{g} + (\mathbf{K}\mathbf{q}-\mathbf{g})^T\delta\,\mathbf{q}$

So the energy release rate is: $G = \frac{1}{2}\mathbf{q}^T\frac{\delta\,\mathbf{K}}{\delta S}\mathbf{q} - \mathbf{q}^T\frac{\delta\,\mathbf{g}}{\delta S}$

In order to compute this energy release rate, we made a sensibility calculation by perturbing the mesh. This method is an energetic one and needs poor CPU-time since only nodes near the crack tip have their geometry changed.

Méthode EDI [9]

This method is based on the J-integral calculation: $J_k\Delta = \int_{A_e}\left(Wn_k - \sigma_{ij}\partial_k u_i n_j\right)dA$

Using a weight function that is null on the boundary; The J-integral can be expressed as:

$$J_kF = -\int_V\left(W\partial_k s - \sigma_{ij}\partial_k u_i\partial_j s\right)dV - \int_V\left(\partial_k W - \partial_j(\sigma_{ij}\partial_k u_i)\right)s\,dV + \int_{A_1+A_2}\left(Wn_k - \sigma_{ij}\partial_k u_i n_j\right)s\,dA + \int_{B_1+B_2}\left(Wn_k - \bar{t}_j\partial_k u_i\right)s\,dA$$

where F is the surface under the curve s beyond the crack front.

The first term is usually prevalent. The second one presumably vanishes if no body forces. The third one is usually small if crack fronts elements are elongated. The fourth one is null if the crack lips are not loaded [13].

A 3D CRACK PROPAGATION EXAMPLE [1].

The structure to be analysed is an axle in a plane engine loaded by a twisting moment (figure 1). A finite element analyse shows that there exists a stress concentration in the vicinity of an oil hole. Due to the shear stress state, the largest stress zone will happen in the direction making an 45° angle with the axle axis.

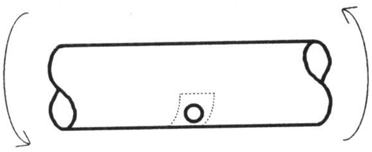

Fig. 1

We assume that a small semi-elliptic crack occurs in the place where the highest tensive stress is present (in the thickness of the axle). The lips plane is perpendicular to the tensile direction. So the main fracture mode is the mode I. This choice is justifiable if we admit that a crack always grows in the way of the mode I. The dimensions of the crack are chosen to be the smallest that can be practically detected.

The propagation law for the given material is a modified *Paris*' law.

As we assumed that the main fracture mode were the first one, the mode II is very small, so the crack will grow in the same plane and the lips are always coplanar.

We had also made the hypothesis that the crack shape will remain an ellipse shape. This assumption will be checked afterwards. This simplification reduces the number of geometrical parameters to three: the displacement of the higher edge a, the displacement of the lower edge c and the displacement of the middle edge b (figure 2). At initial state, the geometrical parameters a, b, c take the values a_0, b_0, c_0.

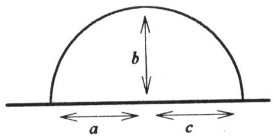

Fig. 2

The principle of the method is to put a small crack by changing locally the mesh of the structure used for the first finite element analysis mentioned above. Keeping the mesh allows us to simulate perfectly the geometry of the structure and the loading conditions. This modelization allows to evaluate the real stress field and stress concentration at the crack front for the most general case of geometry and loading that one cannot find in tables.

A fatigue-life computation needs several finite element analysis. So, in order to save the CPU-time, we made a structural zoom round the oil hole. The structure is reduced to a region around the oil hole. The role of the rest of the structure is transmitted by appropriate imposed displacements on the external nodes.

At first, we built a sub-mesh round the crack front by using the new fissuring box tool. The faces of the fissuring box are compounded with normal squared elements. So this mesh should be easily connectable to the rest of the structure (figure 3). The elements just round the crack are the Barsoum's type.

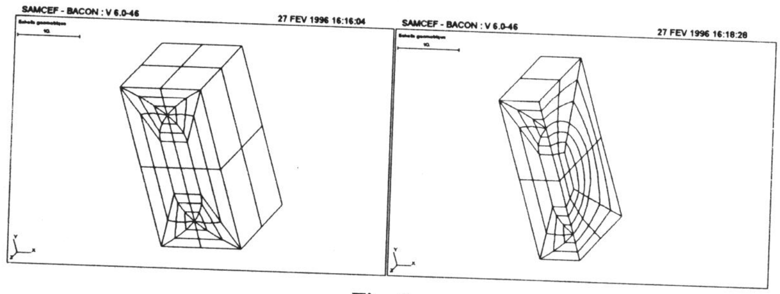

Fig. 3

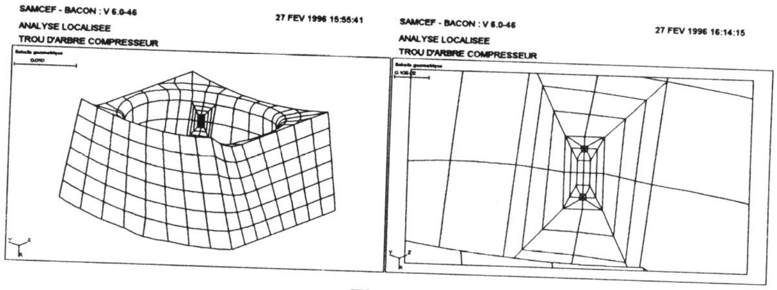

Fig. 4

Then we suppressed some elements from the initial mesh (in the vicinity of supposed crack) and we replaced them by the cracked submesh mentioned above. Since the crack can be very smaller than elements of the initial structure, an intermediate mesh can sometimes be necessary. The complete structure is presented at figure 4.

Each elastic analysis gives the stress intensity factor (we used the VCE method) along the crack front. So we can determine the crack progression per cycle for each point of the crack front (we assumed there is only mode I). We remesh the structure (only the sub-mesh is different), compute the new stress intensity factors and so on. The stress intensity factors increase as the crack grows. The obtained evolution is presented in figure 5.

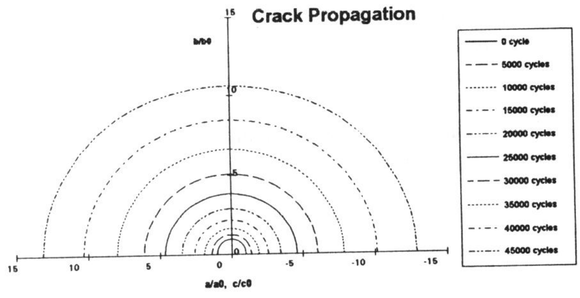

Fig. 5

We can notice that the parameters a and c progress in a quite same way. So the hypothesis that the crack is still an ellipse is good. Meanwhile, the calculation could be tainted with some errors: error due to structural zooming (this error increase with the crack dimensions), error due to the use of a linear elastic mechanics, error due to the dispersion of the Paris' Law, error due to the iterative process of integration, error on the stress intensity factor calculation, ...

LIFE TIME WITH SHOT PEENING

We know that shot peening induces compressive residual stresses at the surface of the body. The role played by these stresses on macro crack propagation can be studied by assuming that they modify the stress intensity factor at the crack tip.Two profiles for compressive residual stresses due to shot peening are presented in figure 6. These curves are given by experimental measures. We have studied the 2D problem for the life time of an edge cracked plate tension specimen.

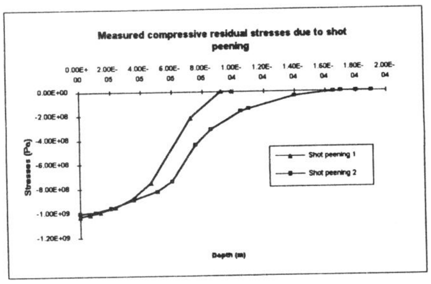

Fig. 6

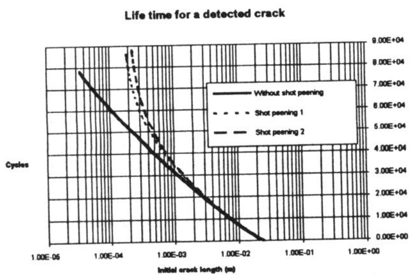

Fig. 7

According to the classical superposition principle for the linear fracture mechanics, the effective stress intensity factor at the crack tip is obtained by the addition of the stress intensity factor due to external loading K_L and the stress intensity factor due to compressive residual stress K_r.

$$K_T = K_L + K_r$$

Integrating the modified Paris law given above, we get the life time curves at figure 7. These curves give the life-time for a given initial crack length. Due to shot peening, we can see that the life time is increased for an initial detected crack.. But the main effect of shot peening is that the threshold initial value for the crack length is increased. These partial conclusions cannot be extended in a quantitative way but qualitative way to tridimensionnal problems.

REFERENCES

[1] Borlee B. "Modélisation et simulation de la croissance d'endommagement et de fissuration des structures métalliques et composites soumises à la fatigue et en présence des contraintes résiduelles", ULg - LTAS Mécanique de la Rupture, rapport semestriel, 15 octobre 1995.

[2] Harrison J. D., Leggatt R. H. "The effect of residual stresses on fracture", in "residual stresses and their effect", the welding institute, Abington Hall, Cambridge, pp. 17-20, 1981

[3] Hellen T.K. "On the method of virtual crack extensions", Int. J. for numerical methods in engineering, Vol. 9, pp. 187-207, 1975

[4] Ingraffea A.R., Manu C. "Stress-Intensity factor computation in three dimensions with quarter-point element", Int. J. for numerical methods in engineering, Vol. 15, pp. 1427-1445, 1980

[5] Knott J.F. "Effects of residual stress on fatigue crack propagation and fracture", dans ?

[6] Kunz L., Knésl Z., Lukas P. "Macroscopic residual stress distribution at a fatigue crack tip", Fatigue of engineering materials and structures, Vol. 2, pp. 279-287, 1979

[7] Metal Improvement Company, Inc. "Application du Shot Peening", Septième édition

[8] Nelson D. V. "Effects of residual stress on fatigue crack propagation", ASTM STP, Vol. 776, pp. 172-195, 1982

[9] Nguyen D.H., Morelle P. "Utilisation de la méthode EDI pour les problèmes tridimensionnels de mécanique de la rupture", Rapport int. MSM 195, Université de Liège, 1989

[10] Nguyen D.H., De Saxce G. "Modified formulea for three-dimensional direct stress-intensity factor computation", Communications in applied numerical methods, Vol. 6, 223-230, 1990

[11] Nikishkov G.P., Atluri S.N. "Calculation of fracture mechanics parameters for an arbitrary three-dimensional crack, by the 'equivalent domain integral' method", Int. J. for numerical methods in engineering, Vol. 24, pp. 1801-1821, 1987

[12] Parker A.P. "Linear elastic fracture mechanics and fatigue crack growth - residual stress effects", pp. 249-271, dans K5

[13] Scibetta M. "Modélisation et détermination des contraintes résiduelles et étude de leurs influences dans les structures fissurées", ULg - LTAS Mécanique de la Rupture, rapport semestriel, 20 octobre 1994.

[14] Wang Renzhi "Effect of residual stresses of shot peening on the fatique behaviour of a high strength steel", Fatigue of engineering materials and structures, Vol. 2, pp. 413-418, 1980