13th International Conference on CANDU Fuel
Holiday-Inn Waterfront Hotel
Kingston, Ontario, Canada, 2016August 15-18

A METHOD FOR PREDICTING THE EVOLUTION OF CRACK GROWTH IN COMPLEX STRUCTURES – A POTENTIAL APPLICATION IN SIMULATING STRESS CORROSION CRACKING

D. Wowk[1], L. Alousis[1] and P. Chan[1]
[1]Royal Military College of Canada
(PO Box 17000, Station Forces, Kingston, Ontario, Canada, K7K 7B4
613-541-6000 ext 6457; diane.wowk@rmc.ca)

ABSTRACT – Stress corrosion cracking can occur in the Zircaloy sheath surrounding nuclear fuel pellets when a critical combination of iodine concentration and stress levels are present. Linear elastic fracture mechanics can be used to predict crack growth, but in complex situations such as pellet-cladding interaction, or when thermal gradients are applied, the required stress intensity factors are not easily determined. This paper presents a method for using finite element simulations to predict the growth of crack fronts of arbitrary geometry and has a potential application in predicting stress corrosion cracking of nuclear fuel sheathing.

Introduction

Pellet-cladding interaction failures can occur in Zircaloy sheathing due to stress corrosion cracking when a thermally expanding pellet comes in contact with the sheath in an iodine rich environment. Cracks can initiate and propagate in the fuel sheath when a critical combination of iodine concentration and stress level is reached [1]. In order to design against stress corrosion cracking, crack growth must be able to be predicted for different pellet-cladding interactions such those arising from variations in power profiles, changes in chemical environment, the use of different materials and surface treatments at the pellet-cladding interfaces. Being able to predict these effects using numerical models allows changes to the design or operating conditions to be evaluated relatively quickly, and enables virtual testing to be completed prior to constructing experimental test programs.

Linear elastic fracture mechanics is often used as the basis for crack growth predictions in nuclear fuel sheathing through relationships between crack growth rates and the associated stress intensity factors (K_I) [2]. This relationship enables the rate of crack growth to be predicted as long as the local stress field in the vicinity of the crack tip is known. The stress field is influenced by such parameters as the magnitude of applied loads, residual stresses, interaction with adjacent components in contact, material properties, and the length and shape of the crack. These effects are characterized by the stress intensity factor K_I, or a geometry factor which is independent of the load magnitude. The geometry factors for some common cases with simple loading and geometric configurations such as elliptical cracks in a pressurized vessel are available in handbooks [3].

Crack growth in nuclear fuel sheathingis a complex phenomenon, and the effects of many aspects influencing the rate of crack growth such as the frictional contact between the pellet and the cladding, local changes in material properties and creep cannot be predicted using existing solutions. In handbook solutions, the material surrounding the crack front is assumed to be homogeneous and local changes in material texture or grain structure are not incorporated. In addition, standard crack front shapes are assumed and only one representative value of the stress intensity is given, although in reality the stress intensity values vary along the crack front. In the case of cracks initiating from the inner surface of the fuel sheath, an elliptical crack front shape may be used [4], but in reality stress intensity values can vary along the crack front due to local stress concentrations. In the case of corrosion, the geometry of the crack front is rarely smooth, and an elliptical crack front may not develop in fuel sheathing undergoing stress corrosion cracking. One of the proven methods for determining stress intensity factors for situations for which handbook solutions do not exist, is finite element analysis. This paper presents a method that uses successive finite element simulations to predict the shape of evolving crack fronts in metallic components.

One of the more common methods for predicting the shape of evolving crack fronts was first presented by Smith and Cooper [5] and then refined by Lin and Smith [6]. This method is based on using 3D finite element simulations to predict the distribution of stress intensity factors (K) along the crack front. The incremental growth for individual points along the crack front is determined using the Paris crack growth law in a direction normal to the crack front. The process is repeated until the crack grows to a specified length. Acceptable results have been produced when this method has been used for a variety of planar cases such as corner cracks at holes, surface cracks and center cracked specimens [6-8]. When cracks are small, they are often irregular as their initial shape is a result of their initiation process. Minute flaws in the material or the presence of corrosion can produce a crack front with severe changes in local curvature. In complex loading cases, there may be regions along the crack front that are below the threshold for propagation resulting in growth for only part of the crack [9]. Several authors have implemented spline curves in their growth programs which enable irregular crack fronts to be represented. Lin and Smith [7] demonstrated fatigue shape development of an initially irregular corner crack from a fastener hole using a cubic spline fit to 9 points.

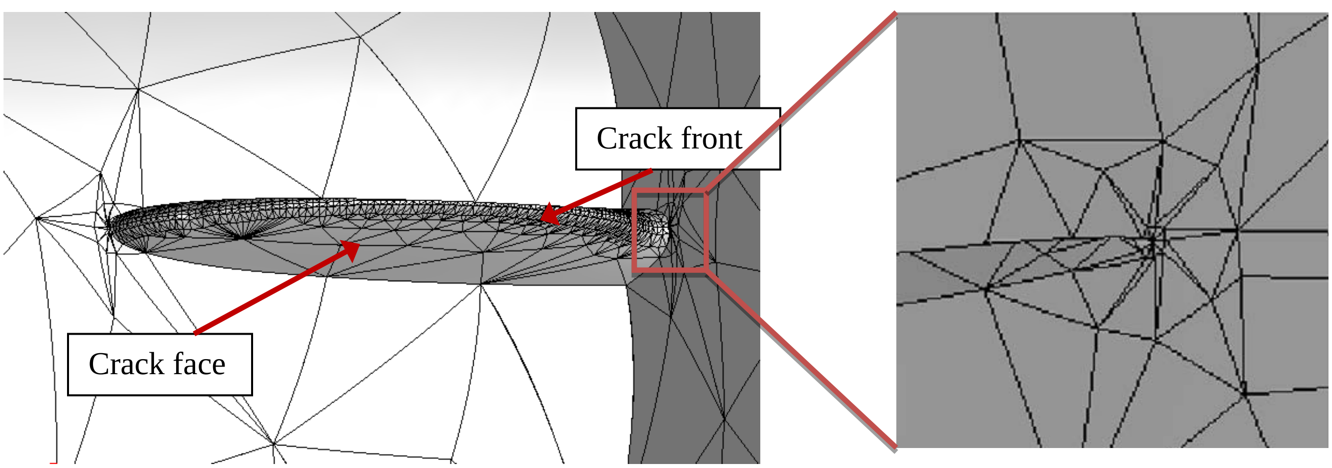
The method presented in this paper uses the commercial finite element software Stresscheck to implement an automated process of creating and propagating arbitrarily shaped crack fronts. This method has the potential to be extended to predict crack propagation in nuclear fuel sheathing subjected to stress corrosion cracking as it could account for geometric considerations such as contact and friction at the pellet-cladding interface as well as the curvature of the sheath. Material properties could be adjusted to account for the effects of changes in iodine concentration, and thermal and pressure gradients could be incorporated. Differences in material properties or loading in the vicinity of the crack tip, as well as variations along the crack front could be accounted for. An outline of the structure of the crack growth program is presented as well as a comparison between the simulation and experimental results for a test case on an aluminum sheet. Finally, preliminary predictions of the distribution of stress intensity factors along the crack front for an elliptical crack in a representative Zircaloy sheath are presented.

1. Materials and Methods

The purpose of the crack growth program presented in this paper is to predict how the shape of an arbitrarily shaped crack will develop as it propagates and to determine the distribution of stress intensity values along the crack front. Stress intensity factors can then be combined with crack growth rate data in order to predict fatigue life. This method is extremely useful for complex geometric and loading conditions where the stress intensity factors are not available in handbooks. The framework of the crack growth method is based on the work by Lin and Smith [6] but has been modified in the current paper to take advantage of features within the commercial p-method finite element software package Stresscheck and to incorporate more realistic crack front shapes. The overall process of the crack growth method can be broken down into three distinct steps; the stress field is predicted using a finite element model, the stress intensity factors along the crack front are determined, and then the new crack geometry is predicted using the Paris growth law. The details of these steps are outlined in sections 1.1 to 1.3. Section 1.4 describes the procedure for determining fatigue life based on the stress intensity factors determined using this crack growth method. These steps were automated in Stresscheck through the use of an Excel interface in conjunction with code written in Visual Basic.

1.1 Finite element prediction of stress field (Step 1)

A 3D finite element model containing the geometry of the component, the linear elastic material properties, and the external loading is generated within the commercial finite element software package, Stresscheck. The geometry of the crack itself is represented, with the crack front being defined by a spline curve fit through multiple points. This allows a crack front of any shape to be simulated. The model is meshed using a graded distribution of elements surrounding the crack front as shown in Figure 1, which follows the recommendations by Wowk et al. on crack tip meshing using p-method finite elements [10]. Figure 1a) shows how the elements are concentrated along the crack front and 1b) shows a close up of the pattern of elements surrounding the crack tip. This pattern extends along the entire crack front and isolates the stress singularity to the first row of elements that surround the crack tip. The stress distribution for the overall component as well as in the vicinity of the crack tip is then calculated.



a) b)

Figure 1: A typical crack tip mesh showing a) the distribution of elements along the crack front, b) a close up of the distribution of elements at the crack tip.

1.2 Prediction of stress intensity factors along the crack front (Step 2)

The mode 1 stress intensity values at a specified number of points along the crack front (K_i) are extracted within Stresscheck using the contour integral method for the prescribed extraction radius. An example of the distribution of extracted stress intensity factors along the crack front for an elliptical corner crack is shown in Figure 2. The location of the stress intensity factors is given by a angle between the two free surfaces of the corner crack that has been normalized by the overall crack front angle of 90°. As is often the case, the stress intensity factors are lower in the region where the crack front intersects the free surface of the component.

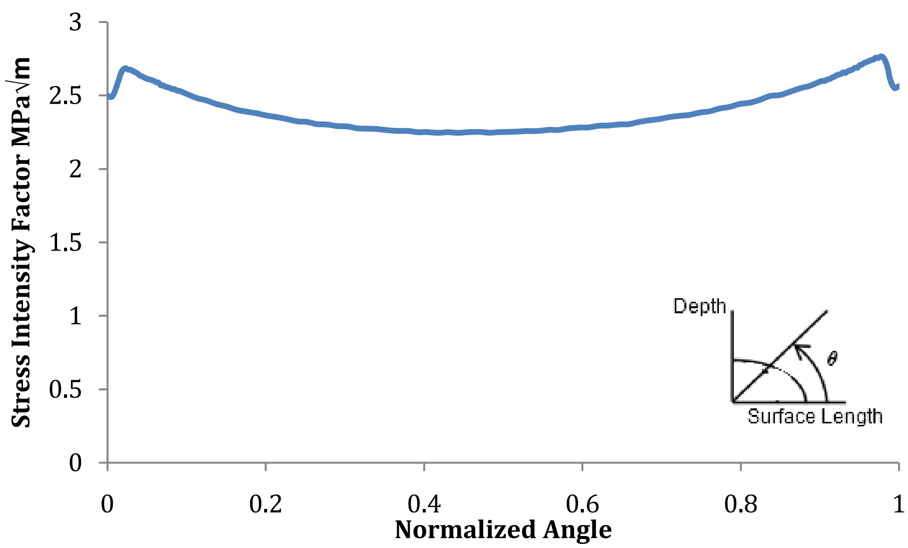


Figure 2: Distribution of stress intensity factors along the crack front of an elliptical corner crack.

1.3 Determining the new crack front shape (Step 3)

The location along the crack front with the maximum stress intensity value (K_{max}) is assumed to grow by a prescribed crack increment Δa_{max}. The growth increment for each of the remaining points along the crack front (a_i) is determined using equation (1), where *m* is a Paris law material constant. Δa_{max} is arbitrarily chosen so that a sufficient number of increments are used to accurately predict the new crack front shape and to maintain stability in the iterative solution. For example, a crack increment of 0.003" was used in the test case presented in Section 2.

$$\Delta a_i = \left(\frac{K_i}{K_{max}} \right)^m \Delta a_{max} \tag{1}$$

The growth increment for each point (Δa_i) is projected in a direction orthogonal to the crack front and is calculated using the slope between the two adjacent points. The coordinates of the points along the new crack front are determined and a spline is fit through these points. This process is outlined in Figure 3.

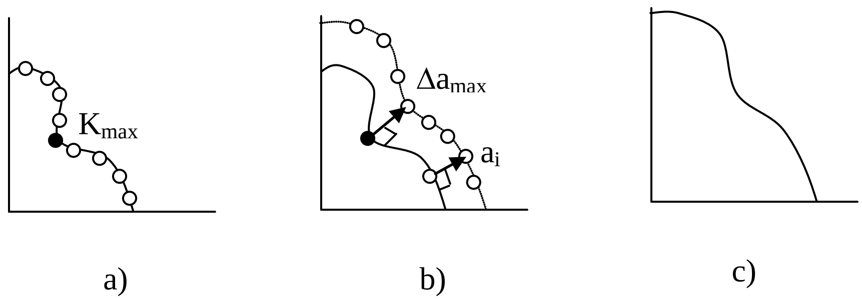


Figure 3: Stages of the crack growth method. a) The K values are extracted for a specified number of points along the crack front. b) Each point is moved in a direction perpendicular to the curve by a length given by equation (1). c) The new crack front geometry is fit to a spline curve.

1.4 Determining fatigue life

The three steps of the crack growth method (sections 1.1-1.3) are repeated using the updated crack front geometry, until the final prescribed crack length is reached. The total number of loading cycles needed to reach the final specified crack length is then determined using crack growth rate data from material testing.

The number of loading cycles (dN) to grow each crack iteration by the length Δa is determined using equation (2). C and m are Paris law material constants and are determined by fitting experimental test data to a log graph of da/dN versus ΔK.

$$dN = \frac{\Delta a}{C\Delta K^m} \qquad (2)$$

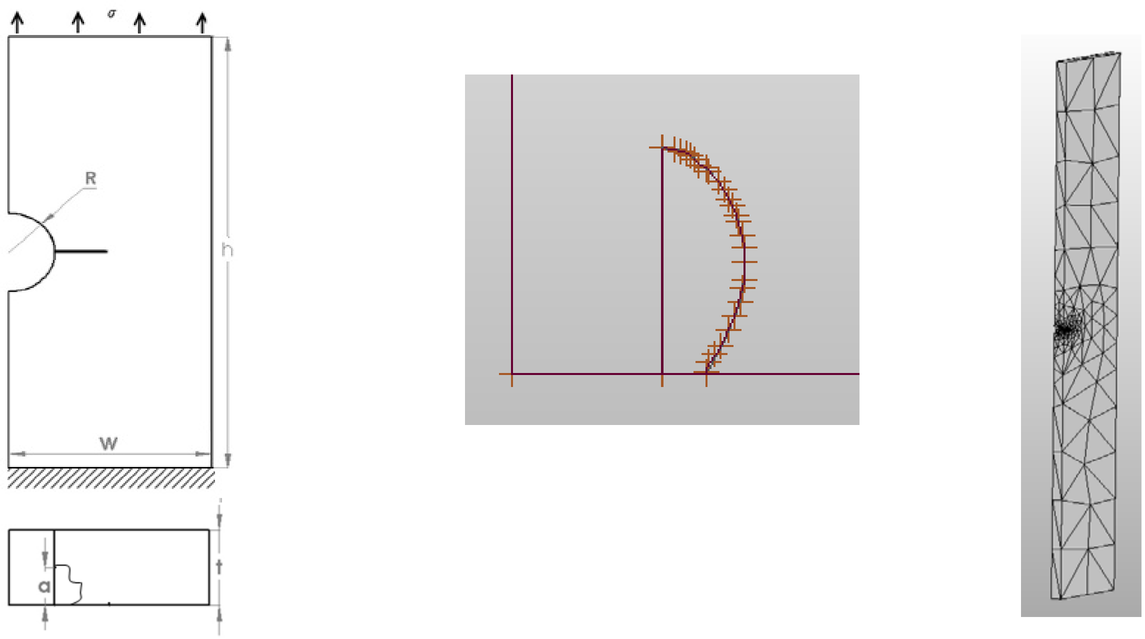
The number of cycles for each crack growth increment (Δa) is then summed between the initial crack length and the final prescribed crack length to determine the total number of loading cycles or fatigue life.

2. Test Case

The test case used to evaluate the proposed crack growth method corresponded to an experimental test on an aluminum plate with a corner crack at a notch, performed as part of a collaborative project between DRDC and NRC [11].

2.1 Model description

The overall geometric configuration is depicted in Figure 4 a), with the plate having a length of h = 12”, a width of w = 2”, a thickness of t= 0.253” and a notch of radius R = 0.067”. 40000psi was applied to one end, while the other end was constrained from moving in the lengthwise direction, but was allowed to contract laterally. Material properties of E = 10200ksi, μ = 0.33, and the Paris material constant of m = 3.4488 were used to represent 7075-T3. The crack front initiated as a corner crack emanating from the notch, and the shape was provided as a series of 28 points as shown in Figure 4b) [11]. The overall mesh is shown in Figure 4c), and the program was run using crack growth increments of Δa = 0.003” until the crack developed into a through thickness crack.



a) b) c)

Figure 4: Test case, a) overall geometric configuration, b) initial crack geometry and c) overall mesh.

2.2 Test case results

A total of 52 iterations were required to grow the crack to through thickness, and took approximately 24 hours on a desktop computer with 12GB of RAM and 4x3.00GHz AMD Phenom II cpus. Results were evaluated in terms of a comparison with the experimentally produced crack shape.

Figure 5 shows the shape of the predicted crack front compared with the experimental results for 7 of the iterations. They are in good agreement to within 10% percent of each other and the overall shape of the crack is captured. The predicted crack front consistently shows less curvature near the free surfaces and an overprediction of the crack length along the bore of the notch. This may be a result of excluding the effects of crack closure, thus predicting the crack to grow faster than it should at the surfaces. Figures 6 a) and b) show close ups of the prediction, experimental results and the corresponding elliptical fit for three different crack lengths. It is clear that the ellipse is not a good representation for irregular crack fronts (Figure 6a) and it cannot capture the tunneling that occurs at the ends of the crack (Figure 6b). This comparison shows that using simplified geometric fits for the shape of the crack front may not be representative of reality and may lead to inaccurate predictions of fatigue life.

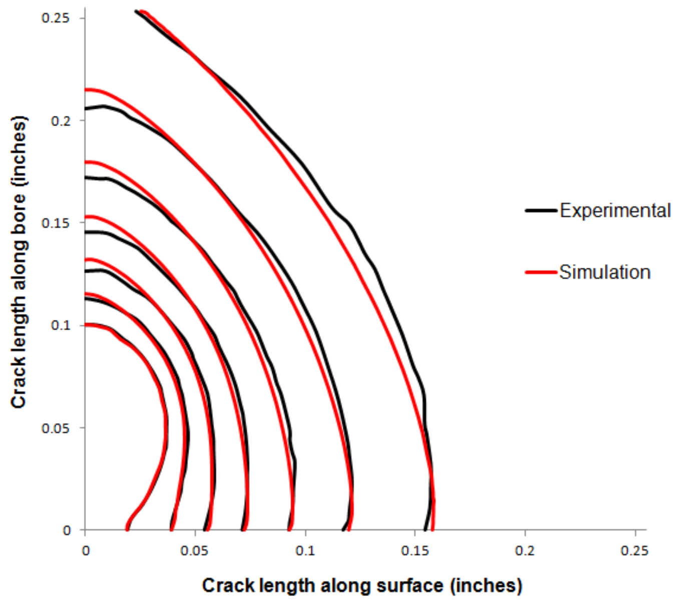


Figure 5: Comparison between the predicted and experimental crack shapes for the test case.

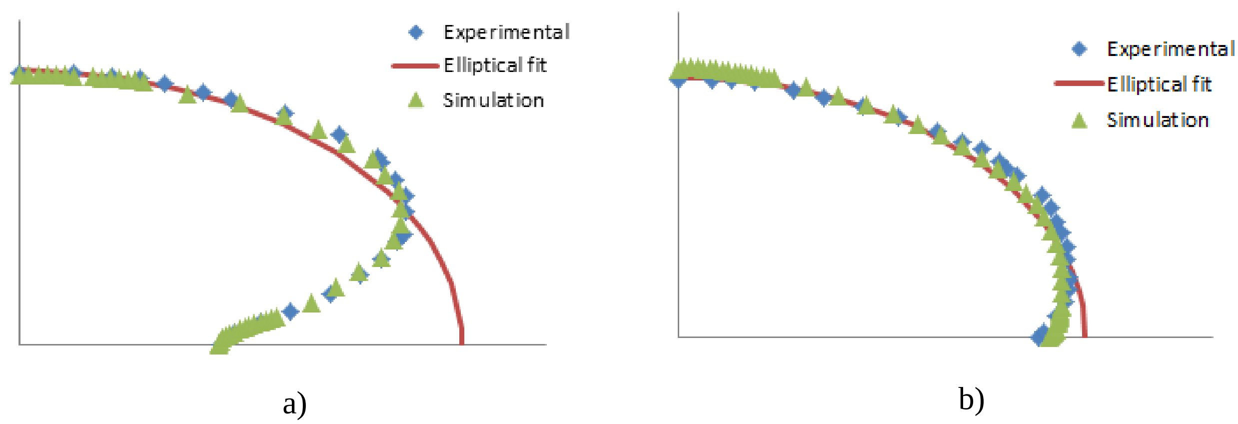


Figure 6: Comparison in the crack shape for a) the initial crack, b) an intermediate crack length.

3. Application to fuel sheath geometry

The test cases presented in section 2. was originally chosen for an aerospace application and therefore has a greater thickness than a typical fuel sheath. In order to demonstrate the methods potential for use in nuclear applications,a preliminary finite element model was created to determine the distribution of stress intensity values along an elliptical crack front in a representative fuel sheath with a radius of 6.5mm and a thickness of 0.41mm. The length of the sheath was limited to 25mm in the simulation and the overall model configuration is shown in Figure 7 a). Linear elastic properties for Zircaloy-4 of E = 100Mpa and μ = 0.32 were assigned. An arbitrary internal pressure of 100kPa was applied to the inside of the tube because for linear analyses, the stress intensity values scale linearly with the magnitude of the load. The crack was assumed to have an elliptical shape, with a depth of 0.3mm and a width of 1mm. Figure 7 b) shows a close up of the crack and the crack tip mesh.

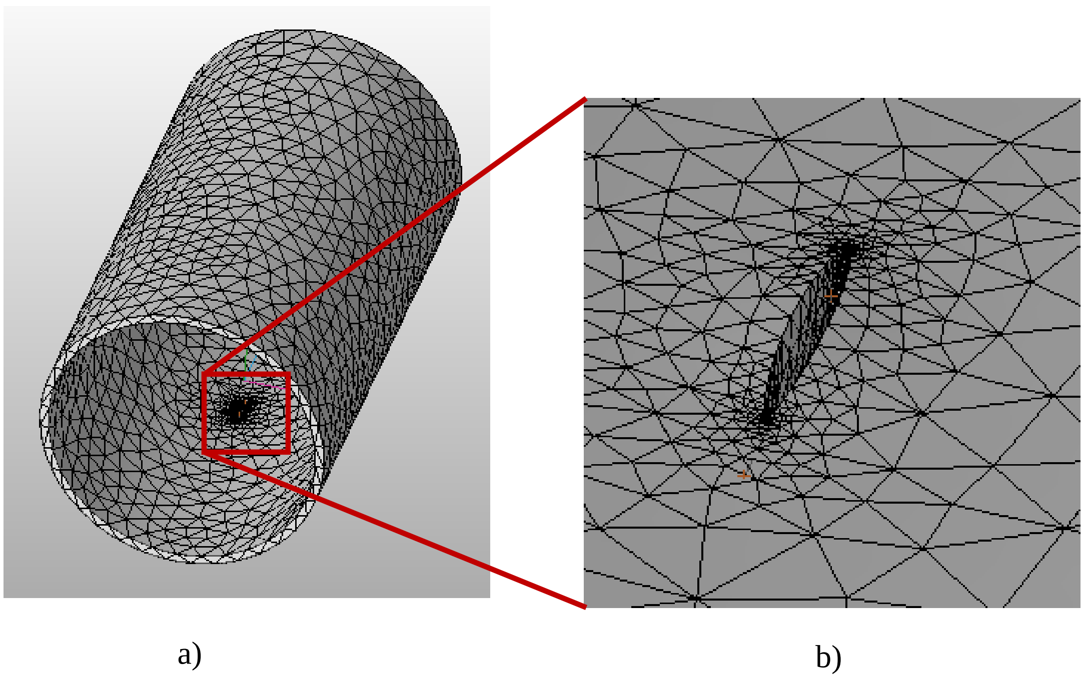


Figure 7: Finite element model of a representative fuel sheath with an elliptically shaped surface crack. The sheath was loaded with an internal pressure.

The stress intensity values were extracted along the crack front, and their distribution is shown in Figure 8. The stress intensity factors are lowest at the free surfaces and at the middle of the crack front. The difference between the values at the middle of the crack and the maximum value is 10% highlighting the fact that the stress intensity values are not constant along the crack front. The results from the sheath geometry were also compared to the results using a simplified flat plate with the same crack geometry in Figure 8. It can be seen that using a flat plate instead of the curved sheath can result in an overprediction of 5% in the stress intensity values. This comparison shows that the physical geometry of the fuel sheath should be represented in crack growth simulations.

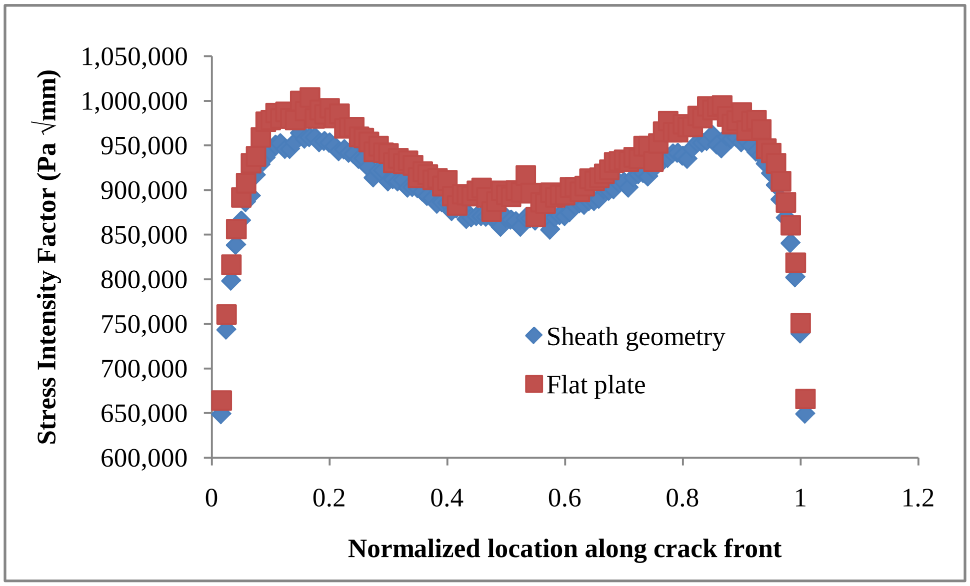


Figure 8: Distribution of stress intensity factors along an elliptical crack front for a representative sheath geometry and a flat plate.

4. Conclusions

The proposed crack growth program used a spline to represent the geometry of an irregular crack front and predictions for a corner crack originating at a notch were in agreement with experimental results. The following conclusions can be made:

- More accurate predictions of the geometry of an irregular crack front are achieved using a spline rather than standard geometrical fits.

- The proposed method can capture the overall trend of tunneling at free surfaces as well as local changes in curvature.

- Predictions of the crack front geometry for a corner crack at a notch corresponded well with experimental data for the test case considered here in terms of overall crack shape.

- This method has the potential to be applied to more complex 3D configurations such as pellet to sheath interaction in the context of stress corrosion cracking in nuclear fuel elements.

5. References

[1] P. S. Sidky, "Iodine stress corrosion cracking of Zircaloy reactor cladding: iodine chemistry (a review)",Journal of Nuclear Materials, 256, 1998. pp. 1-17.

[2] IAEA-TECDOC-1185, "Iodine induced stress corrosion cracking of Zircaloy fuel cladding materials", 2000.

[3] D.P. Rooke, D.J. Cartwright, "Compendium of Stress Intensity Factors", Hillingdon Press, London, 1976.

[4] S. Y. Park, J. H. Kim, M. H. Lee, Y. H. Jeong, "Stress-corrosion crack initiation and propagation behaviour of Zircaloy-4 cladding under an iodine environment", Journal of Nuclear Materials, 372, 2008. pp. 293-303.

[5] R. A. Smith, J. F. Cooper, "A finite element model for the shape development of irregular planar cracks", International Journal of Pressure Vessels and Piping, 36, 1989. pp. 315-326.

[6] X. B. Lin, R. A. Smith, "An improved numerical technique for simulating the growth of planar fatigue cracks", Engineering Fracture Mechanics, 20, 1997. pp. 1363-1373.

[7] X. B. Lin, R. A. Smith, "Fatigue shape analysis for corner crack at fastener holes", Engineering Fracture Mechanics, 59, 1, 1998. pp. 73-87.

[8] R. Branco, F. V. Antunes, "Finite element modelling and analysis of crack shape evolution in mode-I fatigue middle cracked tension specimens", Engineering Fracture Mechanics, 75, 2008. pp. 3020-3037.

[9] M. Heyder, G. Kuhn. "3D fatigue crack propagation: experimental studies", International Journal of Fatigue, 28, 2006. pp. 627-634.

[10] D. Wowk, K. Gamble, R. Underhill, "Influence of p-method finite element parameters on predictions of crack front geometry". Finite Element Analysis in Design, 73, 2013. pp. 1-10.

[11] R. H. Desnoyers, M. Liao, Crack growth measurement report on SENT coupon J407-1E-2. LM-SMPL-2011-0202, 2011.

[12] M. D. Gilchrist, R. A. Smith, "Finite element modelling of fatigue crack shapes". Fatigue and Fracture of Engineering Materials and Structures, 14, 6, 1991. pp. 617-626.