Int. J. Adv. Eng. Pure Sci. 2020, 32(3): 295-302
DOI: 10.7240/jeps.651164

RESEARCH ARTICLE / ARAŞTIRMA MAKALESİ

Investigation of Mixed-Mode Fatigue Crack Growth Phenomenon with a New Computational Procedure

Karışık Modlu Yorulma Çatlak Büyümesi Olayının Yeni Bir Hesaplamalı Yöntem ile İncelenmesi

Murat SARIBAY [1]

[1] *Istanbul Bilgi University, Faculty of Engineering and Natural Sciences, Department of Mechanical Engineering, 34060, Eyup, Istanbul.*

Abstract
Analysis of 3-D fatigue crack growth problems with mixed-mode loading has always been an interesting area in the field of fracture mechanics. Fracture failure under the influence of fatigue loading has been a common experience for various industries' products, such as aerospace and automotive components. Any possible failure in these structures can result in high damage to these components or a serious risk for people's health. The analysis of such failures may involve great challenges and complexities for obtaining the accurate solution. The complexities of the problem may not only be related to the loading type, but also to the specific geometry itself. Such problems are hard and costly to analyze with experimental methods. Therefore, it is important to establish the theoretical aspects of the process initially, and then having a computational procedure to solve the problem at hand. The crack growth law used in this procedure is NASGRO-type, and determination of the propagation angle is based on the maximum hoop stress criterion. Hypermesh and ANSYS APDL software are benefited during preprocessing of the propagation steps and application of submodeling procedure. Solution of the problem is performed with FRAC3D program, its enriched element methodology and newly implemented tools for crack growth. A specific example that includes cracking within an aircraft engine compressor blade is shown for demonstration purpose.
Keywords: Mixed-mode fatigue crack propagation, NASGRO-type crack growth, enriched elements, submodeling

Öz
Üç boyutlu ve karışık modlu yorulma çatlak büyümesi problemleri kırılma mekaniği açısından her zaman ilgi çekici bir alan olmuştur. Hava-uzay ve otomotiv gibi endüstrilerdeki ürünlerin yorulmaya bağlı olarak kırılma sonucu meydana gelen hataları oldukça yaygındır. Bu ürünlerdeki herhangi bir hata veya başarısızlık, komponentlerine yüksek bir hasar veya insan sağlığı açısından yüksek risk olarak sonuçlanabilir. Bu tür hata veya arızaların analizi ile doğru sonucu bulmaya çalışmak belli bir zorluk ve karmaşıklık içerebilir. Problemlerdeki karışıklıklar hem yükleme çeşidiyle hem de belli bir geometri ile alakalı olarak ortaya çıkabilir. Bahsi geçen problemlerin deneysel yöntemler kullanarak analizi zor ve maliyetli olabilir. Bu sebeple, başlangıçta sürecin teorik yönlerini kurmak ve sonrasında çözüm için hesaplamalı bir yöntem oluşturmak eldeki problemin çözümü açısından önemlidir. Çalışmada kullanılan çatlak büyüme kanunu NASGRO modelidir ve ilerleme açısının belirlenmesi de maksimum çevresel gerilme kriteriyle gerçekleştirilmektedir. İlerleme aşamalarının ön-işlem sürecinde ve alt modelleme tekniğinin faydalanılmasında Hypermesh ve ANSYS APDL programları kullanılmaktadır. Problemin çözüm kısmı ise FRAC3D programı ve bünyesindeki zenginleştirilmiş elemanlar ile yeni geliştirilmiş çatlak büyüme araçlarıyla yapılmaktadır. Örnekleme amacıyla uçak motorundaki kompresörün kanat kısmındaki çatlakların incelenmesine yer verilecektir.
Anahtar Kelimeler: Karışık modlu yorulma çatlak ilerlemesi, NASGRO büyüme modeli, zenginleştirilmiş elemanlar, alt modelleme

I. INTRODUCTION

The solution of three-dimensional fatigue crack growth problems with finite element methodology is a challenging task for most researchers. As an extension from previous two-dimensional computational procedures, three-dimensional crack propagation has been the subject of many studies especially beginning from two decades ago. A pioneering study in this field was introduced by Dhont [1], in which he presented a simple algorithm for planar crack growth (only mode I) of a 3-D structure. Following that, other notable works were published to demonstrate different techniques related to 3-D crack growth, such as well-known FRANC3D program [2] and ZENCRACK [3]. The latter one used a card-block approach, and the crack fronts were

Corresponding Author: Murat SARIBAY, Tel: +90 212 3117372, e-posta: murat.saribay@bilgi.edu.tr
Submitted: 26.11.2019, **Revised:** 01.04.2020, **Accepted:** 04.06.2020

introduced to the uncracked mesh during every step of the propagation. Besides those already mentioned, Schollman et. al. [4] gave an algorithm for the solution of fatigue crack growth problems with modified crack closure method, and referred to the program as ADAPCRACK3D. Another well-known solution is the extended finite element method [5], which mainly uses tetrahedron elements for meshing purposes. Dundar and Ayhan [6] contributed to this field with a notable study, and they referred to their procedure as 'Fracture and Crack Propagation Analysis System (FCPAS)'. However, their models were created with tetra elements and it was only applied to simple geometries and loading.

In order to predict the crack propagation path of a cracked structure, one has to benefit from the notions of a crack growth law and a theory that determines the propagation direction. In addition, elements with special formulations have to be created around the crack region to account for the singularities present at the crack tip. Sih [7] introduced his strain energy density theory, which didn't require any special meshing procedure around the crack front. His solution methodology seemed to be easily applicable, since the calculation of stress intensity factors was not necessary. However, many researchers still rely on Paris-Erdogan [8] crack growth law for the simulation of crack growth phenomenon. NASA [9] has extended Paris-Erdogan law to a more general expression, and presented NASGRO-type crack growth law. To determine the crack propagation angle, maximum hoop (circumferential) stress criterion [10] is still commonly used. This criterion is an effective way to determine the propagation angle during growth simulation.

Enriched element methodology used with FRAC3D program [11, 12] is an effective tool for the analysis of fracture problems. Fracture problems are solved accurately with less number of elements around the crack front, when compared to the other programs available. The reason for the success of FRAC3D program can be attributed to its special formulation for so-called enriched elements created around the crack front. The stress intensity factors are directly included as unknowns within the displacement vector. In [13], three-dimensional fatigue crack growth with enriched elements is discussed in detail, but example problems with tetra element models were accompanied with unidirectional and uniform load. Hence, details shown in [13] did not include any special computational methodology that may deal with a real life problem at hand; in other words, a complex problem in terms of geometry and loading. Besides that, only Paris-Erdogan law was considered as the crack growth law.

The objective of this study is to demonstrate a new methodology that can simulate mixed-mode fatigue crack growth phenomenon for structures with complex geometry and loading. A more detailed crack growth law, i.e., NASGRO-type growth, is used in this procedure. The propagation direction is determined with respect to the maximum hoop stress criterion. The calculation of stress intensity factors and growth parameters is performed with a newer version of FRAC3D program. Finite element modeling and remeshing are handled with a special procedure, which includes benefiting from Hypermesh and ANSYS APDL software. Submodeling approach is used for the creation of local models with hexahedron elements. The final part is the solution of the problem with FRAC3D program and obtaining stress intensity factors. The newer version of FRAC3D program includes the growth algorithm based on the propagation theories, which are detailed in the following section. As the final output, FRAC3D gives the nodal coordinates of the next steps' crack front to continue the process.

II. MATERIALS AND METHODOLOGY

In this section, theoretical aspects and the methodology for the analysis of mixed-mode fatigue crack growth problems will be summarized. Following that, the finite element procedure that was developed for the current objective will be described.

2.1 Calculation of Stress Intensity Factors

For any computational procedure that aims to perform a crack growth analysis, the initial task is to determine the stress intensity factor solutions accurately. To do that, finite element methods have been shown to be a useful tool. Since a fracture analysis requires to deal with asymptotic singularities present at the crack tip, the correct evaluation of these parameters is definitely needed. For this purpose, a Fortran 90 software, called FRAC3D, was developed by the author and his colleagues at Lehigh University ME&M Department [11, 12] and has been a remarkable finite element program for the solution of fracture mechanics problems with high complexity (in terms of analysis type, loading, material behavior or geometry). FRAC3D benefits from special enriched elements that account for the asymptotic singularities at the crack tip. These asymptotic fields may be within isotropic or anisotropic materials, or at bi-material interfaces. A newer version of this finite element (FE) code have recently been developed, and it includes the evaluation of crack growth parameters, in addition to the common variables determined for a classical fracture problem, such as stress intensity factors, strain energy release rates, energy density function or the J-integral.

Being an effective tool for the solution of various fracture mechanics problems, FRAC3D benefits from specialized (enriched) elements. Figure 1 shows an example FE model with these enriched elements. Transition elements depicted are used to satisfy displacement compatibility between enriched and regular elements. The main advantage of FRAC3D over other commercial software is its capability to give the accurate solution with less number of elements used around the crack region, which results in a reduction in total number of DOFs for the entire model. This difference reduces the computational times significantly.

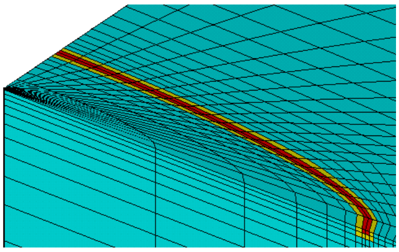


Figure 1. Enriched (red) and transition (yellow) elements along the crack front.

The formulation of enriched elements includes the closed form asymptotic field for crack tip displacements, in addition to the usual polynomial interpolation function. Besides that, the stress intensity factors are given as unknowns within the formulation. As a natural result of this representation, stress intensity factors are calculated directly within the displacement array. In enriched element formulation, element displacements take the following form [11]:

$$u(\chi,\eta,\rho)=\sum_{j=1}^{r} N_j(\chi,\eta,\rho)u_j+\{K_I(\Gamma)F_1(\chi,\eta,\rho)+K_{II}(\Gamma)G_1(\chi,\eta,\rho)+K_{III}(\Gamma)H_1(\chi,\eta,\rho)\}Z_0(\chi,\eta,\rho) \quad (1)$$

$$v(\chi,\eta,\rho)=\sum_{j=1}^{r} N_j(\chi,\eta,\rho)v_j+\{K_I(\Gamma)F_2(\chi,\eta,\rho)+K_{II}(\Gamma)G_2(\chi,\eta,\rho)+K_{III}(\Gamma)H_2(\chi,\eta,\rho)\}Z_0(\chi,\eta,\rho) \quad (2)$$

$$w(\chi,\eta,\rho)=\sum_{j=1}^{r} N_j(\chi,\eta,\rho)w_j+\{K_I(\Gamma)F_3(\chi,\eta,\rho)+K_{II}(\Gamma)G_3(\chi,\eta,\rho)+K_{III}(\Gamma)H_3(\chi,\eta,\rho)\}Z_0(\chi,\eta,\rho) \quad (3)$$

where u_j, v_j and w_j represent the unknown nodal displacements, N_j are commonly known element shape functions in terms of local coordinates, and K_I, K_{II} and K_{III} are mode I, II and III stress intensity factors, respectively. In Equations (1) - (3), F_i , G_i and H_i are the asymptotic displacement functions and Z_0 are the "zeroing" functions that account for the transition elements to ensure compatibility.

For a more detailed description of enriched element formulation, [11] and [12] are highly referenced.

2.2 Crack Growth Model and Propagation Direction

In order to simulate a fatigue crack growth problem, a propagation law is necessary. For this purpose, various laws were introduced to the literature in the past. Most of these fatigue crack growth rate laws were based on the Paris-Erdogan equation, where the growth rate was given as a function of ΔK. The Paris-Erdogan crack growth rate equation can be expressed as [8]

$$\frac{da}{dN}=C(\Delta K)^n \quad (4)$$

where, a is the crack length, N the number of cycles, and C and n are empirically determined constants. In this study, a more sophisticated crack growth rate expression, NASGRO law that was developed by NASA, will be used, i.e. [9],

$$\frac{da}{dN}=C\left[\left(\frac{1-f}{1-R}\right)\Delta K\right]^n\frac{\left(1-\frac{\Delta K_{th}}{\Delta K}\right)^p}{\left(1-\frac{K_{max}}{\Delta K_c}\right)^q} \quad (5)$$

In Equation (5), C, n, p and q are material constants, and f is the crack opening function. While ΔK_{th} is the threshold stress intensity factor, ΔK_c is the critical stress intensity factor. In the same equation, the only quantities which should be computed for a specific geometry and loading are ΔK, i.e., stress intensity factor range $(K_{max}-K_{min})$, and R (K_{min}/K_{max}) as being the stress ratio. Note that K_{max} and K_{min} correspond to the stress intensity factors for maximum and minimum loading cases, respectively.

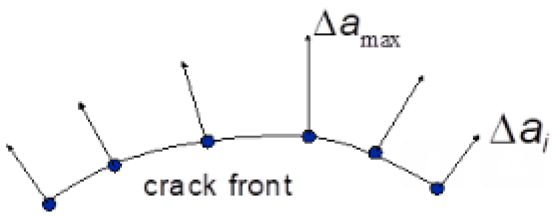


Figure 2. Advancement of crack front based on a given fatigue crack growth rate law.

For the 3-D crack front shown in Figure 2, where the stress intensity factors are known at all points along the crack front, the node with the maximum computed ΔK value (ΔK_j) is virtually advanced on a plane by a small distance (Δa_{max}). The local increment in the number of fatigue cycles for this value of Δa_{max} can be approximated (considering Equation (5)) by

$$\Delta N_j=\frac{\Delta a_{max}}{F(\Delta K_j)} \quad (6)$$

where

$$F(\Delta K_j)=C\left[\left(\frac{1-f}{1-R}\right)\Delta K_j\right]^n\frac{\left(1-\frac{\Delta K_{th}}{\Delta K_j}\right)^p}{\left(1-\frac{K_{max}}{\Delta K_c}\right)^q} \quad (7)$$

As a result, the advancement of all other nodes on the crack front is obtained from Equation (5), i.e.,

$$\Delta a_i = \Delta N_j . F(\Delta K_i) \quad (8)$$

where $F(\Delta K_i)$ is computed for the i[th] node using Equation (7) with ΔK_i substituted for ΔK_j. In this manner, the evolution of the crack front can be tracked by moving the crack front nodes to new positions given by Δa_i and new stress intensity factors can be computed to determine the new crack front geometry. In a small-scale 3-D finite element simulation, the crack tip nodes can be moved through small displacements without remeshing. However, once the crack tip elements become severely distorted, remeshing of the geometry around the crack front is required.

Erdogan and Sih [10] showed that if the crack is subjected to mixed-mode loading, the crack will not remain planar, and it will turn out-of-plane by propagating in the direction of the maximum circumferential stress at the crack tip. This is shown schematically in Figure 3.

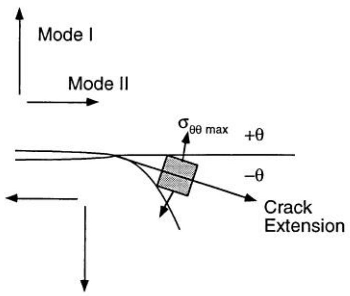


Figure 3. Crack growth under mixed-mode conditions in the direction of the maximum hoop stress [10].

The determination of the propagation direction demonstrated in [10] was used repeatedly in proceeding studies in this field. Erdogan and Sih also stated a mathematical expression for the crack propagation angle, which was given as

$$\left(\tan\frac{\theta_p}{2}\right)_{1,2} = \frac{1}{4}\frac{K_I}{K_{II}} +/- \frac{1}{4}\sqrt{\left(\frac{K_I}{K_{II}}\right)^2 + 8} \quad (9)$$

where K_I and K_{II} are the mode I and mode II stress intensity factors, respectively. This expression is usually referred to as maximum hoop or circumferential stress criterion in the literature.

For mixed-mode loading conditions, an equivalent form of the stress intensity factor representations is required to be evaluated within the crack growth law. This form can be expressed as [13]

$$K_{eqv} = \frac{K_I}{4}\left(3\cos\left(\frac{\theta_p}{2}\right) + \cos\left(\frac{3\theta_p}{2}\right)\right) - \frac{3K_{II}}{4}\left(\sin\left(\frac{\theta_p}{2}\right) + \sin\left(\frac{3\theta_p}{2}\right)\right) \quad (10)$$

which is a function of the mode I and mode II stress intensity factors and the propagation angle. Using Equation (10), the corresponding equivalent stress intensity factors for maximum and minimum loadings (K_{max} and K_{min}) can be calculated. It should be noted that, the equivalent stress intensity factor expression given in Equation (10) does not include any mode III effect. Taking K_{III} or anti-plane shear into account may not be necessary for the current problem, but is considered as a follow-up study.

In a crack growth simulation procedure, the new geometric form of the crack front after each step should be determined mathematically. This can be realized with the calculation of each crack front node's spatial advancement, with the aforementioned technique. In addition, advanced crack front geometry requires the definition of a new local coordinate system, as shown in Figure 4. In this figure, solid line represent the crack curve formed by the nodes along the front, and dashed line is the propagated crack front. The initial and propagated local coordinate systems are also depicted. By using Δa_i given in Equation (8) and the propagation angle θ_p calculated by Equation (9), the local coordinates of the new crack front nodes are determined. More details for this procedure is explained in [13], and will not be restated here.

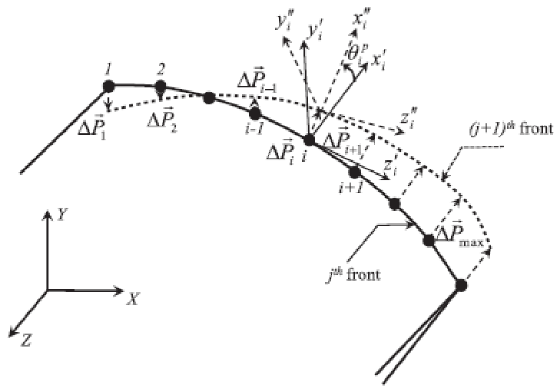


Figure 4. Crack front advancement for a 3-D crack (advanced local CS is also shown) [13].

Having the appropriate crack growth model, propagation direction theory, and the crack front advancement technique discussed, it might be a good idea to continue with the details of the finite element procedure. This subject will be explained in the next section.

2.3 Finite Element Modeling and Meshing Procedure

Finite element methodology is an effective tool for the analysis of fracture problems. An experienced finite element analyst should be aware of the importance for choosing his/her model parameters, such as the element type selection or model size. Using more elements can guarantee the accuracy, however, increases the computational times. On the other hand, FE models with tetrahedral elements are easier to create when compared with hexahedron models, but it is a commonly known fact that using hexahedron elements yields more accurate results in fracture problems. Commercial finite element tools such as ANSYS or ABAQUS are not very reliable for the analysis of cracked structures with complexity.

The analysis of complex problems may require a special technique called ‘submodeling approach’. This complexity may be in the form of geometry, loading or analysis type. It is even possible that more than one of these complexities may be present at the same time. The analysis of a cracked model under the effect of fatigue mixed-mode loading is an example for these types of problems. If hexahedron elements are used around the crack region and for the entire model, the computational issues become significant. Therefore, the submodeling approach may be benefited to provide an optimum solution to the problem at hand. In this approach, the uncracked global model is run with a relatively coarse mesh. A separate local model is also prepared, with model boundaries being contained within the global model. The local model includes the physical model, and has a finer mesh size than the global model itself. In that way, sufficient mesh size to ensure the accuracy of the cracked region is satisfied. The communication between the global and local models is provided with temperature and cut boundary displacement interpolations. To realize this objective, mesh sizes of global and local models do not need to match. That is essential for applying this approach to the solution of structural problems.

Besides the information given in the preceding paragraphs, the simulation of crack growth phenomenon requires the application of the theories and laws as stated in the previous section. As a result, a new methodology was implemented to perform crack propagation simulation. The new procedure involves the following steps to handle the problem at hand:

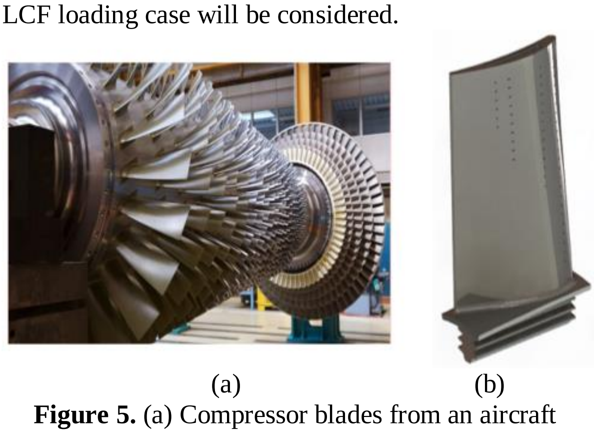
- Creation of the global FE mesh with Hypermesh program [14],
- Analysis of the global model with ANSYS Mechanical APDL [15],
- Creation of the local FE mesh with Hypermesh program (with the aid of TCL scripting tools),
- Applying the submodeling procedure with ANSYS to extract the loading and boundary conditions information for the local model,
- Execution of FRAC3D with local model to perform growth algorithm to determine the new crack front nodes’ coordinates.

The first two steps given above are only performed once during the entire simulation. At the end of the last step, remeshing is done in Hypermesh for the newly created model. And the procedure repeats itself by applying the submodeling procedure and running FRAC3D again. There’s one important point to mention at this point; FRAC3D has a notable advantage over other growth analysis programs; an outcome related to the mesh size around the crack front. Most of the finite element programs that are specialized in the solution of fracture problems require a lot of elements around the crack front to obtain correct stress intensity factors. However, the same DOF issue is not typical for FRAC3D program; you may use much less number of DOFs, and still keep the accuracy in desired level. The success of FRAC3D program for this matter is related to the enriched element formulation embedded within the software.

III. RESULTS AND DISCUSSION

The theories and methodology given in the previous section will be used to solve the current example model, i.e., a Ti-6Al-4V compressor blade that forms an integral part of the Bladed Drum (BLUM) compressor assembly.

Figure 5(a) shows a typical compressor of an aircraft engine, and multiple blades attached to the compressor. These blades have a curved and complex geometry, and may be subjected to low (LCF) and high cycle fatigue (HCF) loadings. LCF loading may include centrifugal loading, thermal stresses and airfoil pressure. As an addition to those, HCF loading is assumed to vary around LCF fatigue, and seen as the result of vibrational loads. In current analysis, only LCF loading case will be considered.



(a) (b)

Figure 5. (a) Compressor blades from an aircraft turbine engine [16], (b) CAD model of the blade [17].

Considering the blade geometry given in Figure 5(b) along with LCF loading explained above, possible cracking may occur at high stress regions at leading and trailing edges of the blade. These regions maybe approximately around the bottom of the blade section, which are close locations to the center of the compressor, in other words, to the DRUM region. In such a case, the cracks formed may propagate to the DRUM region, which may end in a serious damage to the functionality of the compressor. Hence, it is essential to determine the propagation directions of such cracks. Figure 6 depicts the high stress regions on the trailing edge of the current blade geometry under the effect of LCF loading. As a result, it is a fundamental requirement to have a crack propagation procedure to analyze this problem with complex geometry and loading. It will be described that there are many criteria that affects the solution of such problems both effectively and accurately.

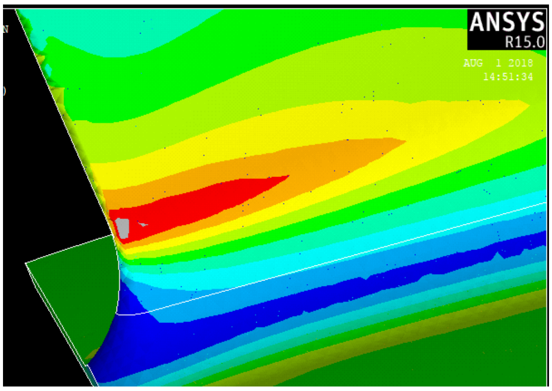


Figure 6. Maximum Von Mises stress region on the trailing edge of the blade.

Before going further with the results of the analysis, it is significant to give some details about the blade's finite element model. On the left of Figure 7, the submodel or local model that contains the initial crack is shown. The close-up view of the global model around the trailing edge of the blade (local model embedded just for visualization purposes) is demonstrated on the right of the same Figure. Both global and local models were created with hexahedron elements. It is harder to mesh structures with hex elements when compared to models with tetra elements, especially for the examples with high geometric complexity. Besides that, the solution of fracture problems requires a sufficient mesh density around the crack region, not to lose from the accuracy of the results. This is an expected feature of cracked structures, due to the singularities present at the crack tip. To ensure sufficient density around the crack region, most of the fracture software uses a high number of elements around the crack region. In addition, a typical crack growth simulation may require more than 100 steps to reach to the targeted crack length. That may bring very long computational times, which will be an important issue for the analyst.

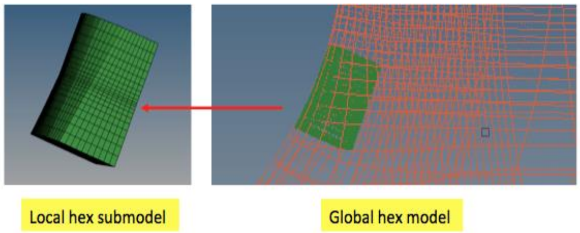


Figure 7. Local and global models.

In contrast to the various fracture software that are capable of solving mixed-mode fatigue crack growth problems, FRAC3D uses less number of DOFs around the crack front, meaning a low element density for the same region. In that way, the same level of accuracy can still be attained with FRAC3D's theory and tools. The local model that includes the initial crack (Figure 7), can be detailed more to understand this phenomenon better. To do that, refer to Figure 8 given below, which shows the bottom half of the local FE model with initial crack size. The blue elements represent the crack surface, which is detached from the top half of the local model. It is apparent that the initial crack form is assumed to have a semi-circular shape. So the curved crack front's nodal locations can be addressed in angles, from 0 degrees to ~180 degrees. Besides that, the elements that are closer to the crack front are smaller in size, compared to the ones in outer regions, as expected. There are 21 quadratic hexahedron elements along the crack front here; hence 43 nodes for the same curved length. The optimum mesh size was found out by increasing the number of elements on the crack front. Using 21 elements with FRAC3D program was seen to be sufficient to ensure accuracy. The aforementioned programs that simulate crack propagation phenomenon require hundreds of nodes used along the crack front to determine the correct solution. That is a quantitative comparison between other software and FRAC3D in this aspect.

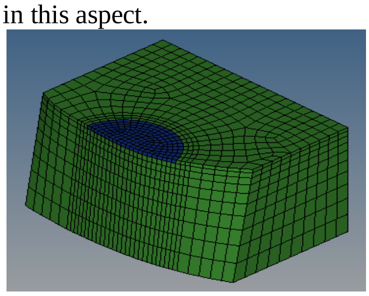


Figure 8. Mesh for bottom half of the local model with initial crack (Crack surface is shown with blue elements).

The initial orientation of the crack surface on the trailing edge of the blade was specified depending on the direction of maximum principal stress for that crack length. Therefore, the stress intensity factor solutions along the crack front demonstrated in Figure 9 shows non-zero values only for mode I results, i.e.,

K_I. That's an anticipated outcome due to assumed direction of the initial crack.

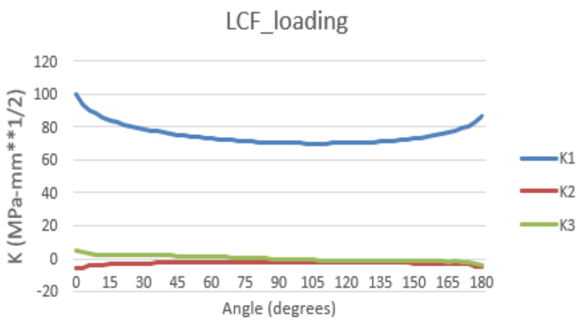


Figure 9. Stress intensity factor solutions for the initial crack on the trailing edge.

The width of the blade model is 85 mm. In order to demonstrate the effectiveness of the current methodology development, the crack was propagated ~6 mm, and the propagation path can be seen as the yellow line in Figure 10. This propagated length corresponds to the 20th step of the simulation. The local model for this step is shown in Figure 11, and the positioning of the same model within the global model can be seen in Figure 12. By using this procedure, the crack can further be extended to determine the final location of the propagated flaw. It was seen that the current development is effective considering accuracy and computational times to investigate these types of problems with high complexity in terms of loading and geometry. Since the local model with hexahedron elements is regenerated automatically with Hypermesh program at each step during propagation, the main challenge for these problems is being handled with less computational cost.

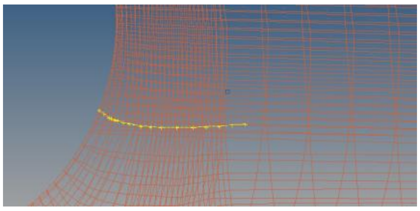


Figure 10. Crack propagation path after 20 steps.

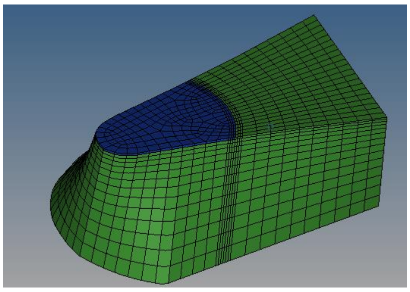


Figure 11. Local FE model for the propagated crack, at 20th step, blue elements indicating the crack surface.

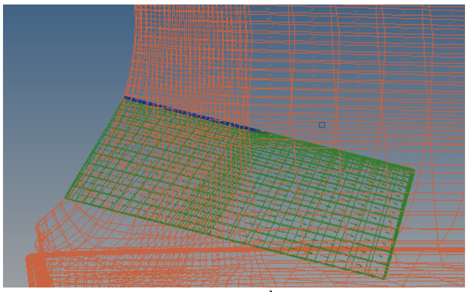


Figure 12. Local model at 20th step positioned within the global model

The simulation for the crack growth phenomenon under the influence of mixed-mode LCF loading has a few crucial points to discuss. These points may be decisive factors for the accuracy of the analysis results. The process starts with the random selection of a suitable Δa_{max} parameter (Equation (6)). In other words, there is no direct criterion that determines this parameter for a specific problem. In the beginning steps of the analysis where the crack lengths are shorter, Δa_{max} parameters are selected as lower values. As the crack grows, the value of this parameter can be increased for the following steps. This is a typical procedure for the analysis of crack propagation problems. Besides those, the author's experience on this subject shows that the FE models with many tetra elements are more sensitive to the selection of Δa_{max} parameter, when compared to the FRAC3D solution with hex elements. That means the value of this parameter (Δa_{max}) is increased slower, hence leading to more number of steps until reaching to the same crack length.

In FRAC3D solution of the current problem, the Δa_{max} parameter starts with 0.16 mm, and ends with 0.75 mm for the 20th step. If higher numbers are used for this parameter, it was seen that the new crack front nodes' coordinates turn out to be highly misaligned, leading to the difficulty for the creation of the crack geometry for the next propagation step.

The submodeling procedure described above carries an important risk if it is not performed in an accurate manner. One of the basic rules during the application of this procedure mentions the significance of the selection of submodel boundaries. This rule states that the boundaries should be sufficiently away from the critical regions within the submodel. In the case of a fracture problem, these critical regions are the locations that are close to the crack itself. Hence, the boundaries of the local model shown in Figure 7 should be selected by keeping a distance to the crack, in all three directions of the local coordinate system shown in Figure 4. Otherwise the results of the analysis can be misleading.

CONCLUDING REMARKS

A new computational procedure was presented to investigate the simulation of mixed-mode fatigue crack growth phenomenon for complex problems. A blade model subjected to LCF loading was of special interest in this study. This new methodology can also be used to analyze even more complicated problems, such as examples with the inclusion of HCF loading.

The procedure involves creation of the model with hexahedron elements, which is a challenging issue for complex geometries. The automation of remeshing process becomes necessary to accomplish this objective. Expectedly, models with hexahedron elements lead to more accurate results when compared to tetra element models. On the other hand, FRAC3D program has the technical capability to give accurate results with meshes relatively coarser. Current features of FRAC3D, such as the capabilities to analyze dynamic problems, structures with elastic-plastic material properties, anisotropic behavior, etc., will be considered along with this new crack growth procedure. That will also give us the chance to simulate propagation problems, which have never been tried before.

Being a special computational tool, this new methodology possesses a few significant features for the sake of analysis accuracy. These features are mostly related to the specification of maximum increment along the crack front, having the optimum size for the local model, and keeping the mesh density around the crack region at reasonable levels in terms of computational expense.

ACKNOWLEDGMENT

The author would like to express his sincere thanks to Prof. Nied, from ME&M Department of Lehigh University, US. His initiation, supervision and guidance throughout this study have been very significant to accomplish this task.

REFERENCES

[1] Dhont, G. (1998). Automatic 3-D mode I crack propagation calculations with finite elements, *Int. Jour. of Num. Met. in Eng.*, 41, 739-757.

[2] Carter, B.J., Wawrzynek P.A., Ingraffea, A.R. (2000). Automated 3-D crack growth simulation, *Int. Jour. of Num. Met. in Eng.*, 47, 229-253.

[3] Hou, J., Goldstraw, M., Maan, S., Knop, M. (2001). *An evaluation of 3-D crack growth using ZENCRACK*, DSTO-TR-1158, Defense Science and Technology Organization.

[4] Schollmann, M., Fan, M., Richard, H.A. (2003). Development of a new software for adaptive crack growth simulations in 3-D structures, *Eng. Frac. Mech.*, 70, 249-268.

[5] Sukumar, N., Chopp, D.L., Bechet, E., Moes, N. (2008). Three-dimensional non-planar crack growth by a coupled extended finite element and fast marching method, *Int. Jour. of Num. Met. in Eng.*, 76, 727-748.

[6] Dundar, H., Ayhan, A.O. (2016). Non-planar crack growth analyses of multiple cracks in thin-walled structures, *Int. Jour. of Fatigue*, 92, 596-604.

[7] Sih, G.C. (1990). Mechanics of fracture initiation and propagation, Kluwer Academic Publishers.

[8] Paris, P., Erdogan, F. (1963). A critical analysis of crack propagation laws, *Jour. of Basic Eng.*, 85, 528-533.

[9] *Nasgro Fracture Mechanics and Fatigue Crack Growth Analysis Software Reference Manual* (2016). Version 8.1, NASA Johnson Space Center and Southwest Research Institute, Texas, ABD.

[10] Erdogan, F., Sih, G.C. (1963). On the crack extension in plates under plane loading and transverse shear, *Jour. of Basic Eng*, 85, 519-525.

[11] Ayhan, A.O., Nied, H.F. (2002). Stress Intensity Factors for Three-dimensional Surface Cracks Using Enriched Finite Elements, *Int. Jour. for Num. Met. in Eng.*, 54, 899-921.

[12] Saribay, M., Nied, H.F. (2014). Dynamic stress intensity factors for suddenly loaded structures using enriched finite elements, *Theor. Appl. Fract. Mech.*, 70, 59-67.

[13] Ayhan, A.O. (2011). Simulation of three-dimensi onal fatigue crack propagation using enriched finite elements, *Computers and Structures*, 89, 801-812.

[14] Altair Hyperworks, Version 11.0, Copyright 1986-2019 Altair Engineering

[15] ANSYS Mechanical APDL, Version 15.0.7, Copyright 2014 SAS IP, Inc.

[16] https://ffden2.phys.uaf.edu/webproj/212_spring _2015/Timothy_Sherry/Tim_Sherry/Compressor.htm

[17] https://grabcad.com/library/compressor-blade-jet-engine-1