A Software Tool for the Fatigue Growth Analysis of Multiple 3D Cracks from Fastener Holes

Yixiu Shu[1], Yazhi Li[1], Zhenxing Fan[1]
[1]School of Aeronautics, Northwestern Polytechnical University, Xi'an, 710072
* Corresponding author: shuyixiu@126.com

Abstract A software tool is developed to carry on automatic fatigue crack propagation analysis for the multiple three-dimensional (3D) cracks initiated from fastener holes. It is a computer code package being capable of handling the cases of multiple 3D cracks with quadratic curve fronts. The object-oriented programming language is used to develop user interfaces for the input of the initial data and output of the simulation results. To achieve life-span numerical simulation for the multiple 3D crack growth, ANSYS Parametric Design Language is used to create finite element model, conduct the stress analysis and regenerate finite element mesh for new crack configurations. Stress intensity factors of the cracks are determined using the crack opening displacements and incorporated into crack propagation laws to estimate the crack extension. An algorithm for the fast integration of crack growth equations is adopted to predict the fatigue crack growth life. As examples, two scenarios of corner cracks initiated from countersunk fastener hole in 2324-T39 alloy sheets are analyzed throughout their fatigue growth lives. The simulation results correlate well with the experimental ones.

Keywords Fastener hole, multiple 3D cracks, fatigue crack growth, numerical simulation, software tool

1. Introduction

Widespread fatigue damage (WFD) is one of the major concerns for the fleet of aging airplanes and significantly decreases the residual strength and residual fatigue life of the aircraft structures. The multiple site damage (MSD) is a kind of WFD happened in similar places of a single structure element and often appears as the multiple cracks initiated from the fastener holes of the aircraft panel structures. The link-up of these cracks is one of the critical conditions and probably leads to the final structure failure. Although most of the available studies are referring to the two-dimensional cracks [4-8], cracks in the panel structures usually initiate as three dimensional (3D), especially the corner cracks from fastener holes of thick panels. A cracks initiated from fastener hole side can be spotted from panel surface only if it has grown out of the area covered by fastener head, washer or nut, or penetrated the panel thickness. The period of the crack growth from initiation to visible takes a large proportion of its total life. Therefore, it is significant to study the behavior of multiple 3D crack growth from very beginning to final failure and achieve an adequate estimation of the crack growth life.

This paper deals with the fatigue growth analysis of multiple 3D cracks initiated from an array of countersunk holes, the scenario that is even more complicated than the kind of cracks from the regular through holes. There are two essentials to conduct such analysis. The first is to attain the stress intensity factors (SIFs) along the front curves of each crack. Because of the varying and complexity of the crack geometry, there is no closed form solution or data available for the SIF. The solution of the SIFs must be determined numerically, mainly by the finite element analysis, such as the calculations of Shen[9], Fawaz[10] and Park[11]. For the simulation of the scenario throughout the whole crack growth history, it is essential to develop a robust technique to take on the task of finite element modeling for given crack pattern and do remeshing over and over again for the various crack sizes and shapes relative to structure geometry during the crack growth. The second requirement is to apply an approach being able to make fast integration of crack growth equation and achieve consistent growth of the multiple cracks with error control, so as to attain adequate crack growth life prediction efficiently. The applicable approaches include the trial and error method [15], variable optimization method [16] and load cycle segmentation method [11-13], etc., which have been used mainly for multiple 2D cracks. The reason of the more complexity of the present scenario over the multiple 2D crack issue is that the growth of every single 3D crack can be taken as that of several 2D cracks.

This paper describes the development of a computer tool based on commercial finite element software package, aiming to accomplish efficient and automatic simulation for the multiple 3D crack growth and life prediction. Design philosophy, analysis procedure and finite element modeling technique are described in detail. Examples are given to demonstrate the capability of the tool. Comparison is made between simulation results and the experimental ones.

2. Principal of multiple 3D crack growth analysis

2.1 Brief introduction of the tool

Although most of the present commercial finite element packages already have the function module to analyze fracture parameters, difficulty still remain in creating and updating finite element meshes for different crack patterns during crack growth, particularly for the situation of multiple site 3D cracks. So we decided to develop a tool to handle the analysis process of multiple 3D cracks over the whole fatigue growth life, without artificial intervention. The basic steps and module of performing such an analysis are shown in Fig. 1. The commercial software ANSYS are adopted as the computational engine, since the ANSYS Parametric Design Language (APDL) is easily edited and compiled to perform pre- and post- data processing, damage accumulation and result visualization during the analysis. The object-oriented language Visual Basic (VB) is used to develop user interfaces to facilitate operation and result representation.

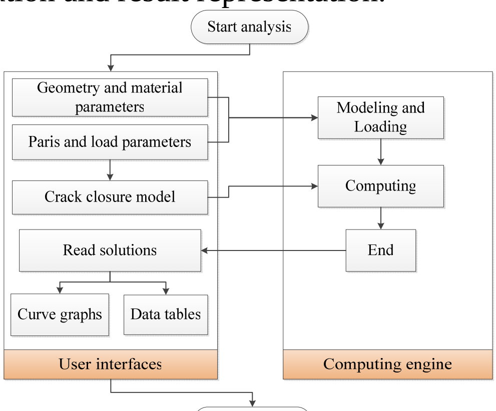


Figure 1. Flow chart of the tool

The tool has been tested under Windows XP and Windows 7 operation systems. It focuses on the sheet specimens with coplanar fastener holes (both countersunk and simple circular through ones) and multiple 3D cracks alongside them. The specimens are subject to constant amplitude loading and cracks may initiate from both sides of each fastener hole. Whole process of multiple crack growth can be simulated. Finite element modeling is parameterized, so as to adapt different geometries of structure and cracks. The parameters of geometry, material properties, loads and constraints are written to the file according to the APDL convention to parameter definition and passed to ANSYS. The analysis process is then running in background.

2.2 Philosophy of crack growth analysis

2.2.1 The fatigue crack growth model

3D cracks are supposed to propagate along the direction normal to their fronts. The Paris-form of crack growth rate equation is used [1]:

$$\frac{da}{dN} = C(\Delta K)^n \tag{1}$$

where ΔK is the stress intensity factor range.
Elber [2] discovered the phenomenon of fatigue crack closure and ΔK was replaced by its effective value

$$\Delta K_{eff} = K_{\max} - K_{open} = U \Delta K \tag{2}$$

where K_{open} is the value of stress intensity factor that causes crack opening during load cycling. U is crack closure function defined as the ratio of ΔK_{eff} to ΔK .

Elber carried out experiments on the Al-alloy 2024-T3 and found that for the stress ratio $R = -0.1$ to 0.7, U can be described as:

$$U = 0.5 + 0.4R \tag{3}$$

An improved function with a more realistic behavior for negative R-values was proposed by Schijve [3]:

$$U = 0.55 + 0.33R + 0.12R^2 \tag{4}$$

The crack growth life prediction program AFGROW [17, 18], developed by J. A. Harter uses a crack closure model in which a closure factor C_f is introduced.

$$K_{open} = C_f K_{\max} \quad ; \quad C_f = 1 - \left[\left(1 - C_{f0}\right)\left(1 + 0.6R\right)\left(1 - R\right)\right] \tag{5}$$

where C_{f0} is the value of C_f for $R = 0$. thus

$$\begin{aligned} \Delta K_{eff} &= K_{\max} - K_{open} = \frac{1 - C_f}{1 - R} \Delta K \quad \text{if } K_{open} \geq K_{\min} \\ \Delta K_{eff} &= K_{\max} - K_{\min} = \Delta K \quad \text{if } K_{open} < K_{\min} \end{aligned} \tag{6}$$

The Schijve model and the AFGROW model are implemented in our tool using APDL code.

2.2.2 Stress intensity determination

Stress intensity factors are determined by the finite element analysis of cracked structure as well as the crack tip opening displacement (CTOD) approach. As shown in Figure 8, for 2D crack case, K value at point P located on crack surface and close to crack tip can be calculated :

$$K_{\mathrm{IP}} = \frac{2G}{1+\kappa} \cdot \sqrt{\frac{2\pi}{r_P}} \cdot \left|v\left(r_P\right)\right| \quad \left(P = J, K\right) \tag{7}$$

where: $\kappa = \begin{cases} 3 - 4\upsilon & \text{plane strain} \\ \dfrac{3-\upsilon}{1+\upsilon} & \text{plane stress} \end{cases}$.

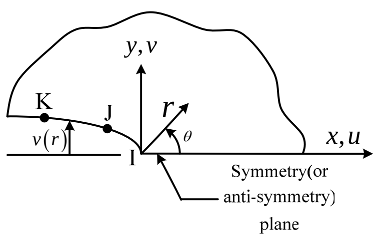


Figure 2. Normal section of a crack

Then the stress intensity factor at crack tip can be determined by displacement extrapolation techniques:

$$K_{\mathrm{I}} = \frac{c}{c-1} K_{\mathrm{I}J} - \frac{1}{c-1} K_{\mathrm{I}K} \tag{8}$$

where: $c = L_K / L_J$, L_K and L_J are distances from the point *K*/*J* to the crack front respectively.

2.2.3 Denotation of different crack configurations

For the present 3D cracks, SIF at any point of a crack front curve is determined by taking Figure 2 as the plane passing that point and normal to the tangent of the curve. The crack front of a corner crack is assumed as a quadratic curve, as is shown in Figure 3. The crack growth is simulated stepwise using remeshing technique combining with fatigue crack growth model. In each analysis step, three sampling points are chosen on every crack front, one in the middle of the curve and two others are close to both ends of the curve. The increments of crack growth on these points are assumed along the directions normal to the crack front curve. We are not choosing both ends of the curve as the sampling points, since the curve may not be perpendicular to the free boundaries on its ends and this will bring error to the stress intensity determination. The crack front curve at the end of present step is updated as a new quadratic curve according to the new positions of the sampling points.

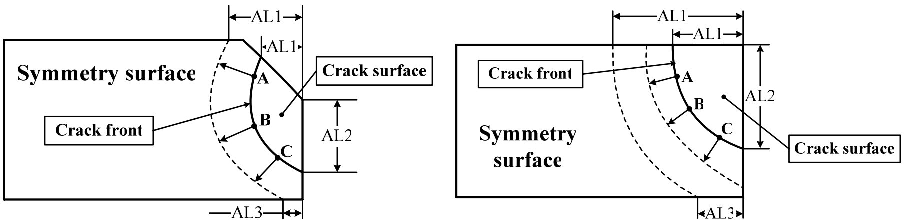


Figure 3. Crack shape and location of sampling points.

2.2.4 Criterion for the multiple crack growth

Fatigue cracks are growing cycle by cycle by their nature, but we cannot afford doing in this way in our simulation. It is possible to boost the calculation by splitting total number of load cycles into several steps. The number of cycles in each step is determined with error control. By this way the crack size increments are calculated stepwise. A criterion proposed by Zhang [11, 12] for multiple 2D crack case is adopted for the present multiple 3D cracks. All the sampling points on every crack front curve should meet the following criterion for crack size increment:

$$\Delta a_{ij} \le 2a_{ij} \left[\left(1 - et\right)^{-\frac{1}{2n}} - 1 \right] \tag{9}$$

where a_{ij} is the crack size at point *j* in computing step *i*, Δa_{ij} is the corresponding crack size increment, *n* is the exponent parameter in equation (1) and *et* is the error limit (normally as 0.01~0.1). Substituting equation (1) into equation (9), we obtain the upper bound of the number of load cycles:

$$\left(\Delta N\right)_{ij} = \frac{\Delta a_{ij}}{C \cdot \left(\Delta K_{ij,eff}\right)^{n}} \le \frac{2a_{ij}\left[\left(1 - et\right)^{-\frac{1}{2n}} - 1\right]}{C \cdot \left(\Delta K_{ij,eff}\right)^{n}} \tag{10}$$

2.2.5 Failure criterion

The specimen is considered failing if any one of the following conditions is met: all cracks have separated its ligament (crack front approaches plate edge or the opposite hole edge or the distance between two approaching crack fronts is less than a small amount, such as 0.3mm (crack linking up)); or the average level of net section tensile stress has reached the yield strength of the material.

3. Finite element modeling technique.

3.1 FE model of fatigue crack growth

As the example to demonstrate the modeling technique of our software, consider a rectangular plate containing a central countersunk hole with both side corner cracks. As shown in Fig. 4, half-length model is used because of the symmetry of the specimen. The solid model is divided into three parts by two parallel cross sectional planes. The part including the hole is part 1 and the one next is part 2. Part 1 is re-divided into two blocks so that each crack belongs to a single block. Crack front curves are assumed quadratic. Two narrow volumes with rectangular cross section are created along each crack front (Fig. 5). A narrow volume is the crack front body used to build the mesh locally to the crack front.

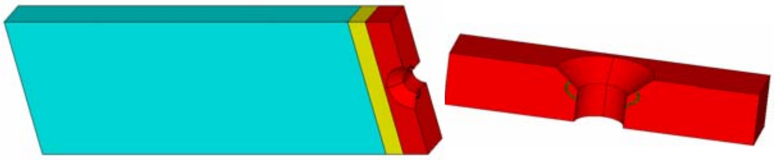


Figure 4. Model divisions

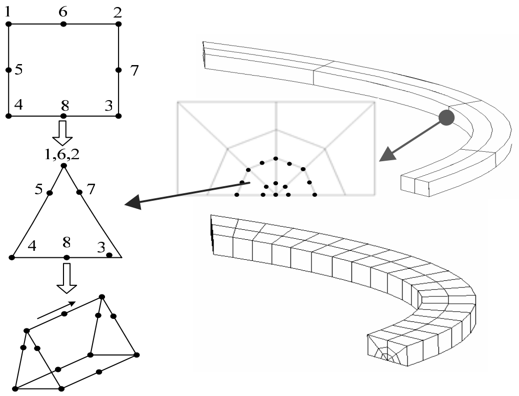


Figure 5. Finite element mesh for crack tip volumes

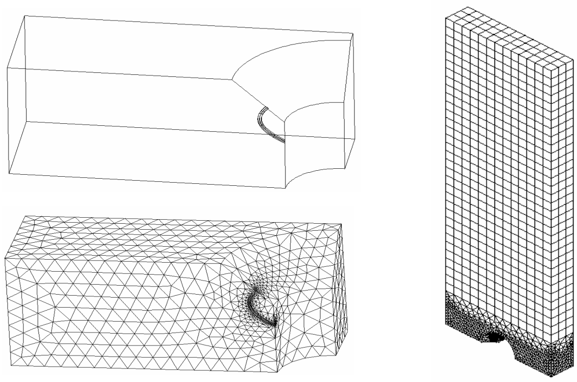


Figure 6. Finite element mesh

Cut each crack front body with three planes normal to the crack front curve, one in the middle and two others close to both ends of the body. We first mesh the cross sectional area created by the middle plane with eight-node isotropic 2D elements in which the four elements around crack tip are converted to the 8-node triangular elements by collapsing the three nodes on one of the four edges of each element to crack tip, and the mid nodes on the edges adjoining to crack tip have been moved toward crack tip to the quarter length position to produce crack tip singularity. We then sweep the area mesh both ways along the crack front to create the 3D mesh of the crack front body. The rest volume of part 1 is modeled by tetrahedron 3D elements. Part 3 is built with twenty nodes 3D elements. Part 2 is the transitional volume between part1 and part 3 and is modeled by tetrahedron 3D elements. Meshing details are shown in figure 5 and figure 6.

3.2 Considerations for the crack geometry change during remeshing

With the use of APDL language and the technique described in section 3.1, it is easy to accomplish remeshing for the crack configuration shown in Figure 3. Nevertheless, the remeshing will fail when one of the crack front curve ends is moving across a vertex of the cross sectional area, because of the element distortion. So, "transitional zones" are specified around those vertexes for different crack scenarios, as shown in Figure 7. The radius of a transit zone is about 5~6 times of the crack tip element size (fixed as 0.2mm in our tool). If one of the crack front ends has just entered the transit zone, user defined crack front will be instead.

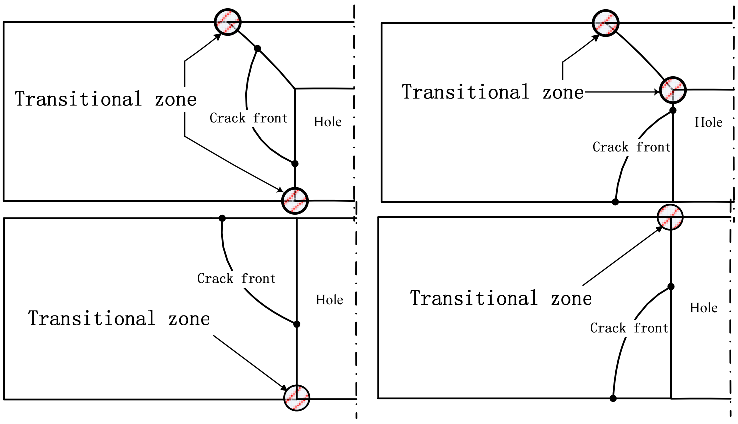


Figure 7. The transitional zones for different corner cracks from hole side

As example, corner cracks have initiated from a countersunk hole (Figure 8). There are two conditions for a crack front curve end moving across the vertex: before and after crossing. If a crack front curve end has moved in the transitional zone and has not cross the vertex, we bring the end back to the intersection point of the model boundary and the transitional zone. But the increment calculated in this step is stored and added to next step of calculation until the crack cross the vertex. In another condition that the end has passed the vertex but is still within the transitional zone, we move the end forward to the intersection point of the model boundary and the transitional zone. The two procedures make it possible to avoid element distortion and program crushing down. Simulation results shown in Figure 9 reveal that the procedures have only disturbed the form of a-n curve a little locally and not fundamentally affected the curve.

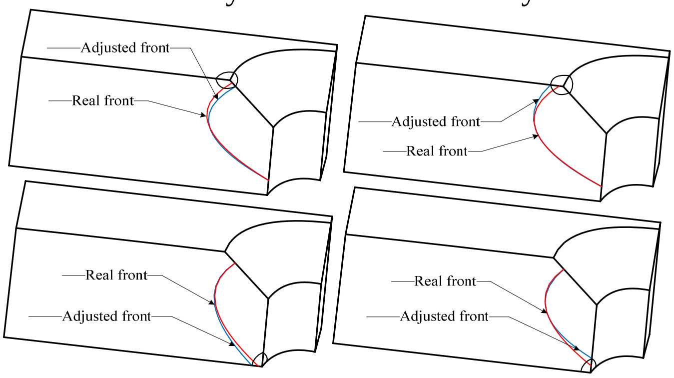


Figure 8. Crack adjustment when crossing a vertex.

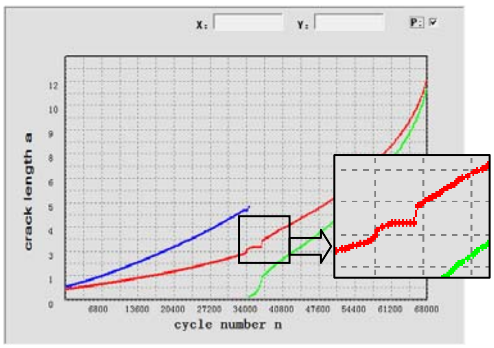


Figure 9. The effect of transitional zone treatment on a-n curves

4 User interfaces of the software.

4.1 Data preparation for the crack growth simulation.

Figure 10 shows the main interface of our software. There are two areas on the interface: the menu tree (area ①) and the operation area (area ②). Buttons for selecting models are listed on the menu tree on the left side of the interface and submenu belonging to each model can be unfolded by click on the corresponding button. The corresponding functions are then displayed on the operation area.

Parameters list on the operation area need to be input correctly. Explanations to the parameters are illstrated on the schematic diagrams at right hand side of the area.

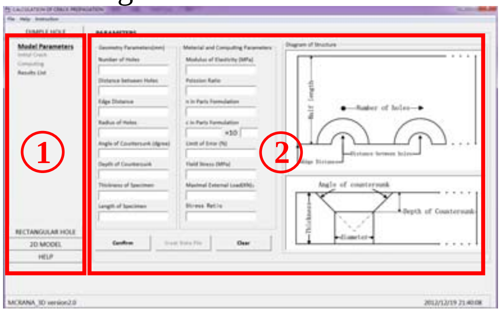


Figure 10. Main interface

Initial cracks are defined by inputing the parameterized sizes for each crack, as shown in Fig. 11. Two possible sites of crack initialization are indicated and each crack is quantified by three crack size parameters. For countersunk holes, crack located at countersink edge when the value of AL3(AR3) is zero and at the right angle edge when the value of AL1(AR1) is zero. Similarly, upper edge and bottom edge respectively for round through holes. All the three parameters for a crack are set to zero if it does not exist. Up to ten cracks are allowed in the software. A couple of crack growth equations are available. During execution, the real-time display of crack pattern can be observed on the computing window shown in Fig. 12.

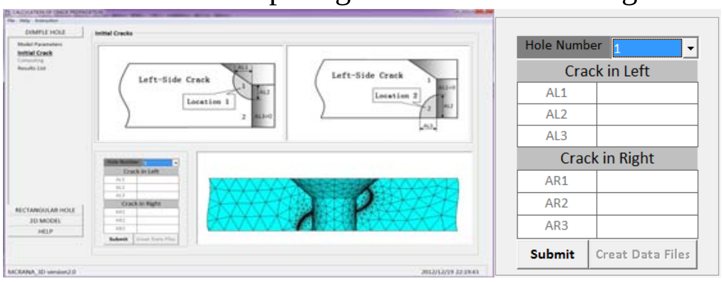


Figure 11. Interface for setting initial cracks.

Figure 12. Computing interface

4.2 Result Lists

Results are displayed in the form of data tables and curve graphs: data of parameterized crack sizes, stress intensity factors at sampling points, curves of crack sizes vs. load cycles and curves of stress intensity factors vs. crack sizes. All the data and curves can be saved as data files or image files. Figure 13 shows the examples of result display.

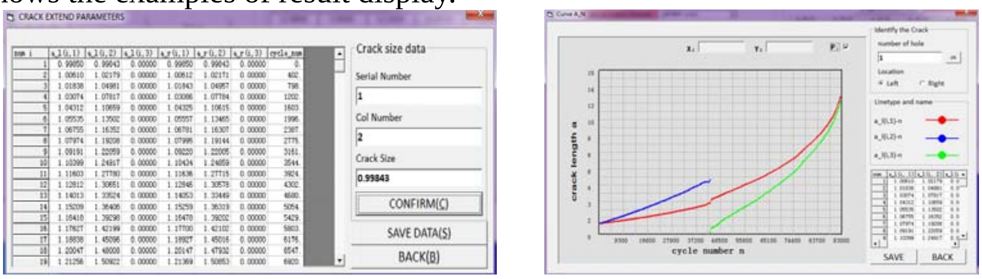


Figure 13. Two of the four interfaces of results representation

5. Simulation examples

5.1 Specimens.

Test has been carried out for the specimen with central countersunk hole, as shown in Fig. 14.

Material of the specimen is 2324-T39. The tensile strength is 8.6Mpa and the yield stress is 448MPa. The Paris law constants for 2324-T39 are $C = 2.74284\times10^{-7}$ and n=2.5269, with the units of ΔK in $\mathrm{MPa}\sqrt{\mathrm{mm}}$ and da/dN in mm/cycle.

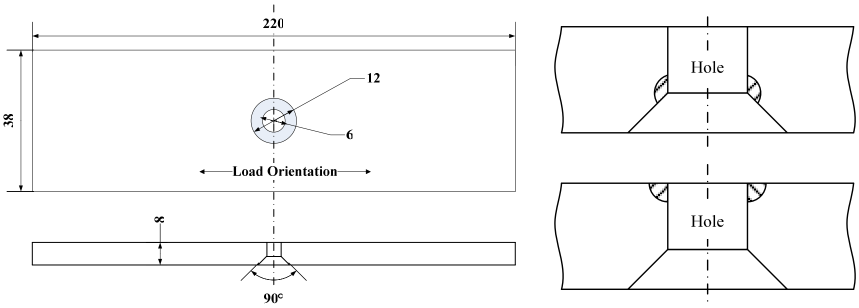


Figure 14. Specimen with a central hole and location of initial cracks.

There are two scenarios of initial cracks. Scenario 1 is double side sector-shape corner cracks from the edge of the countersunk and round through hole. Scenario 2 is the double side quarter disk corner cracks from the edge of the through-hole and back surface. The sizes of the initial cracks are listed in Table.1. The sizes of both side initial cracks are unequal for both scenarios.

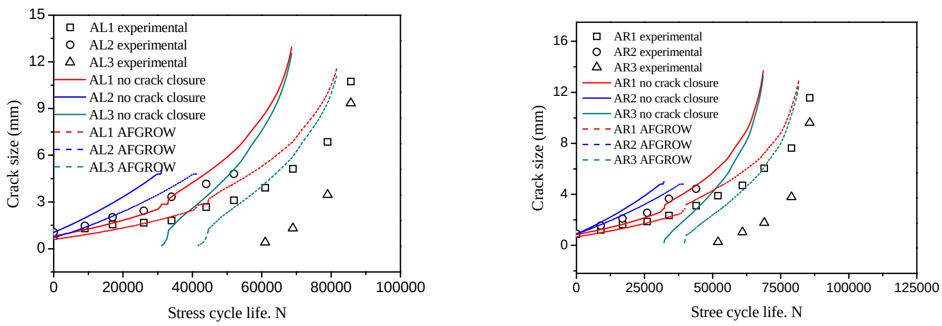
Table.1 Sizes of initial cracks

Crack sizes	AL1	AL2	AL3	AR1	AR2	AR3
Model1	0.809	1.05	0	0．85	0.90	0
Model2	0	0.75	0.567	0	0.66	0.614

Constant amplitude tensile cyclic load was applied. The peak load is 26KN and the stress ratio is 0.06.

5.2 Simulation results.

Two crack growth models were used in simulation: Paris law and AFGROW model with $C_{f0} = 0.2$. Analysis was made throughout the total life of crack propagation: corner cracks - partially through the thickness cracks - fully through cracks - final failure. Figure 15 gives the testing and predicted crack growth curves for scenario 1. Figure 16 shows the images of crack growth for the scenario during the simulation. Similar results for the scenario 2 are given in Figures 17 and 18. Table 2 gives the test results and predictions for the fatigue crack growth lives. The results indicate that the software tool works well and the simulation of the Paris law is quite conservative, while the AFGROW model gives more satisfactory estimation.



(a) (b)

Figure 15. Crack growth curves of scenario 1

(a). Left crack (b). Right crack

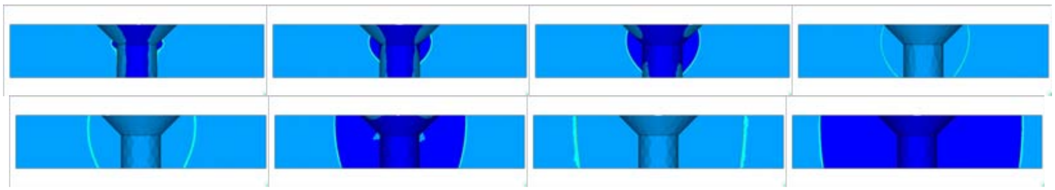
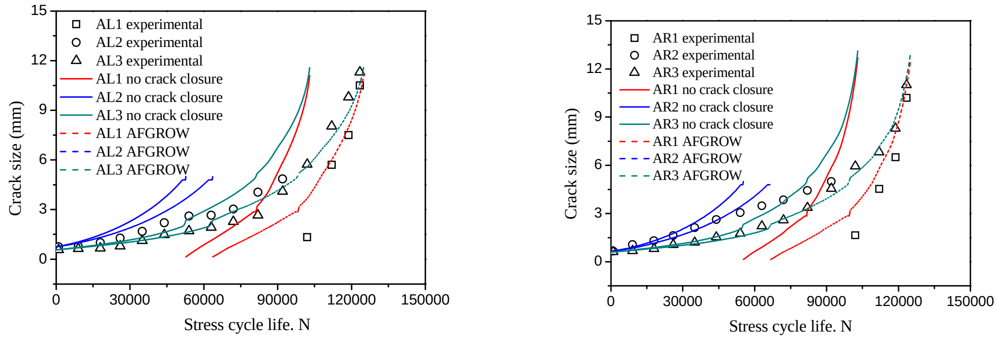


Figure 16. Images of crack growth of scenario 1



(b) (b)

Figure 17. Crack growth curves of scenario 2

(a). Left crack (b). Right crack

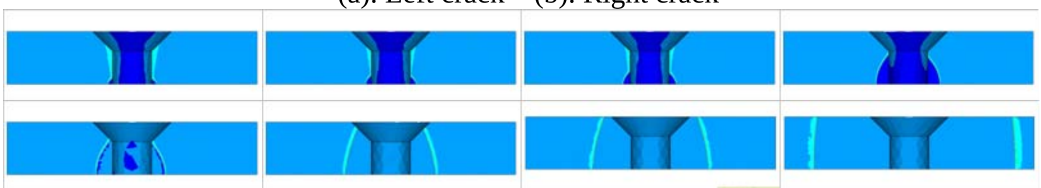


Figure 18. Images of crack growth of scenario 2

Table.2 Test and simulated results

	Crack location	Test life	No crack closure	AFGROW model
Scenario 1	countersunk side	85688	68587	81733
Scenario 2	rear side	123397	103027	125054

Further development of the tool involves extending the model type to the mechanically fastened joints through the consideration of fastener-hole contact.

6. Conclusions

A software tool has been developed using the ANSYS APDL and Visual Basic programming languages to simulate the multiple crack growth. This tool is capable of modeling metallic plate with coplanar countersunk holes or round through holes with different scenarios of multiple 3D cracks. 2D modeling function for through cracks is also available in our tool. Parameterized modeling is used to facilitate the pre-processing procedure. The Paris-form crack growth law and two crack closure models are included in the software. As the output of the execution, the lists of stress intensity factors and the curves of crack sizes vs. load cycles can be obtained. A useful function of our tool is the real-time visualization of the crack geometries during crack growth simulation. The results of example calculations indicate that the simulated crack growth behavior and fatigue growth lives agree well with the experimental results.

Acknowledgement

The research of this paper is a part of the project funded by Shanghai Aircraft Design and Research

Institute of COMAC.

References

[1] P. C. Paris, F. Erdogan. A critical analysis of crack propagation laws. J Basic Engng 1960, 85:528-34.
[2] Elber W. The significance of fatigue crack closure. Damage tolerance in aircraft structures. ASTM STP 1971, 486:230–42.
[3] Schijve J. Fatigue of structures and materials. Kluwer Academic Publishers, 2001.
[4] D. S. Dawicke and J. C. Newman, Jr. Analysis and prediction of multiple-site damage (MSD) fatigue crack growth, 1992, NASA-TP-3231.
[5] S. Pitt, R. Jones. Multiple-site and widespread fatigue damage in aging aircraft, Engineering Failure Analysis, 1997, 4(4): 237-257.
[6] R. M. V. Pidaparti, M. J. Palakal, Z.A. Rahman. Simulation of structural integrity predictions for panels with multiple site damage, Advances in Engineering Software, 2000, 31: 127-135.
[7] D. A. Cope, T. E. Lacy. Stress intensity determination in lap joints with mechanical fasteners, The 41th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics and Materials Conference and Exhibit, 2000.
[8] F. S. Lucas, J. P. M. Goncalves, F. M. F. Oliveira. Multiple-site damage in riveted lap-joints: experimental simulation and finite element prediction [J]. International Journal of Fatigue, 2000, 22: 319-338.
[9] H. J. Shen, W. L. Guo. 3D finite element analysis of SIF of the structure with dimple hole [J]. Acta Aeronautica et Astronautica Sinica, 2002, 23(2): 106-110 (In Chinese)
[10] S. A. Fawaz, B. Andersson. Accurate stress intensity factor solutions for corner cracks at a hole, Engineering Fracture Mechanics, 2004, 71: 1235-1254
[11] C. S. Wang. A study on prediction of MSD occurrence probability and residual strength/crack growth for typical structures. Dissertation of Beijing Beihang University, 2006: 69-71 (In Chinese)
[12] J. Y. Zhang, et al. Crack growth life estimating for MSD panel. Advanced Materials Research, 2008, 33(37): 175-180.
[13] X. Wang, M. Modarres, P. Hoffman. Analysis of crack interactions at adjacent holes and onset of multi-site fatigue damage in aging airframes, Int J Fract, 2009, 156: 155-163
[14] C. Y. Park, A.F. Grandt Jr., J. J. Suh. Stress intensity factors for surface cracks at countersunk holes, Engineering Fracture Mechanics, 2006, 73: 1878-1898
[15] M. F. Zhang. Durability and damage tolerance design manual of commercial aircraft structures: Damage tolerance design and analysis. Beijing: Aviation Industry Press, 2003. (In Chinese)
[16] Y. Q. Zhou, S. N. Wang. A variable optimization method-A new technique for multiple crack growth analysis in structure details. Acta Aeronautica et Astronautica Sinica. 1995, 16(2): 178-181 (In Chinese)
[17] AFGROW manual, Air Vehicles Directorate, US Air Force, 2006.
[18] J. A. Harter. Comparison of contemporary FCG life prediction tools. Int J Fatigue 1999; 21: 181-5.