Automatic prediction of the fatigue crack trajectory in plane finite element models

E A Matrosova, V V Sergienko, G P Egorov and D A Sadikov
Irkut Corporation, 68, Leningradsky prospect, Moscow, 125315, Russia
elenalekandrovna@mail.ru

Abstract. A module for automatic prediction of the fatigue crack trajectory in plane finite-element models in Ansys APDL software package was developed. Crack propagation is modeled by exclusion of elements from the active set in the direction, which is determined based on criterion of the maximum energy flux to the crack tip. The module determines the parameters of fracture mechanics (stress intensity factor and J-integral) at each step of crack propagation.

1. Description of automatic module algorithm

Shell-type finite element (FE) is used for modeling many thin-walled parts of an airframe that are subject to typical flight loads resulting in development of cracks with curved trajectory. This module is designed to automate and improve the accuracy of the damage tolerance calculations, as well as for the effective design of new structures with braking and controlled crack curvature, which will result in reduction in the cost for routine structural inspections and ensure compliance with regulatory requirements for damage tolerance. Currently, the module can be used to calculate models only from plane elements, which is the basis for adapting the module to solving spatial problems.

The initial data for the operation of the module are plane FE model and a vector defining the length and initial direction of the crack. At each crack propagation step, vector of crack front increment is formed and respective elements to be excluded from FE model are calculated by means of special software script. Then the next static calculation of the changed FE model is carried out.

At the next stage, several integration contours are created in the coordinate system of the crack tip. Value of energy flux to the crack tip (J-integral) is calculated for each integration contour. In order to increase the stability of the calculation, obtained numerical values of the J-integral on each contour are analyzed according to the specified procedure: individual extreme values are excluded from consideration and remaining values are averaged.

Each integration curve at the next crack propagation stage is checked for its extension beyond FE model (figure 1(b)). If most of the contours (more than 2) lie inside the FE model, the average value of the J-integral, stress intensity factor (SIF) and crack curvature angle at current step are calculated using the criterion of maximum flow to the crack tip (3).
Figure 1. (a) Elements modeling a crack, (b) contours that extend beyond the body.

If calculated crack trajectory deviation angle at the next step is greater than the specified allowable maximum, this step is recalculated with a reduced value of the crack increment.

Thus, data array containing information on the values of the parameters of fracture mechanics, the accumulated length of the crack, the angles of its curvature at each step, as well as the coordinates of the points of the trajectory formed by the segments of the crack, is generated in the course of module operation.

2. Determination of parameters of fracture mechanics
The stress-strain state of body is considered to be known from the solution of the corresponding elastic problem with a crack. The J-integral is the difference in the energies of the body before and after a small increment of the crack growth, which is spent on the formation of the corresponding increment of the crack area, and is determined according to [1] as follows:

\[
J = \int_\Omega \left( W_n ds - \sigma_0 n_j \frac{du_j}{dx} ds \right),
\]

where \( \Omega \) is the integration contour, \( W \) is the specific potential strain energy at contour point, \( n_j \) is the normal to the contour, \( \sigma_0, u_j \), are the stress and displacement components at the contour point. In the case of two-dimensional cracks, the energy flux vector can be decomposed into projections along the \( x \) and \( y \) axes of the local coordinate system associated with the crack tip, as shown in figure 2.

Figure 2. Local coordinate system that relates to crack tip.
According to [1] and [2], the projections of the energy flux vector to the crack tip can be determined by the following formulas:

\[
J_x = \sum_{i=1}^{N} W^{(i)} (y^{(i)} - y^{(i)}) - \sum_{i=1}^{N} \left( T_x^{(i)} \Delta U X_{x}^{(i)} + T_y^{(i)} \Delta U X_{y}^{(i)} \right)(S^{(i+1)} - S^{(i)})
\]

\[
J_y = \sum_{i=1}^{N} W^{(i)} (x^{(i)} - x^{(i)}) - \sum_{i=1}^{N} \left( T_x^{(i)} \Delta U Y_{x}^{(i)} + T_y^{(i)} \Delta U Y_{y}^{(i)} \right)(S^{(i+1)} - S^{(i)})
\]

where \( i \) is the number of points on the integration contour, \( N \) is the number of points on the integration contour, \( y^{(i)} \) and \( x^{(i)} \) are the coordinates of the point in LCS, \( T_x^{(i)} = \sigma_x^{(i)} n_x^{(i)} + \tau_x^{(i)} n_y^{(i)} \) is the projection of the stress vector on x axis, \( T_y^{(i)} = \tau_y^{(i)} n_x^{(i)} + \sigma_y^{(i)} n_y^{(i)} \) is the projection of the stress vector on the y axis, \( \Delta U X_{x}^{(i)} \) and \( \Delta U Y_{x}^{(i)} \) are the change in movement when the integration contour is shifted from \(-\Delta x / 2 \) to \( \Delta x / 2 \), \( \Delta U Y_{y}^{(i)} \) and \( \Delta U Y_{y}^{(i)} \) is the change in movement when the integration contour is shifted from \(-\Delta y / 2 \) to \( \Delta y / 2 \), \( S^{(i)} \) is coordinate along the length of the integration circuit from.

In accordance with the selected criterion, the crack will develop in the direction of the maximum energy flux to its pick, which allows determining curvature angle at each step of the increment of the crack length as the arctangent of the ratio of projections of total energy flux vector:

\[
\theta = arctg \left( \frac{J_y}{J_x} \right)
\]

SIF is the main engineering characteristic of fracture mechanics, since this parameter is used for determination of such material characteristics as development rate of the fatigue crack, crack resistance parameters, and others. According to [1], in the case of a plane stress state, SIF can be determined from the following relationships:

\[
K_I = \frac{\sqrt{E}}{2} \left( \sqrt{J_x - J_y} + \sqrt{J_x + J_y} \right)
\]

\[
K_{II} = \frac{\sqrt{E}}{2} \left( \sqrt{J_x - J_y} - \sqrt{J_x + J_y} \right)
\]

3. The results of the module operation

The module was verified on standard problems, the analytical solutions [3] of which are known, as well as the experiment described in [4]. The error in SIF circulation using the developed module with respect to analytical solutions does not exceed 3%.

Figure 3(a) shows the experimental sample from [4] while crack propagation path is indicated by the orange line. The dimensions of the sample are shown in figure 3, the magnitude of the applied force \( P = 1.33 \) kN, Young's modulus \( E = 71 \) 700 MPa, Poisson's ratio \( \mu = 0.3 \), initial crack is 18 mm.

Figures 3(b) – 3(c) also show the results of the numerical calculation of the model of the sample, given in [4] and obtained using the developed module. As can be seen from the figure 3, both calculated trajectories are very close to the experimental one, and their local differences are explained by different approaches used in the calculation.
In the course of module development, we have studied its sensitivity to the quality of FE mesh. The study showed that the use of unstructured FE mesh does not affect the results and accuracy of the calculation. This is a particular advantage of the module, since complex FE models frequently used for static calculations have an unstructured FE mesh, and its transformation into structured one in the area of assumed crack propagation requires additional time spending (which may be considerable).

![Image](image.png)

**Figure 3.** (a) Experimental sample, (b) calculation result given in [4], (c) the calculation result using developed module.

To validate the solution, a calculation was performed using the CINT procedure built into ANSYS APDL. That procedure allows to determine the parameters of fracture mechanics, but it requires the presence of model geometry and rebuilding FE mesh at each crack growth step. Graphs demonstrating a satisfactory coincidence of the trajectories and SIF at each step are shown in figure 4.

![Graph](graph.png)

**Crack trajectory**

![Graph](graph2.png)

(a)
4. Conclusion
The results obtained using the developed module and using the procedure built into ANSYS APDL are in good agreement. The possibility for using the developed module to automate the solution in a 2D FE statement of problems for determining the trajectory of development of fatigue cracks and the corresponding parameters of fracture mechanics at each crack propagation step was confirmed. Unlike the ANSYS APDL built-in procedure, the module does not require binding of the FE mesh to the body geometry. Rebuilding the FE mesh at each step of the crack propagation is not required as well. Thereby the module allows us to carry out calculations on already developed FE models and automate the calculation process even in the case of complex models with large number of elements.

References
[1] Morozov E M and Nikishkov G P 2008 Finite Element Method in Fracture Mechanics 2nd ed (Moscow: Publishing house LCI) p 256
[2] Morozov E M, Muizemnek A Y and Shadsky A S 2018 ANSYS in Hands of Engineer: Fracture Mechanics (Moscow: LENAND) p 456
[3] Ito Y, Murakami Y, Khasebe N, Yuuki R, Toya M, Togo K, Miyata Kh, Miyadzaki N and Aoki S 1990 Handbook of Stress Intensity Factors vol 1, ed Murakami Y (Moscow: Mir) p 448
[4] Damkilde L and Lambersen S H 2015 Numerical Analysis of Crack Propagation and Lifetime Estimation Master Thesis (Esbjerg: Aalborg University Esbjerg) p 84