7 PRACTICAL APPLICATIONS OF THE FINITE ELEMENT METHOD FOR LEFM

The previous sections of this chapter presented a number of example problems illustrating particular aspects of the finite element method in LEFM. In this Section, many of these aspects are integrated in a series of three practical problems from common engineering practice. The first problem employs the 2D, plane stress finite element method to compute accurate stress intensity versus crack length histories for crack growth in a lug. This would be a necessary computation for predicting fatigue crack growth rate, crack trajectories, and crack instability. The second problem uses a shell finite element model to simulate curvilinear fatigue crack growth in a pressurized aircraft fuselage section. Again, such a simulation would be useful for making damage tolerance and residual strength predictions. The final problem involves 3D solid finite element analysis of a spiral bevel, power transmission gear. Here the issue is accurate prediction of fatigue crack shape: a fractured tooth is a less serious event than a fractured gear.

7.1 Computing Stress Intensity Factor Histories in a Lug

The objective of this example is to illustrate the process of using a finite element program to compute stress intensity factor histories accurately, so that they can be used for fatigue crack growth rate (FCGR) and life predictions. The example is a relatively common structural detail, modeled as a plane stress problem, as shown in Figure 78. This figure shows a simple lug under non-symmetric loading. A contact-fit pin is inserted in the hole, and the pin load, P, is distributed to the hole by way of an elastic contact analysis. A crack is initiated from a location of high tensile stress concentration along the lug bore, and then allowed to propagate in mixed mode. FRANC2D (Cornell Fracture Group, 2002) is used as a finite element solver.

Figure 79 shows the initial finite element model for problem. The lug and its frictionless, contact-fit pin are both steel and are modeled with quadratic-order triangular and quadrilateral elements. The elastic contact problem between the pin and the lug is solved. Six-noded, zero-thickness interface elements with non-linear constitutive capability are inserted around the contact surface. The constitutive model used to represent the normal contact conditions is shown in Figure 80. No tension is allowed across the contact, and a high compressive normal stiffness is assigned to minimize intrusion of the pin into the lug

Figure 81 presents the uncracked, deformed shape and shows that separation of the pin from the lug has occurred from about the 8 o'clock to the 2 o'clock positions, while non-overlapping contact occurs along the rest of the contact. Figure 82 shows contours of major principal stress for the initial, uncracked configuration. This solution involved about 9200 DOF, and, with an error tolerance of 0.0005 on both equilibrium and displacement change between time steps, required about 5200 time steps and 2.5 minutes on a PC running Windows 2000 on a 1GHz Pentium III processor. This figure indicates two locations of high stress concentration around the lug bore, as expected. A slightly higher concentration occurs in the lower left quadrant, at about the 8 o'clock position, and a short crack (1.7 mm, 0.067 in.) is initiated at the location of highest circumferential tensile stress near this point. Growth of this crack is then simulated by:

1. Computing K_I and K_{II} using the equivalent domain formulation of the elastic J-Integral, see Section 3.5, above;
2. Calculating the direction of crack growth using the maximum circumferential tensile stress theory, see Section 4.1, above:
3. Extending the crack by 1.9 mm (0.075 in.) in this direction;
4. Resolving the finite element problem, including the non-linear contact between lug and pin;
5. Repeating steps 1-4 until the crack has extended about 64mm (2.5 in.) at which point fatigue life or residual strength limits are likely to have been reached.

Experience has shown that the treatment shown in this example, a rosette of eight six-noded triangular elements immediately surrounding the tip, quarter-point versions of these elements, and element size ranging from a few to as much as 25 percent of crack length, will produce very accurate values of stress intensity factors.

Figure 83 shows the corresponding amplified displaced shape, while Figure 84 shows the resulting stress intensity factor histories. Figure 83 shows that the crack trajectory is not quite radial, and is responding to the asymmetrical loading and geometry. The trajectory is dictated here by the maximum circumferential tensile stress theory that requires that K_{II} remain zero along the crack path. With a finite element model that discretizes the trajectory into finite, straight segments, there will always be residual, non-zero values of K_{II} computed at each crack tip location. If the segments are short enough, these residuals should be small compared to the K_I values. Figure 85 shows that the values of K_{II} are oscillating around zero, and are indeed small, in this case never reaching more than 3.5% of K_{I}. The highest values usually occur early in the trajectory while the finite element model is adjusting to the stress field that is evolving as a result of crack growth, as shown in Figure 85. A key practical issue suggested here is the length of crack growth increment. This length should be sufficiently short to accurately discretize a curvilinear trajectory and provide enough data points for the accurate integration required for FCGR calculations, while not being so short that excessive computation times accrue. Here 32 increments were used. The number of DOF's grew to nearly 14,000 at the last increment, and a total of about 2 hours of computing time was required.

A finite element code with fracture mechanics features can be thought of as a general stress intensity factor calculator. As such it can be used to attack practically interesting variants of problems. For example, it is possible that two fatigue cracks might initiate in this lug problem, one from each of the locations of initially high stress concentration. This possibility is also simulated here, under the assumptions that the cracks initiate simultaneously, and that they have equal rates of growth. The resulting trajectories under these simplifying assumptions are shown in Figure 86. The corresponding mode I stress intensity factor histories are given in Figure 87. This figure shows that, even if initiation were simultaneous, rate of growth would not be equal; the left crack would have higher growth rate. However, even under these simplifying assumptions, Figure 87 also shows that the growth rate of the left crack would be higher than it would be if it were the only crack to occur.

7.2 Predicting Fatigue Crack Life and Trajectory in an Aircraft Fuselage Section

Simulations of curvilinear crack growth in a generic narrow body fuselage panel were performed in Potyondy *et al.* (1995) and Chen *et al.* (1997). The predicted crack trajectories were compared with the measured values from a full-scale pressurization test. This problem demonstrates the applicability of the stress intensity factor calculation techniques for shell structures, Section 3, and the direction criteria developed in Section 4 for predicting curvilinear crack growth in fuselage structures.

7.2.1 Description of experiment

A narrow body fuselage panel with tear straps, stringers, stringer clips, and frames was tested by the Boeing Commercial Airplane Group. Skins and tear straps were made of 0.036 inch (0.91 mm) thick, 2024-T3 clad aluminum alloy. Stringers, frames, and stringer clips were made of 7075-T6 clad aluminum alloy. The tear straps were hotbonded to the skins at midbay and at each frame station. The structural features of the test panel are shown in Figure 88. More information about panel dimensions can be found in Gruber *at al.* (1996).

The panel had a 5.0 inch (127 mm) initial saw cut in the T-L orientation centered on the midbay tear strap and just above the stringer tear strap. The saw cut went completely through both the skin and midbay tear strap. The panel was inserted into a test fixture with a radius of curvature of 74 inches (1880 mm) to match narrow body airplanes. A cyclic pressure of 7.8 psi (53.63 kPa) was applied to propagate the crack.

During the test, the positions of the crack tips were recorded. The detailed test data can be found in Potyondy (1993).

7.2.2 Numerical model

In this study, a 4-stringer-bay wide and 2-frame-bay long panel was analyzed. The entire curvilinear crack growth simulation consists of more than 20 inches (508 mm) of crack extension. All structural components including skins, stringers, and frames were modeled by quadrilateral shell elements using the STAGS (Rankin *et al.,* 1997) and FRANC3D (Cornell Fracture Group, 2002) codes. Each node of the shell element has six degrees of freedom. A typical finite element mesh used in the simulation is shown in Figure 89. Geometrically nonlinear analyses were performed. For this example, only internal pressure on the skin of the shell model was applied to the structure. Thus, a simple numerical model using symmetric boundary conditions imposed on all the boundary edges was used to simulate a cylinder-like fuselage structure. Uniform axial expansion was allowed at one longitudinal end. On this boundary edge, an axial force equal to (PR/2)L was assigned where P is the applied pressure, R is the radius of the panel, and L is the arc-length of the edge.

7.2.3 Fracture parameter evaluation

Deformation and stress fields near the crack tip were used to compute fracture parameters for the crack growth simulations. The modified crack closure integral method, Section 3.8, was used to compute the membrane and bending stress intensity factors, K_{I}, K_{II}, k_1, k_2. Crack growth directional criteria for isotropic and orthotropic media from Section 4 were used to predict the propagation angle in this thin shell structure. The equivalent domain integral method for T -stress developed in Section 4.6 is only valid for two-dimensional problems. The derivation of its counterpart for shell structures subjected to large displacements and rotations is not yet available. Instead, a simple displacement correlation method was used to evaluate the T -stress (Sutton *et al.*, 1997).

7.2.4 Results of crack growth simulation

The effect of T -stress and r_c on crack trajectory prediction was studied first. Figure 90 plots the predicted crack trajectories with $r_c = 0$ and $r_c = 0.09$ inch (2.29 mm) as well as the experimental measurements. Figure 91 shows the computed deformed shapes during curvilinear crack growth. Bulging caused by the applied pressure is observed. Moreover, severe flapping is predicted as the crack turns. Figure 92 shows the computed stress intensity factors and T -stress versus the half crack extension at the right crack tip. Predicted results suggest:

1. The T -stress has a very mild influence on the early crack trajectory prediction because of its small magnitude. However, as the crack approaches the tear strap, T

-stress increases and plays an important role in the crack turning prediction. For the case with r_c = 0.09 inch (2.29 mm), a sharp turning caused by T -stress is predicted as the crack approaches the tear strap.

2. The computed fracture parameters for r_c = 0 and r_c = 0.09 inch (2.29 mm) are comparable at the early stage of curvilinear crack growth. However, sharp turning as the crack approaches the tear strap alters the deformation and stress fields. This drastically changes the computed values of fracture parameters.
3. Predicted crack paths from both numerical simulations at the right and left crack tips are almost symmetric about the midbay, but the measured crack paths are not. This observation gives a preliminary indication of the experimental scatter that might occur in the panel test.

The predicted crack growth trajectories depicted in Figure 90 are comparable to the experimental measurements, but with some discrepancy. The disagreement during early stages of crack growth might be related to the fracture toughness orthotropy of the fuselage skins. Therefore, additional analyses, using the orthotropic directional criterion were also performed. From coupon test results, the fracture toughness for this material and thickness was about 100 ksi √inch (109.8 MPa √m) in the L direction and 105--120 ksi √inch (115.3—131.8 MPa √m) in the T direction (Potyondy, 1993). Thus, the fracture toughness was assumed to be 10% higher in the T than in the L direction. The predicted crack trajectories with r_c = 0.09 inch (2.29 mm) are compared with those from the isotropic prediction and experimental measurements in Figure 93. During early stages of crack growth, the predicted trajectories for the orthotropic case agree better with the experimental measurements than the isotropic case. Crack growth simulation with fracture orthotropy also predicts crack turning as the crack approaches the tear strap. However, when the crack grows further into the tear strap region, the inclusion of fracture orthotropy adversely alters the crack path prediction and does not predict the flapping observed in the panel test. Several possible reasons for this are discussed in Chen *et al.* (2002).

7.3 Predicting Evolution of 3D Crack Shape in a Spiral Bevel Gear

Predicting evolving crack shapes is important in determining the failure mode of a gear. Cracks propagating through the rim may result in catastrophic failure, whereas the gear may remain intact if one tooth fails and this may allow for early detection of failure. Tooth-bending fatigue failure in spiral bevel gears was investigated using the boundary element method in Spievak *et al.* (2001). That effort was significant in developing a method for predicting three-dimensional, non-proportional, fatigue crack growth incorporating moving loads. In this example, the problem studied by Spievak *et al.* is approached with the FEM, using many of the techniques described in previous sections of this chapter.

For accurate mixed-mode SIF calculation with the FEM, the equivalent domain J-integral method described in Section 3.5 is used. Crack trajectory predictions are made using the maximum circumferential stress theory, Section 4.1. Fatigue crack growth rates are determined using a modified Paris model accounting for crack closure (Spievak *et al.*, 2001). Moving loads along the face of a tooth are taken into account by discretizing the loading into a number of steps that create a non-proportional load history on a crack. Solution time for each load step is substantially decreased by employing parallel FEM analysis.

The FRANC3D (Cornell Fracture Group, 2002) simulation system is used for this example. The method used to simulation crack growth in a gear is composed of the following steps:

1. Create initial geometry model of the spiral bevel pinion.
2. Specify boundary conditions and material properties on the geometry model.
3. Initiate a crack by locally modifying the geometry model.
4. Create a surface mesh composed of triangular elements.
5. Create a 3D solid mesh of the model composed of tetrahedral.
6. Calculate the magnitude and location of the nodal contact loads for each load step on the loaded elements.
7. Using the mesh and geometry models, and load files for each step of loading, run analysis on a parallel PC-cluster. As a result of these analyses, SIF values at each discrete crack front point and displacement and stress values for each load step are calculated.
8. Calculate the amount of crack extension at each discrete crack front point as a result of all steps of non-proportional moving load.
9. Determine new crack front by piecewise least squares fit of the propagated points corresponding to each discrete crack front point.
10. Remesh surface locally and repeat steps 5 to 10.

7.3.1 Modeling of moving tooth loads

Contact in a spiral bevel gear occurs in three-dimensions following a path along the tooth surface, starting from the fillet of the toe to the top of the heel. In analyzing the spiral bevel gear, it is assumed that the contact between the gear and its pinion follows Hertz theory of elastic contact. Hertzian contact holds as long as the significant dimensions of the contact area are small compared to the dimensions of each body and to the relative radii of curvature of surfaces. Hertz theory assumes that surfaces are continuous and non-conforming, strains are small, each solid can be considered as an elastic half-space, and surfaces are frictionless. Under these assumptions, only normal pressure acts between two bodies due to Hertz contact producing normal displacements of the surfaces. Hertzian contact assumptions are relaxed in Ural *et al.* (2002), wherein actual contact analysis is performed between the gear and its pinion.

7.3.2 Parallel finite element analyses

A parallel PC-cluster was used for the FE analyses in the crack growth simulations, step 7, above. The hardware consists of 32 2-way, Pentium III processors at 733 MHz running in a Windows environment. Table 22 presents some typical timing results for this example problem.

7.3.3 Determining new crack front

Crack extension due to one non-proportional load cycle, step 8 above, is computed using the approach developed in Spievak *et al*. (2001). As previously mentioned, a modified Paris model that accounts for crack closure effects is used. A crack is assumed to advance when its maximum SIF is large enough to overcome closure and is larger than the maximum SIF of the previous load step. Thereafter, for every point along the crack front:

1. Calculate the angle of crack growth corresponding to each load step using maximum circumferential stress theory, Section 4.1.
2. Calculate the final coordinates of the crack front and trajectory angles by approximating the contributions from each load step by a straight line.
3. Determine number of cycles necessary to have a reasonable amount of crack advance compared to pinion's geometry.
4. Update geometry using the new crack front points.
5. Repeat for each crack extension step.

7.3.4 Results of crack growth simulation

Analyses were performed on a spiral bevel pinion for 39 crack growth steps, Figure 94. One crack growth step consists of 15 static finite element analyses in order to simulate moving loads on the pinion tooth. After the 39th crack growth step, the analysis was stopped because it was concluded that propagating the crack further would not provide additional insights.

Figures 95 through Figure 97 show mode I, II and III stress intensity factors for the initial crack configuration for the first eleven load steps. Load steps 1-4 correspond to double tooth contact, load steps 5-11 correspond to single tooth contact, and load steps

12-15 again correspond to double tooth contact. Since a crack is not assumed to advance when its maximum SIF is smaller than the maximum SIF of the previous load step, only the first 11 steps practically contribute to crack growth calculations. In these figures, crack front position corresponds to the points near the crack front at which SIF values are evaluated. In the initial step of crack growth simulations, the crack front was discretized into 93 points at which SIF values were calculated.

Figure 98 shows the crack trajectory predictions on the tooth surface and cross section of the tooth for several crack growth steps, including the initial and final configuration of the crack. Figure 99 is a close up view of the last step of crack growth from the toe end of the tooth. This figure also shows the elliptical contact area for load step 14. Crack trajectory comparisons in Figure 100 show that, on the tooth surface, crack trajectory predicted by FEM is close to the experimental result. However, FEM results show a steeper propagation angle at mid-crack than that observed. In contrast, on the toe end FEM results exhibit a less steep propagation angle than the experiment. These discrepancies can be due to a number of factors that are further investigated in Ural *et al.* (2002).