SIMULATIONS OF CRACK GROWTH IN PIEZOELECTRIC STRUCTURES WITH MODERN, AUTOMATIC AND EFFICIENT FINITE ELEMENT SOFTWARE

Ł. Jański, M. Kuna and M. Scherzer
Institute of Mechanics and Fluid Dynamics, TU Bergakademie Freiberg, Germany

1. Motivation

Sensors and actuators are nowadays standard components of many modern adaptive mechanical systems. The role of these components implies frequently the application of piezoelectric and ferroelectric materials by their construction. The implementation of these sensors and actuators into mechanical systems leads to common problems associated with mechanical loading e.g.: providing of a satisfactory strength, durability and fracture resistance. Electromechanical sensors and actuators, however, are loaded not only mechanically but also electrically. This means also the case when the external force has a purely mechanical character and their internal response is electromechanical. Such behaviour can be observed due to the electromechanical coupling property of piezoelectric and ferroelectric materials. To assure a satisfactory fracture resistance of sensors and actuators, the knowledge of electrical as well as mechanical fields in the vicinity of cracks is necessary. This information makes the evaluation of the cracks behaviour under electromechanical static or cyclic loads possible. Electrical and mechanical fields in the vicinity of cracks can be obtained with classical solution strategy of complex functions theory. This strategy, however, can be in general used only for infinite domains. On the other hand, real engineering tasks always refer to finite domains with special electromechanical boundary conditions. The finite element method is usually used to obtain electrical and mechanical fields in the vicinity of cracks for real problems. Various variants of this method have already been tested for stationary cracks in homogeneous piezoelectric structures [1]. There are still many open questions concerning fatigue crack growth under electromechanical alternating loads. The present work should give at least some answers to these questions. To reach this aim, a special finite element tool is developed for modelling of a crack growth in piezoelectric structures and simulations of the crack growth are realised. The structure of this tool, shortly described in the following part of this abstract, is crucial for the effectivity of the simulations.

2. FE-program structure

The developed finite element tool is composed of four modules. The piezoelectric boundary value problem is solved with the finite element method in an adaptive manner in the main module. The finite element discretisation of the piezoelectric boundary value problem leads to an indefinite formulation. The Bramble-Pasciak preconditioner [2] is used to avoid complications arising from this fact. The preconditioned linear system of equations is solved with the conjugate gradient method. Because high gradients of the stress and the dielectric displacement associated with a crack tip are expected in the solution, an automatic, adaptive algorithm for the finite element mesh density optimisation is implemented. Information associated with the edge hierarchy established in the adaptive process is used for the construction of a very efficient hierarchic preconditioner.

Fracture parameters, e.g. mechanical and electrical intensity factors, are calculated in the second module. The implementation necessity of the crack tip finite elements in each adaptive step is avoided with the application of the interaction integral technique [3]. The asymptotic solution [4] is chosen as the auxiliary field in the interaction integral technique and also utilised by the construction of the Irwin's matrix [5].

Fracture criteria are evaluated and the decision undertaken whether the crack propagates or not in the third module. If the decision is positive, the parameters such as the length and the orientation of an incremental crack advance are calculated. At this juncture, classical fracture criteria of the linear piezoelectric fracture mechanics e.g. based on the circumferential stress or the mechanical energy release rate can be used in the first approximation.

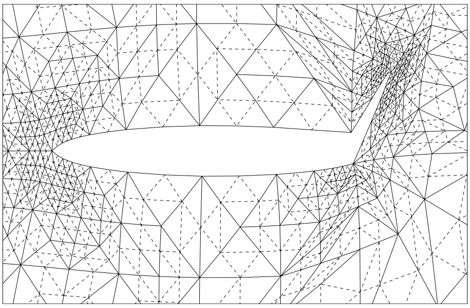

Figure 1. Finite element mesh around a crack with a kink.

The crack propagation is automatically realised in the finite element mesh in the fourth module. On one hand the length and the orientation of the incremental crack advance obtained in the former module are physically determined, on the other the crack propagation can be only realised along finite element edges which depend strictly on a meshing strategy. Consequently, new finite element nodes and edges must be constructed to let the crack grow. In Figure 1, an example of a finite element mesh around a crack with a kink which automatically propagated from the crack is presented.

3. Simulation results

Mechanical and electrical intensity factors are calculated for configurations, e.g. kinked crack, Griffith's crack, for which analytical solutions are know and appropriate factors compared to prove the efficiency of the first and the second module. Kinks are automatically generated for various orientations to test the fourth module. Crack propagation simulations are carried to check the third module and the whole finite element software. The results are analysed and discussed.

4. References

[1] M. Kuna (2006). Finite element analyses of cracks in piezoelectric structures: a survey, *Archive of Applied Mechanics*, **76**, 725-745.

[2] A. Meyer and T. Steidten (2001). Improvements and Experiments on the Bramble-Pasciak Type CG for Mixed Problems in Elasticity, *Preprint-Reihe des Chemnitzer Sonderforschungsbereiches 393,* **Nr.:01-13**.

[3] M. Stern and E.B. Becker (1976). A contour integral computation of mixed-mode stress intensity factors, *International Journal of Fracture*, **12**, 359-367.

[4] M. Scherzer and M. Kuna (2004). Combined analytical and numerical solution of 2D interface corner configurations between dissimilar piezoelectric materials, *International Journal of Fracture*, **127**, 61-99.

[5] L. Janski and P. Steinhorst (2007). Modellierung von Risswachstum in piezoelektrischen Materialien mittels moderner adaptiver FEM-Losungsstrategien, *DVM-Bericht*, **239**, 153-162.