Simulation of Failure Processes with Cohesive Elements

An important aspect of the design procedure is taking failure processes and failure states into account. Especially structural elements with composite materials and with glued or welded connections are worth considering by such analysis. Within the framework of a finite element analysis, the approach of cohesive elements is suitable for these investigations. The capabilities of the application are manifold as we have computation of mechanical properties in thin layers, simulation of delamination processes and of crack development. This class of elements is available in ANSYS since Release 10.0.

Taking the process zone of a growing crack tip into account, the development of the crack (Figure 1) under the formation of new free surfaces is described mathematically by Barenblatt [1]. First implementations of these theoretical foundations, including an application to the computation of void nucleation in a two phase material, can be found for instance in the work of Needleman [2] in the context of the discrete interface finite element method.

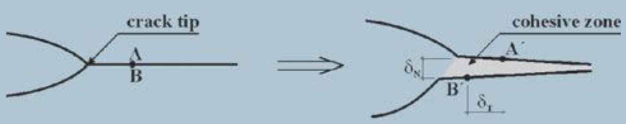

Figure 1: Crack Tip and Representation of Crack Propagation

Till this day, a large number of applications for this class of elements can be found mostly used in an experimental and scientific environment for quasi static and short time dynamic simulations in terms of delamination and failure. Recently, formulations of cohesive elements have been implemented into commercial finite element codes.

Element Formulation

Cohesive elements are introduced at the stage of the model generating process along the edges of volume elements at assumed possible crack paths. If the crack path location is goal of the computation, it is necessary to implement different possible crack paths in advance. The type of cohesive elements employed depends on the class of volume elements applied for the structural investigation. Node pairs linked with the interface element

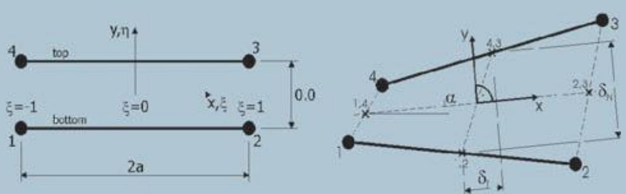

Figure 2: Cohesive-Element and Local Coordinate System (2D)

coincide ($\delta^i = 0$) in the reference configuration. Equivalently to contact computations, the spatial dimension of such elements is one order lower than the surrounding mesh. Figure 2 (left) shows a one-dimensional interface element in the reference configuration for two-dimensional simulations with a linear approximation of geometry and displacements.

The right part of Figure 2 depicts the construction of the local coordinate system and the determination of the deformations which describe the separation of the cohesive surfaces.

Material Formulation

The description of crack opening is defined by a constitutive formulation which relates separations δ^i and tractions T^i at the crack faces in normal and tangential direction according to the local coordinate system. Starting from a traction-free and undeformed condition up to a completely separated state, different formulations are available. Apart form the shape of the selected traction-separation-law, these functions are characterized by the maximum value of traction and separation as well as by the area under the function interpreted as fracture

$$\Phi = \frac{27}{4} T_0 \delta_0 \left[\frac{1}{2} \left(\frac{\delta^N}{\delta_0} \right)^2 \left(1 - \frac{4}{3} \left(\frac{\delta^N}{\delta_0} \right) + \frac{1}{2} \left(\frac{\delta^N}{\delta_0} \right)^2 \right) \cdots \right.$$
$$\left. \cdots + \frac{1}{2} \beta \left(\frac{\delta^T}{\delta_0} \right)^2 \left(1 - 2 \left(\frac{\delta^N}{\delta_0} \right) + \left(\frac{\delta^N}{\delta_0} \right)^2 \right) \right]$$

energy Γ_0. As an example, a polynomial based potential is introduced according to Needleman [2]. The material properties T_0 and δ_0 denote maximum tractions and separations under pure tension load. The value of β gives the relation between normal and tangential separations with similar amount of fracture energy Γ_0.

Tractions normal and tangential to the crack surface according to the local coordinate system are calculated as

$$T^N = \frac{\partial \Phi}{\partial \delta^N} \quad \text{and} \quad T^T = \frac{\partial \Phi}{\partial \delta^T}$$

which is partial derivative of the potential with respect to the accompanying traction. The principle results are graphically presented in Figure 3.

This formulation takes also the dependency between the separation and traction values of different delamination modes into account, which results in a coupled approach. The dissipated energy up to complete separation is inde-

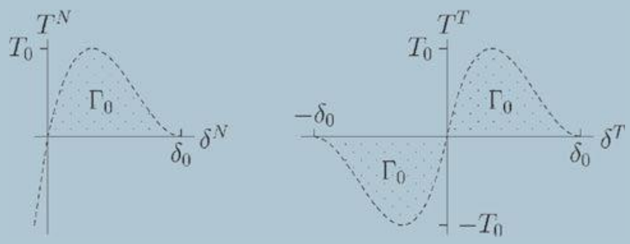

Figure 3: Traction-Separation-Law (Needleman [2])

pendent of the composition of the whole deformation.

Based on such elastic potentials, the formulation can be extended according to the investigated physical phenomena. Beside some damage enhancements for simulations under periodical loading conditions and rate dependent approaches (Corigliano et al. [3]), it is also possible to take viscoelastic properties (Geißler, Kaliske [4]) of the interface into account.

Examples

A first computation shows the simulation of a wedge splitting test of a concrete specimen. Material parameters and results of the experiments are published in Wille et al. [5]. Cohesive elements are placed along the observed crack path in the middle of the concrete body. The simulations are performed on a model considering symmetry conditions. Figure 4 shows the comparison of experimental and numerical results. In contrast to computations with a continuum softening degradation on element level, here, the results are independent of the used mesh.

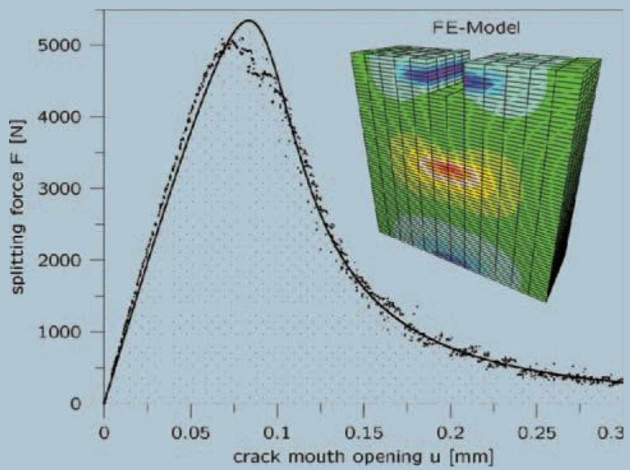

Figure 4: Numerical Wedge Splitting Test

A second example shows the simulation of a delamination process of fiber reinforced composite material. The model parameters (Figure 5) and the material properties are obtained from the work of Alfano and Crisfield [6].

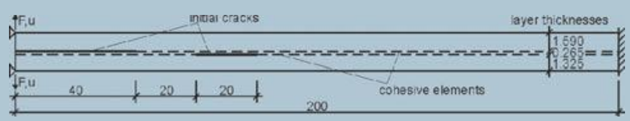

Figure 5: Dimensions of the Test Specimen for Delamination Simulation

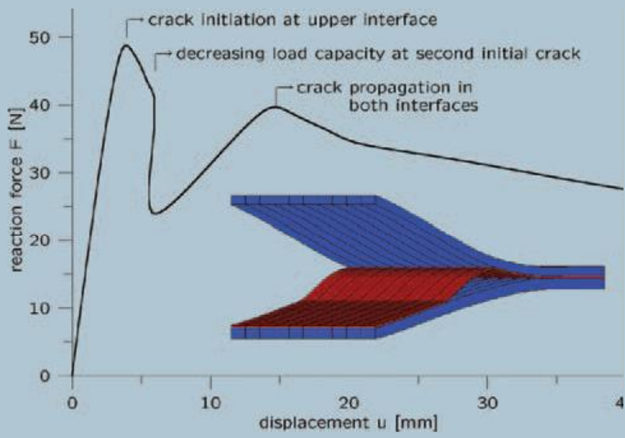

Figure 6: Load-Displacement Curve

The strong nonlinear global load-displacement curve (Figure 6) with different local limit points is mainly influenced by the preexisting cracks and shows good results compared to experimental data.

Summary and Outlook

The implementation of cohesive elements also in commercial finite element software represents a substantially extension of the simulation capabilities. Beside the calculation of mechanical behavior of thin interface layers and durability investigations of component connections up to simulation of crack propagation and failure of structures exist many new application fields.

Further development of the finite element method for discrete failure is a current research topic. Two main, but generally different approaches in discrete fracture finite element simulation are on the one hand adaptive generation of additional cohesive elements during the analysis and on the other hand representation of discontinuous displacement fields by enhanced shape functions on the level of continuum elements.

References

[1] Barenblatt, G.I.: The mathematical theory of equilibrium cracks in brittle fracture. Advances in Applied Mechanics, 1962, 7, 55-129.
[2] Needleman, A.: A continuum model for void nucleation by inclusion debonding. Journal of Applied Mechanics, 1987, 54, 525-531.
[3] Corigliano, A.; Mariani, S.; Pandolfi, A.: Numerical modeling of rate-dependent debonding processes in composites. Composite Structures, 2003, 61, 39-50.
[4] Geißler, G.; Kaliske, M.: A cohesive element approach for viscoelastic interface properties. Proceedings in Applied Mathematics and Mechanics, 2004, 4, 191-192.
[5] Wille, K.; Dehn, F. & Tue, N.V.: Bruchmechanische Kenngrößen hochfester Leichtbetone. Bauingenieur 80, 2005, 327-333.
[6] Alfano, G.; Crisfield, M.A.: Finite element interface models for the delamination analysis of laminated composites: mechanical and computational issues. International Journal for Numerical Methods in Engineering 50, 2001, 1701-1736.

Authors
Prof. Dr.-Ing. habil. Michael Kaliske,
Gordon Geißler
Universität Leipzig, Institut für Statik und Dynamik der Tragstrukturen, www.uni-leipzig.de/statik

Contact
Dr.-Ing. Armin Fritsch, CADFEM GmbH, Grafing
Phone +49-(0)8092-7005-40, E-Mail afritsch@cadfem.de