2D Modeling of Elastic Wave Propagation in Solids Containing Closed Cracks

S. Delrue[1], V. Aleshin[2], O. Bou Matar[2], K. Van Den Abeele[1]

[1]Wave Propagation & Signal Processing Research Group, KU Leuven Kulak, Kortrijk, Belgium
[2]Institute of Electronics, Microelectronics and Nanotechnologies, Lille, France

Abstract

Within the field of Non Destructive Testing (NDT) of materials, nonlinear ultrasonic techniques are becoming increasingly popular, since they provide extreme sensitivity in detecting the presence of incipient damage. However, the next step forward would be to fully characterize the detected defects (e.g. by estimating their geometric parameters), allowing to make some prediction about lifetime or serviceability of the tested samples and structures. This can for instance be done by comparing the experimentally obtained nonlinear indicators with the results obtained by an effective numerical model. On the other hand, such numerical models can also be used to assist in the further development and optimization of the existing experimental ultrasonic NDT techniques.

In order to obtain a better understanding and analysis of the nonlinear behavior of early-stage damage features, we developed and investigated the results of a two-dimensional numerical model for elastic wave propagation in solid materials containing closed cracks. The model contains two components: the constitutive crack model, and the wave propagation model.

In the constitutive crack model, implemented in MATLAB®, a real crack in a structure is approximated by a number of mesoscopic cells (Figure 1a). In each cell we search for a link between loads (normal N and tangential T) and displacements (normal a and tangential b). The mesoscopic load-displacement relationship integrates the microscopic contact behavior and takes into account roughness of internal crack faces and friction between them, together with the associated effects of memory and hysteresis. The normal reaction curve N(a) is obtained using conventional models of contact mechanics. The tangential reaction (Figure 1b) is calculated using the original method of memory diagrams that automates and greatly simplifies the account for friction and hysteresis [1]. The full constitutive model allows to describe three different defect states: contact loss, total sliding, and partial slip when both stick and slip areas are present in the contact zone.

The wave propagation problem is implemented in the Structural Mechanics Module of COMSOL Multiphysics®. Internal cracks are modeled using the 'thin elastic layer' boundary condition that allows using the customized constitutive relationships by introducing the above described MATLAB function into the COMSOL model using the LiveLink™ for MATLAB®. At each time step of the procedure, COMSOL calculates normal and tangential

displacements at each mesh node on the crack interface. These displacements are then used as an input in the MATLAB function in order to calculate the corresponding normal and tangential loads at these mesh nodes. Finally, these loads are re-introduced in the thin elastic layer boundary condition in COMSOL. Figure 2 shows a schematic overview of this procedure.

The final objective of this model is to create some kind of a numerical laboratory capable of modeling various nonlinear experiments in different kinds of materials and geometries and for different defect configurations.

Reference

[1] Vladislav Aleshin et al., Method of memory diagrams for mechanical frictional contacts subject to arbitrary 2D loading, Int J Solids Struct 60-61, 84-95 (2015)

Figures used in the abstract

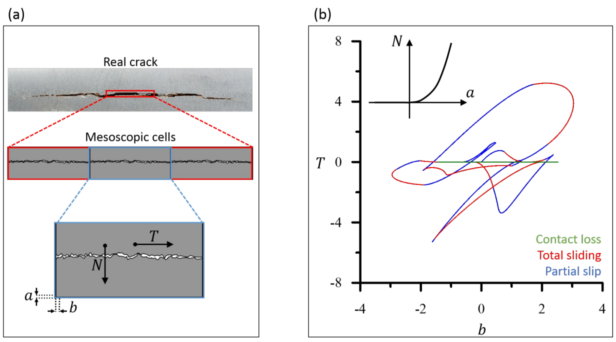

Figure 1: (a) Real crack in a sample, approximated by a number of mesoscopic cells in which loads (normal N and tangential T) and displacements (normal a and tangential b) are defined; (b) Normal and tangential load-displacement curves for a non-trivial displacement history in which the defect states switch several times.

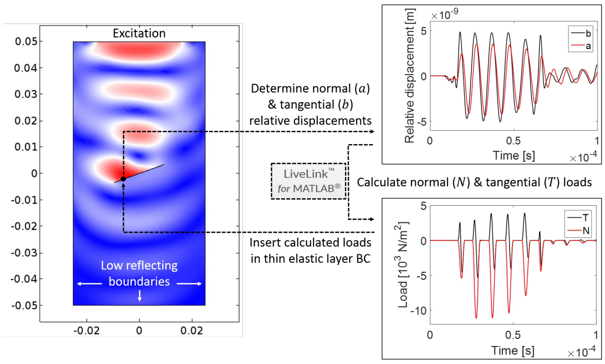

Figure 2: Schematic overview of the followed procedure in the COMSOL simulation. Left figure shows a snapshot of the time domain simulation in COMSOL. During each time step, the normal (a) and tangential (b) relative displacement are calculated at each mesh node on the crack interface. Using MATLAB, normal (N) and tangential (T) loads are calculated using the displacements. Finally, these loads are re-introduced as internal boundary data, before the next time step is being calculated.