Advances in Engineering Software 74 (2014) 27–39

Contents lists available at ScienceDirect

Advances in Engineering Software

journal homepage: www.elsevier.com/locate/advengsoft

An open source program to generate zero-thickness cohesive interface elements

Vinh Phu Nguyen *,[1]

School of Engineering, Institute of Mechanics and Advanced Materials, Cardiff University, Queen's Buildings, The Parade, Cardiff CF24 3AA, United Kingdom

ARTICLE INFO

Article history:
Received 5 November 2013
Received in revised form 30 March 2014
Accepted 17 April 2014
Available online 15 May 2014

Keywords:
Interface cohesive elements
Fracture mechanics
Cohesive zone models
Finite element method
Open source code
Intrinsic cohesive laws

ABSTRACT

An open source program to generate zero-thickness cohesive interface elements in existing finite element discretizations is presented. This contribution fills the gap in the literature that, to the best of the author's knowledge, there is no such program exists. The program is useful in numerical modeling of material/structure failure using cohesive interface elements. The program is able to generate one/two dimensional, linear/quadratic cohesive elements (i) at all inter-element boundaries, (ii) at material interfaces and (iii) at grain boundaries in polycrystalline materials. Algorithms and utilization of the program is discussed. Several two dimensional and three dimensional fracture mechanics problems are given including debonding process of material interfaces, multiple delamination of composite structures, crack propagation in polycrystalline structures.

Introduction

Cohesive crack models (CM) which were pioneered in [1,2] is a continuation of linear elastic fracture mechanics with which the unrealistic stress singularity ahead the crack tip is avoided. Application of CMs as fracture models used in the context of the finite element method however appeared substantially later in [3]. From a numerical point of view, CMs have been incorporated in a finite element (FE) context using zero-thickness interface elements, elements with embedded discontinuities and elements with discontinuous enrichment via the extended/generalized finite element method (XFEM/GFEM). A comparative study on the modeling of discontinuous fracture using these techniques was given in [4] and a review of computational methods for fracture in quasi-brittle solids has been recently reported in [5]. It should be emphasized that the term "cohesive elements" usually used to refer to cohesive interface elements is misleading since elements with embedded cohesive cracks or XFEM with cohesive cracks are also cohesive elements. Therefore, in this contribution, we suggest the name "cohesive interface elements" (in subsequent discussion interface elements are used for brevity) to indicate interface elements equipped with a cohesive law.

Elements with embedded discontinuities are a powerful tool to model crack propagation, see e.g., [6–9], among others. Its most distinct advantage is the simplicity of implementation into existing FE codes compared to XFEM. Recently XFEM, which is a local partition of unity (PUM) [10] based finite element method, pioneered in [11] has become the dominant numerical method for both weak discontinuities (material interfaces) [12–14] and strong discontinuities (cracks), see e.g., [15–17] among others. Its popularity is arguably due to the fact that XFEM can be applied to numerous varieties of problems (fluid mechanics, biofilm growth, multiphysics, etc.). However, it comes with complications including numerical integration of elements crossed by the discontinuities, complexity concerning the implementation of the method in existing finite element packages. Furthermore, finding the enrichment functions to model intersecting discontinuities is not a feasible task, especially for three dimensions. Also, parallelizing the XFEM is not straightforward.

For polycrystalline structures, a common fracture phenomenon is inter-granular cracking, i.e., cracks propagate along the grain boundaries. This fracture mechanism is naturally modeled with interface elements that are inserted, prior to computation, along the grain boundaries because the crack path is known in advance. Recently, this problem was tackled with the generalized finite element method (GFEM) [18]. The advantage of GFEM is that only a structured simple mesh is required since the grain topology is independent of the mesh and modeled with special enrichment functions. However extending these enrichment functions to three dimensions and intra-granular fracture is not easy. Moreover,

* Tel.: +44 0 29 2087 0524.
E-mail address: nguyenpv@cardiff.ac.uk

[1] This work was carried out when the author was a PhD candidate at the Faculty of Civil Engineering and Geosciences, Delft University of Technology, The Netherlands under the supervision of Prof. Bert Sluys.

http://dx.doi.org/10.1016/j.advengsoft.2014.04.002
0965-9978/

imposing Dirichlet boundary conditions on enriched nodes is not a trivial task. In contrary, using interface elements, both two dimensional (2D) and three dimensional (3D) inter and intra-granular fracture can be modeled. Such a 2D application has been recently reported in [19] and 3D cases in [20].

Delamination of laminated composites has been traditionally modeled using interface elements [21–23]. Recently, PUM-based finite elements has been used to model delamination with mesh independent of the delamination surface [24,25]. However, to model 3D delaminated composites, PUM-based method must employ either a layer of 3D solid elements or solid-like shell elements which increases the computational efforts significantly. Furthermore, PUM-based method is not suited for two dissimilar structures bonded together such as skin-stringer interface. Using interface elements, an effective cohesive element for shell analysis was presented in [26].

In addition to the natural application of interface elements for problems in which crack paths are known *a priori* such as material interface debonding [27,28], delamination, inter-granular cracking, other advantages of interface elements include (i) crack initiation does not rely on some fracture criteria, (ii) straightforward implementation, and (iii) complex fracture mechanism such as crack branching, crack coalescence can be handled with ease [29,30,19,31,20]. Other applications of interface elements are mesoscopic modeling of concrete materials [32], dynamic fracture and fragmentation of solids [33,34] and shell fracture [35,36]. Other computational methods that are capable of dealing with complex fracture mechanics problems (with large deformation) include meshfree methods, e.g., [37–42], and peridynamics see [43,44] among others.

There are two ways by which one can implement interface elements. In the first approach, interface elements are introduced, where necessary, before the simulation starts. The cohesive laws used in this approach is referred to as *intrinsic cohesive laws*. In the second approach, interface elements are inserted during the simulation and the corresponding cohesive laws are referred to as *extrinsic cohesive laws*. Both approaches have their own advantages and shortcomings. In the former, the shortcomings are (i) mesh sensitivity (this however does not apply for problems in which the crack path is known in advance, for example delamination analysis, inter-granular fracture of polycrystals or debonding of material interfaces) and (ii) reliance on a high dummy stiffness to model the perfect bond prior to fracture. Mesh sensitivity issue can be eliminated with sufficiently refined random meshes. For the artificial compliance introduced by the interface elements, one remedy is to initially constraint the interface elements (using the master–slave method [19] or a discontinuous Galerkin [45,20]) and only active them when necessary. Another possibility is to find the proper value for the dummy stiffness for interface elements as in [46]. In the latter, by only introducing interface elements when a fracture criterion is met all the shortcomings of intrinsic cohesive elements are removed. However, it brings complex implementation and parallelism issues.

Although implementation of intrinsic cohesive elements is straightforward (and thus available in major commercial FE packages), the pre-processing stage in which one has to insert interface elements into an existing finite element mesh is quite complicated and there does not exist any open source tool to do that. The aim of this manuscript is therefore to present a simple pre-processing program which is able to insert one and two dimensional interface elements in a finite element mesh. This program is independent of mesh generation programs and freely downloadable on our website. We believe that it is helpful for researchers doing fracture analysis using interface elements, which play an important role despite of the emergence of novel methods such as XFEM, but do not have access to commercial preprocessors. It should also be useful for Discontinuous Galerkin based analyses [47]. The code also generates interface elements that are suitable for discontinuous Galerkin based methods. Moreover, through the open source code, adaptation to individual needs is much easier than using commercial preprocessors. The tool has been used in [48–50] and we hope through this manuscript it will be shared with the fracture and the discontinuous Galerkin communities. For completeness, this paper also presents implementation details of 2D interface elements in a nonlinear FE setting.

The structure of the rest of the paper is as follows. In Section "Finite element formulation", the governing equations of a cracked solid are given together with the semi-discrete equations discretized by finite elements. Section "Automatic generation of interface elements" presents the algorithms that are used to insert one and two dimensional interface elements in a finite element mesh. In the next section, the installation and usage of the presented program is discussed. Various numerical examples are given in Section "Numerical examples" including material interface debonding, two dimensional crack propagation problems in homogeneous structure and in polycrystalline structure; two and three dimensional delamination analysis. Finally some conclusions are drawn in Section "Conclusions".

Finite element formulation

Strong and weak forms

The governing equations include the equilibrium equation, the natural, essential boundary conditions and the traction continuity on the crack surface

$$\nabla \cdot \boldsymbol{\sigma} + \mathbf{b} = 0 \quad \mathbf{x} \in \Omega \tag{1a}$$

$$\mathbf{n} \cdot \boldsymbol{\sigma} = \bar{\mathbf{t}} \quad \mathbf{x} \in \Gamma_t \tag{1b}$$

$$\mathbf{u} = \bar{\mathbf{u}} \quad \mathbf{x} \in \Gamma_u \tag{1c}$$

$$\mathbf{n}_d^+ \cdot \boldsymbol{\sigma} = \mathbf{t}_c^+; \mathbf{n}_d^- \cdot \boldsymbol{\sigma} = \mathbf{t}_c^-; \quad \mathbf{t}_c^+ = -\mathbf{t}_c = -\mathbf{t}_c^- \quad \mathbf{x} \in \Gamma_d \tag{1d}$$

where $\boldsymbol{\sigma}$ is the Cauchy stress tensor, $\mathbf{u}$ is the displacement field and $\mathbf{b}$ is the body force vector. The traction applied over the boundary Γ_t with outward unit normal vector $\mathbf{n}$ is denoted by $\mathbf{t}$, $\bar{\mathbf{u}}$ is the applied displacement over the Dirichlet boundary Γ_u; $\Gamma_u \cup \Gamma_t = \Gamma, \Gamma_u \cap \Gamma_t = \emptyset$; $\mathbf{t}^c$ is the cohesive traction across the crack Γ_d with unit normal vector $\mathbf{n}_d$. For simplicity, only small strain assumption is considered where the strain is taken as the symmetric part of the displacement gradient $\boldsymbol{\epsilon}^s = \frac{1}{2}(\nabla \mathbf{u}^s + \nabla^{\mathrm{T}} \mathbf{u}^s)$. Constitutive relations for the bulk and the cohesive crack are given later.

The weak formulation reads, see [51] for details

$$\delta W^{\mathrm{ext}} = \delta W^{\mathrm{int}} + \delta W^{\mathrm{coh}} \tag{2}$$

with

$$\delta W^{\mathrm{int}} = \int_{\Omega} \nabla^s \delta \mathbf{u} : \boldsymbol{\sigma} \mathrm{d}\Omega \tag{3a}$$

$$\delta W^{\mathrm{ext}} = \int_{\Omega} \delta \mathbf{u} \cdot \mathbf{b} \mathrm{d}\Omega + \int_{\Gamma_t} \delta \mathbf{u} \cdot \bar{\mathbf{t}} \mathrm{d}\Gamma_t \tag{3b}$$

$$\delta W^{\mathrm{coh}} = \int_{\Gamma_d} \delta [\![\mathbf{u}]\!] \cdot \mathbf{t}^c \mathrm{d}\Gamma_d \tag{3c}$$

where $[\![\mathbf{u}]\!]$ denotes the displacement jump (see Fig. 1).

Discretization

The bulk is discretized by standard continuum elements and the discontinuity surfaces (cracks, material interfaces) are discretized by zero-thickness interface elements which are one dimensional