International Journal of Adhesion & Adhesives 31 (2011) 357–362

Contents lists available at ScienceDirect

International Journal of Adhesion & Adhesives

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

Simulation of adhesive joints using the superimposed finite element method and a cohesive zone model

Young Tae Kim*, Min Jung Lee, Byung Chai Lee

Department of Mechanical Engineering, Korea Advanced Institute of Science and Technology, 373-1, Guseong-dong, Yuseong-gu, Daejeon 305-701, Republic of Korea

ARTICLE INFO

Available online 15 February 2011

Keywords:
Finite element stress analysis
Superimposed finite element method
Cohesive element
Adhesive joint

ABSTRACT

Adhesive joints have been widely used in various fields because they are lighter than mechanical joints and show a more uniform stress distribution if compared with traditional joining techniques. Also they are appropriate to be used with composite materials. Therefore, several studies were performed for the simulation of the bonded joints mechanical behavior. In general for adhesive joints, there is a scale difference between the adhesive and the substrate in geometry. Thus, mesh generation for an analysis is difficult and a manual mesh technique is needed. This task is not efficient and sometimes some errors can be introduced. Also, element quality gets worse.

In this paper, the superimposed finite element method is introduced to overcome this problem. The superimposed finite element method is one of the local mesh refinement methods. In this method, a fine mesh is generated by overlaying the patch of the local mesh on the existing mesh called the global mesh. Thus, re-meshing is not required.

Elements in the substrate are generated. Then, the local refinement using the superimposed finite element method is performed near the interface between the substrate and the adhesive layer considering the shape of the element, the element size of the adhesive layer and the quality of the generated elements. After performing the local refinement, cohesive elements are generated automatically using the interface nodes. Consequently, a manual meshing process is not required and a fine mesh is generated in the adhesive layer without the need for any re-meshing process. Thus, the total mesh generation time is reduced and the element quality is improved. The proposed method is applied to several examples.

1. Introduction

Adhesive joints have been widely used in various fields because they are light and present relatively a more uniform stress distribution if compared with traditional joining techniques. In addition, adhesive joints are appropriate to be used with composite materials because they avoid holes or a welding process. As the use of adhesive joints is enlarged, there have been many studies conducted to describe the behavior of adhesive joints and predict their failure load [1–5].

The geometry of adhesive joints consists of a very thin adhesive layer and thick substrates. The substrate thickness is usually in the order of 10–1000 times thicker than the adhesive layer thickness. However, failure characteristics of adhesive joints are determined by the thin adhesive layer. Thus, the modeling method and the mesh density of the adhesive layer influence the numerical results of the adhesive joints [6–11]. Therefore, the mesh density in the adhesive layer needs to be fine. To achieve this, a manual mesh technique may be required to connect the rough elements in the substrate with the fine mesh in the adhesive layer. This technique reduces the modeling efficiency, element quality and can sometimes introduce certain errors by the user.

In this study, the superimposed finite element method is introduced to overcome this problem. As one of the mesh refinement technique, the superimposed finite element method was proposed by Fish [12]. The idea of this method is to overlay a fine local mesh into the concerned area, which is discretized as a rough global mesh. A re-meshing process is not needed by introducing the superimposed finite element method, and the performance is similar to a fine mesh. Due to this advantage, this method is applied to various fields such as laminated composites [13], elasto-dynamic problems [14], crack propagation [15], shape optimization [16], etc. Recently, an efficient superimposed finite element method has been proposed by Park et al. [17]. As the boundary in the global mesh coincides with the boundary in the local mesh, computational cost is reduced. This method is applied to adhesive joints for an efficient generation of a fine mesh in the thin adhesive layer. An initial global mesh is generated in the substrate. Then, a fine local mesh is

* Corresponding author. Tel./fax: +82 42 350 5031.
E-mail address: ytaekim@gmail.com (Y.T. Kim).

0143-7496/$ - see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.ijadhadh.2010.11.015

overlaid on the global mesh located at the interface between the adhesive and the substrate. For describing the transient behavior, a finer local mesh is generated near the interface as a hierarchical local mesh is superimposed. Finally, cohesive elements are generated in the adhesive layer using the global nodes and local nodes at the interface.

This method has some merits. The element quality is maintained after superimposing the local mesh because a fine local mesh can be generated, which maintains the quality of the global elements. In addition, the number of global elements is reduced because the fine mesh is only generated near the adhesive layer. Finally, a manual mesh generation technique is not required and thus, the modeling time can be reduced and user errors can be excluded.

In this paper, a cohesive zone model is used at the interface as the modeling method and an in-house code is used as the analysis tool. The numerical results achieved by using the superimposed finite element method are compared with the load-opening displacement curve of a double cantilever beam by using the Timoshenko beam theory to confirm that the proposed method is appropriate. The numerical results achieved by the proposed method are also compared with the three point bending test results of a mixed-mode single leg bending joint.

2. Superimposed finite element method

2.1. Description

The governing equation of the finite element method is obtained from the variation of the total potential energy functional $\mathbf{\Pi}$ in Eq. (1).

$$\mathbf{\Pi}=\frac{1}{2}\int_{\Omega}\varepsilon^{T}\mathbf{D}\varepsilon\,d\Omega-\int_{\Gamma}\mathbf{u}^{T}t\,d\Gamma-\sum_{i}\mathbf{u}_{i}^{T}\mathbf{f}_{i} \tag{1}$$

where Ω is the domain, ε is the strain, $\mathbf{D}$ is the material matrix, t and Γ are the traction boundary condition and the region where the traction is applied, $\mathbf{f}_i$ and $\mathbf{u}_i$ are the external force and the displacement in the point where the external force is applied. The variation of Eq. (1) is Eq. (2)

$$\begin{aligned}\delta\mathbf{\Pi}&=\int_{\Omega}\delta\varepsilon^{T}\mathbf{D}\varepsilon\,d\Omega-\int_{\Gamma}\delta\mathbf{u}^{T}t\,d\Gamma-\sum_{i}\delta u_{i}^{T}f_{i}\\&=\sum^{N}\int_{\Omega^{e}}\delta(\varepsilon^{e})^{T}\mathbf{D}\varepsilon^{e}\,d\Omega^{e}-\int_{\Gamma}\delta\mathbf{u}^{T}t\,d\Gamma-\sum_{i}\delta\mathbf{u}_{i}^{T}\mathbf{f}_{i}=0\end{aligned} \tag{2}$$

where Ω^e is the domain of each element, ε^e is the strain of each element, N is the number of elements.

For the superimposed finite element method, the displacement is defined by Eq. (3).

$$\mathbf{u}=\begin{cases}\mathbf{u}_G=N_G u_G & \text{on } \Omega_G\\ \mathbf{u}_G+0=N_G u_G & \text{on } \Gamma_{GL}\\ \mathbf{u}_G+\mathbf{u}_L=N_G u_G+N_L u_L & \text{on } \Omega_L\end{cases} \tag{3}$$

where N is the shape function, G and L are the global domain and local domain, respectively. In the global domain, the displacement vector is represented by the general interpolation function called the shape function. In case of the two dimensional element, the shape function of the global domain is like in Fig. 1. In the local domain, the displacement is represented by the sum of the global displacement vector and local displacement vector. In this domain, interpolation function must be used the hierarchical shape function. The hierarchical shape function is obtained by adding new shape functions to the existing ones. In Fig. 2, the local element patch is composed of four elements. For presenting the behavior of local elements, some shape functions are added in the original shape functions like in Fig. 3. This modified shape functions and added

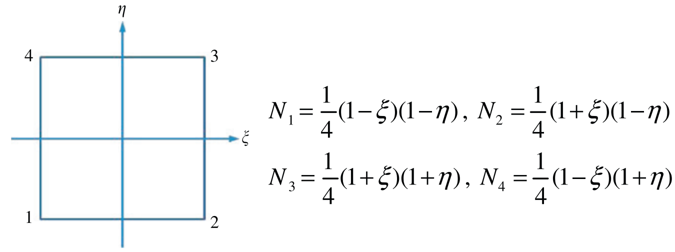

Fig. 1. Two dimensional element and shape functions.

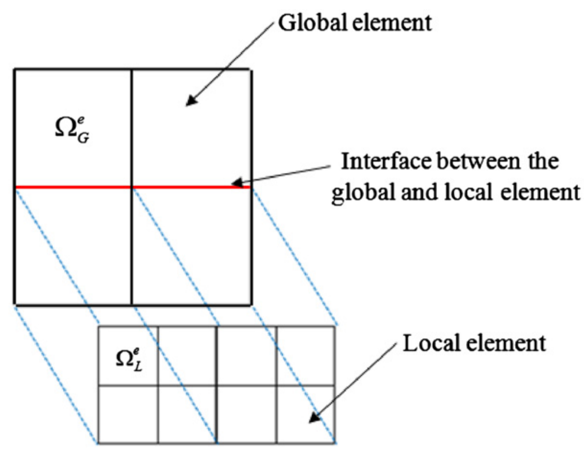

Fig. 2. Global mesh and local mesh for the superimposed finite element.

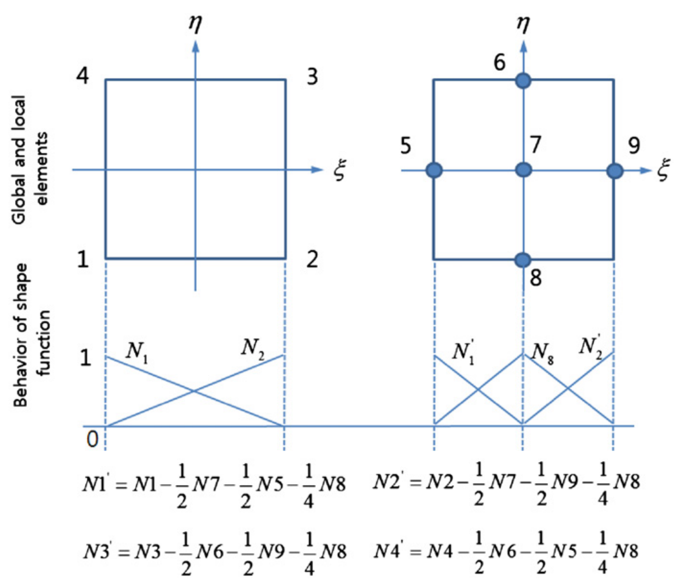

Fig. 3. Local element patch composed by four elements and its hierarchical shape functions in two dimensional element.

shape functions are called the hierarchical shape function. At the interface between the global domain and the local domain, the displacement vector is represented by the global displacement only for the displacement continuity.

The strain vector, which is the derivative of the displacement is represented by Eq. (4).

$$\varepsilon=\begin{cases}\varepsilon_G=B_G u_G & \text{on } \Omega_G\\ \varepsilon_G+0=B_G u_G & \text{on } \Gamma_{GL}\\ \varepsilon_G+\varepsilon_L=B_G u_G+B_L u_L & \text{on } \Omega_L\end{cases} \tag{4}$$