

Numerical Analysis of Static Behavior in a Three-point Bending Test of Aluminum Foam Sandwich Beams using the Extended Finite Element Method

Youngwon Hahn

Dassault Systèmes SIMULIA

Seong Sik Cheon

Kongju National University

Copyright © 2009 SAE International

ABSTRACT

In this paper, the numerical analysis of a three-point bending test of an aluminum foam sandwich structure is performed with the new extended finite element feature supported by Abaqus 6.9. The sandwich beam consists of two aluminum skins and one aluminum foam core. Three different sets of model dimensions are selected for comparison with the reference results (J. Yu, E. Wang, J. Li, Z. Zheng, "Static and low-velocity impact behavior of sandwich beams with closed-cell aluminum-foam core in three-point bending", International Journal of Impact Engineering, 35, 2008, pp 885-894). Failure modes in this paper can be categorized into three parts: face yield (FY), indentation (IN), and core shear (CS). Face yield occurs on the surface of the core when the thickness of the skin is small. Indentation and core shear occur if the thickness of the skin is relatively large. Failure modes in the numerical simulation were found to be in good agreement with the test results obtained from the reference paper.

INTRODUCTION

The foam sandwich structure is widely used and very attractive in many industrial fields due to its particular properties (low specific weight, high damping, high energy absorption, etc). It shows generally better performance in both shear and bending loadings, compared to a honeycomb sandwich structure which performs well only in bending. [1].

In this paper, an aluminum foam sandwich (AFS) structure is considered. The AFS structure consists of thin metal/composite skin sheets and an aluminum foam core. It is used in many practical applications such as cookware, rockets, racing bicycles, etc. It is also used in the automotive industry in order to increase crash performance due to its efficient energy absorption capabilities.

The three-point bending test is one of the classical tests used to determine the static behavior in bending loading. It is generally used to test the foam sandwich structure. There are several papers that study its dynamic and static behaviors under three-point bending (or four-point bending) conditions [2, 3]. Numerical studies have also been performed on foam sandwich structures. Styles et al [4] performed finite element analysis and compared with test results [2]. He also performed parametric studies.

For the application of the foam sandwich structure, Kim et al [5] did parametric study of dynamic stiffness and damping characteristics for a precision machine tool structure. Hanssen et al [6] performed a numerical simulation of a bird strike on an aluminum foam sandwich panel and compared its failure with dynamic test results.

Even though there are many papers concerning numerical studies of foam sandwich structures, there are few papers that address the prediction of the crack initiation location and propagation in a static test since the numerical modeling for the crack is cumbersome and requires much effort.

In this paper, the numerical analysis of crack initiation and propagation in an aluminum foam sandwich structure in three-point bending is performed with the new crack modeling capability in Abaqus 6.9. The new capability uses a special numerical technique called the eXtended Finite Element Method (XFEM) which can predict crack initiation and propagation using a conventional finite element model. Three different AFS structures are simulated and their results are compared with the test results obtained by Yu et al [7].

The failure mode in Yu et al [7] is categorized into three parts: face yield (FY), indentation (IN), and core shear

(CS). Face yield occurs on the surface of the core when the thickness of the skin is small. Indentation and core shear occur if the thickness of the skin is relatively large.

This paper consists of three parts: First, the XFEM numerical technique is introduced briefly. Second, the three numerical models of AFS are described. Finally the numerical results are shown and compared with test results.

XFEM in Abaqus

The eXtended Finite Element Method (XFEM) was first introduced by Belytschko and Black [8]. It extends the conventional finite element method with additional enrichment functions in order to define a crack discontinuity.

The enrichment function used in Abaqus is shown in equation (1).

$$\mathbf{u} = \sum_{I=1}^N N_I(x) [\mathbf{u}_I + H(x) \mathbf{a}_I + \sum_{\alpha=1}^4 F_{\alpha}(x) \mathbf{b}_{I\alpha}] \quad (1)$$

where $N_I(x)$ are the conventional nodal shape functions used in the finite element method; the first term on the right-hand side of the above equation, $\mathbf{u}_I$, is the nodal displacement vector; the second term is the product of the nodal enriched degree of freedom vector, $\mathbf{a}_I$, and the associated discontinuous jump function $H(x)$ across the crack surfaces; and the third term is the product of the nodal enriched degree of freedom vector, $\mathbf{b}_{I\alpha}$, and the associated elastic asymptotic crack-tip functions, $F_{\alpha}(x)$. The second term is valid only for nodes whose shape function support is cut by the crack interior; and the third term is used only for nodes whose shape function support is cut by the crack tip [9]. This method is only applicable in a stationary crack.

The alternative approach to simulate crack initiation and propagation within the framework of XFEM is based on traction-separation cohesive behavior and the phantom node concept [10]. This XFEM-based cohesive segments method can be used to simulate crack initiation and propagation along an arbitrary, solution-dependent path in bulk materials, since the crack propagation is not tied to the element boundaries in a mesh. In this case, the near-tip asymptotic singularity is not needed, and only the displacement jump across a cracked element is considered. Therefore, the crack must propagate across an entire element at a time to avoid the need to model the stress singularity.

If there is no crack, each phantom node is completely constrained to its corresponding real node in the element. However, if an element is cut by a crack, some real nodes in the cracked element separate from the phantom nodes. The magnitude of the separation is

governed by the cohesive law until the cohesive strength of the cracked element is zero, after which the phantom and the real nodes move independently. The discontinuity is represented by a displacement interpolation using real and phantom nodes in each of the split elements. This method provides an effective and attractive engineering approach and has been used for the simulation of initiation and growth of multiple cracks in solids [10]. It has proven to exhibit almost no mesh dependence if the mesh is sufficiently refined [9].

Another key point for crack geometry with XFEM is the level set method [11]. This makes it possible to model an arbitrary crack without remeshing. The crack is defined by two almost orthogonal signed distance functions, ϕ and ψ . The first, ϕ , describes the crack surface, while the second, ψ , is used to construct an orthogonal surface so that the intersection of the two surfaces gives the crack front as shown in Figure 1.

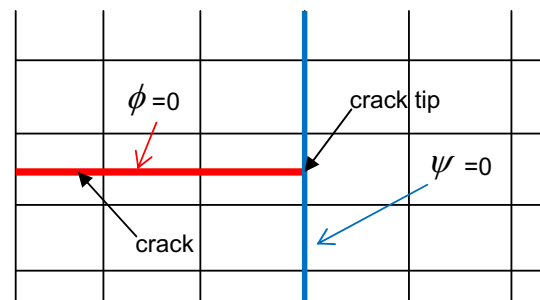


Figure 1. Signed distance functions to describe the crack.

Damage modeling is achieved by a traction-separation cohesive law as shown in Figure 2. Currently two criteria are available for damage initiation: the maximum principal stress criterion and the maximum principal strain criterion [9].

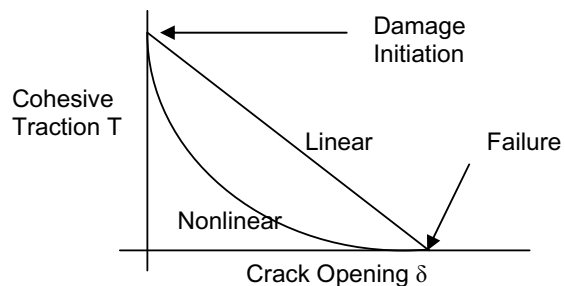


Figure 2. Linear/Nonlinear damage model.