Composite Structures 106 (2013) 326–339

Contents lists available at SciVerse ScienceDirect

Composite Structures

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

Numerical assessment of the impact behavior of honeycomb sandwich structures

Costantino Menna *, Alberto Zinno, Domenico Asprone, Andrea Prota

Department of Structures for Engineering and Architecture (DiSt), University of Naples Federico II, Via Claudio 21, 80125 Napoli, Italy

ARTICLE INFO

Article history:
Available online 22 June 2013

Keywords:
Phenolic sandwich structures
Nomex
Damage mechanisms
Numerical simulation

ABSTRACT

Composite sandwich structures are widely used in the high-performance applications where weight reduction is one of the most attractive design parameters. However, structural sandwich components have low resistance to out-of-plane impact due to the thin outer composite skins and the highly deformable cores. The present paper deals with a finite element study on the impact response of sandwich panels, obtained combining phenolic resin-based glass fiber reinforced plastics as skins and phenolic resin-impregnated aramid paper honeycomb structure (Nomex) as core. The numerical analysis has been performed using the LSDYNA software enabling to account for the main sandwich failure modes occurring during impact. The honeycomb core structure and composite skins have been modeled by means of solid and shell elements respectively. The properties of the finite element model have been calibrated on a series of experimental outcomes in order to achieve numerical parameters for both composite facesheet and orthotropic honeycomb material models. The major concerns are related to damage mechanisms, influence of strain-rate effects and energy absorbing capability. The model is validated using the results from experimental impact tests performed on different initial impact conditions. Good agreement was obtained between numerical and experimental results in terms of impact damage and force–displacement trend.

1. Introduction

Due to their advantageous specific mechanical properties (in terms of stiffness-to-weight ratio) and corrosion resistance, composite sandwich structures are becoming an attractive alternative to metals in many different engineering applications, especially for mass transport applications where weight reduction is one of the most important design parameter. Although the peculiar morphology of sandwich structures offer very attractive properties, their application is often restricted by their vulnerability to transverse impact [1],

Structural sandwich components generally exhibit a low resistance to out-of-plane impact due to the thin outer composite skins and the highly deformable cores. When localized transverse loading is applied to a sandwich structure, the facesheet locally deflects up to failure and the core crushes, leading to a damage ranging from permanent indentation to complete penetration [1,2]. The loss of load carrying capability they suffer as a consequence of impact events can be light or significantly injurious depending on the failure mechanisms which irreversibly have taken place. Hence in order to reliably predict the structural behavior and guarantee structural safety of composite sandwich structures, a deep understanding of the impact behavior (e.g. impact and penetration damage varying in the range of low to high velocities and impacting masses) has become essential. Available literature offers a wide range of studies on low velocity/energy impact response of sandwich structures in terms of experimental research, analytical formulations and numerical implementations [3–8].

Depending on the initial impact energy, impacts can induce damage of different entity to the skins, the core material, and the core-facing interface. However, the impact performance and the damage extent of sandwich composites depend on several factors (e.g. skin and core materials, geometries, boundary conditions [3]). Furthermore, the different damage mechanisms may take place individually or interact. In the case of low-energy impacts, the response of sandwich panels may be governed by bending and little damage occurs if the kinetic energy of the impacting object is absorbed elastically by the panel. For high-energy impacts, a failure condition is reached when local contact stress exceeds local strength triggering a damage sequence given by the laminate bending failure, core/skin interface delamination, core compression failure and its plastic deformation up to complete penetration [1]. Core deformation and failure represent therefore crucial factors for the energy absorption capability and impact behavior of sandwich panels [5].

Experimental and analytical studies have been conducted to understand the mechanical response of honeycomb sandwich struc-

* Corresponding author. Tel.: +39 081 7683672; fax: +39 081 7685921.
E-mail address: costantino.menna@unina.it (C. Menna).

0263-8223/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compstruct.2013.06.010

tures composed by different skins and core materials under various loadings [6–10]. Numerical methodologies based on finite element (FE) approach represent a standard tool in the development process of composite and sandwich application industries allowing time and test-prototyping cost savings. The main source of complexity associated with FE impact modeling of sandwich structures is related to the adequate material constitutive models, proper prediction of the damage level in each sandwich constituent, and the definition of interaction laws for various damage mechanisms. Several strategies have been developed in the FE simulation environment for this class of structural materials. A combination of testing and numerical analysis is provided by the multiscale approach [11], where various levels of structural complexity are progressively validated numerically through associated experimental tests up to a complete prediction of the full scale product behavior. Especially for cellular based, folded core materials, virtual testing FE strategies are often developed. According to this approach, specific material properties of cellular cores with complex geometries are achieved with detailed FE simulation models on a parametric basis, overcoming the lack of experimental databases [12,13].

Furthermore, the design of sandwich panels for high dynamic loads requires to have information about the sandwich dynamic structural behavior and in particular on the influence of loading rate on the material properties. In case of high loading rate an increase in material stiffness and strength compared to the static behavior may occur. When this effect is neglected, dynamic FE simulations based on static material data often disagree with experimental dynamic results and therefore attention should be paid when design approaches are based on static data. Strain rate effects can affects both skins and core material. For former material types, especially when rate sensitive materials are used, a Dynamic Increase Factor (DIF) may be characterized [14–16]. Concerning core materials, strain rate effects on axial behavior of both aluminum [17,18] and Nomex [19,20] honeycomb structures have been experimentally investigated through dynamic compressive tests performed with different techniques (i.e. drop weight, gas gun, and split Hopkinson bar). These results reported that dynamic loading leads to a marginal increase of the initial stiffness and peak compressive strength, a significant increase of the crush strength and a reduction of the deformation for the fully compacted final region.

1.1. Significance and scopes

In the present paper, a numerical strategy is presented in order to proper simulate the dynamic behavior of Nomex core sandwich structures combined with E-glass phenolic facesheet under impact load through the finite element code LS-DYNA. Emphasis has been put on the complete penetration process occurring during impact. The adopted procedure is based on a multiscale virtual testing approach by means of the progressive validation of the numerical sandwich model on the basis of experimental tests, ranging from coupon tests (for E-glass phenolic skins and Nomex core material) to sandwich assembly tests. This allows to verify the capability of the model to take into account the main damage mechanisms taking place during penetration. Finally, the dynamic impact behavior is modeled on this basis and validated through different impact conditions, highlighting important issues on the strain rate sensitiveness affecting the analyzed sandwich plates. The main outcomes in terms of force displacement curves, energy absorption and damage mechanisms are assessed and compared to the experimental results.

2. Materials and methods

In the present study the impact behavior of phenolic-impregnated sandwich structures has been investigated. The composite skins were obtained from pre-impregnated satin-weave E-glass fiber reinforced phenolic resin skins with a cured ply thickness of 0.25 mm. The sandwich core is made of 48 kg/m^3 Nomex hexagonal honeycomb with a nominal cell size of 3.18 mm and made of phenolic resin-impregnated aramid paper. Since the skins are pre-impregnated, there is no additional adhesive used to bond the skins to the core. The materials were laminated in an autoclave at a temperature of 135 °C, a vacuum pressure of 2.5 bar and a curing time of 90 min. All the sandwich specimens considered in the following experimental activities have been assembled with the "*L*" direction of the honeycomb core (Fig. 1) along the primary direction and different configurations (in terms of size and thickness) for skins and Nomex core are handled in the paper.

Different experimental tests were performed in order to characterize the static and dynamic behavior of the examined sandwich structures. The results are used to get the basic information to build up the numerical sandwich model and to validate it. Further details about the experimental activity, which are not presented hereafter, are available in [21]. The experimental tests concern the sandwich constituent level (in plane mechanical characterization of the phenolic skins, compressive behavior of Nomex core) and the sandwich assembly (four point bending tests and indentation tests). The numerical simulation of these tests allows to calibrate the constitutive models adopted for the sandwich materials at different scale level and to validate the capability of the model to predict the occurrence of damage mechanisms. The dynamic impact behavior is finally investigated through the simulation of the drop weight tests performed at different initial impact conditions. In the following sections, first the characteristics of the adopted material models are presented and afterwards, the details of the experimental tests and the numerical models are illustrated for each type of test.

3. Modeling of sandwich materials

A finite element model has been built up through the commercial FE code LS DYNA (Version 971) in order to model the sandwich impact behavior. The FE sandwich model consists of two different sandwich materials including solids and shell elements. Particularly, in order to save computational time and avoid instabilities (due to large deformations) homogeneous equivalent solid elements have been adopted for the Nomex honeycomb core, shell elements for the phenolic facesheets and solid elements for the rigid steel impactor. A failure based nodal connection has been used to model the bonding between the Nomex core and the phenolic facesheets. The material models adopted for the sandwich materials are described in the following sections.

3.1. E-glass phenolic skins

In order to adequately model the complete process of penetration in a sandwich structure, the energy absorbing capability and

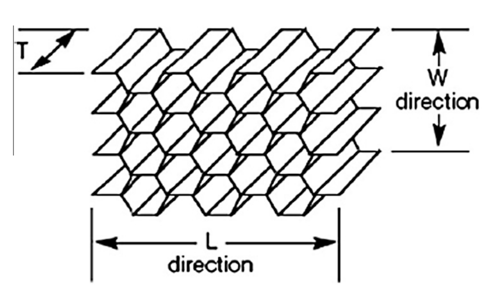

Fig. 1. Shape geometry of hexagonal honeycomb core.