Composite Structures 107 (2014) 1–10

Contents lists available at ScienceDirect

Composite Structures

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

Study on low-velocity impact of embedded and co-cured composite damping panels with numerical simulation method

Sen Liang [a,*], Keyao Liang [b], Lei Luo [a], Qian Zhang [a], Changsong Wang [a]

[a] School of Mechanical Engineering, Qingdao Technological University, Shandong Province 266033, PR China
[b] School of Mathematics, Shandong University, Jinan City, Shandong Province 250100, PR China

ARTICLE INFO

Article history:
Available online 6 August 2013

Keywords:
Impact performance
Embedded and co-cured composite damping structure
Finite element analysis
Numerical simulation

ABSTRACT

According to the designable mechanical characteristics of composite material, the embedded and co-cured composite damping structure (ECCDS) consisting of fiber reinforced resin layers and one (or more) damping layer(s) embedded into composite material component is manufactured. The finite element model and experiment model of ECCDS are established and low velocity impact performances of the composite damping panels are simulated and tested by using the explicit dynamic analysis software LS-DYNA and the Drop-weight Impact equipment, respectively. The structural behaviors under impact of the damping panels are compared with those of the composite panels without the damping layer. The results of numerical simulation and experiment are in good agreement with each other, which demonstrates that the finite element model and simulation method using present study are valid. Finally the impact properties versus the different geometrical parameters of damping layers are investigated also. The simulation model and method can provide an important foundation for the theoretical predict of impact characteristics of ECCDS.

1. Introduction

Impact loading is one of the frequent loadings encountered. Structures in a variety of engineering fields such as aeronautical, automotive and civil engineering industries suffer impact loading very often [1–6]. Typical examples of impact on structures include the bird or debris strike on the flying vehicles and car collision in traffic accident. The embedded and co-cured composite structure (ECCDS) is a new type of pre-processing damping structure [7,8] which consists of fiber reinforced resin matrix and viscoelastic damping material layers embedded into the composite. In this composite structure, the location and distribution of viscoelastic layers can be determined according to the design requirement. Compared with the traditional damping structure, the damping layer of this structure is co-cured into the composite material, which protects it from the influence of external environment. Therefore this structure has many advantages over a free damping structure and a constrained damping structure, such as anti-aging, not desquamation and long-life etc. As a result, many people focused their researches on this structure [7–11] as soon as the embedded and co-cured composite damping structure was presented by Rotz and Barrett [12] in 1990's. Recently, the researches mainly involved in the co-cured processing [8,10,12] and the experiment measurement [13–15] as well as analytical prediction [16,17] of damping performance, etc. According to the authors' investigation, although many mechanical characteristics of the embedded and co-cured composite have been developed, very few studies on impact performance of this structure have been reported so far and thus the aim of this investigation is to explore the capability of impact resistance of the innovative embedded and co-cured composite. In this paper, LS-DYNA explicit dynamic analysis software is employed to simulated impact process of ECCDS. The finite element model and experiment model of ECCDS are established and low velocity impact characteristics of the composite damping panels are simulated and tested by using the explicit dynamic analysis software and the Drop-weight Impact equipment, respectively. The structural behaviors under impact of the damping panels are compared with those of the composite panels without the damping layer. The experiment results including the impact energy absorbed, impact peak force and damage property & shape of the composite damping panel are recorded and analyzed. The numerical simulation and experiment results are in good agreement with each other, which demonstrates that the finite element model and simulation method using present study is very valid. Finally, the impact properties versus the different geometrical parameters of damping layers are investigated also. The research conclusions can provide an important foundation for the advanced development of damping composite material of the integrative function-structure, which has strong potential to resist impact.

* Corresponding author. Tel.: +86 532 85071299; fax: +86 532 85071699.
E-mail address: liangsen98@mailst.xjtu.edu.cn (S. Liang).

0263-8223/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compstruct.2013.07.056

2. Impact behaviors of ECCDS with finite element analysis

Using ANSYS/LS-DYNA to analyze the impact behaviors of ECCDS is very important, which can be classified into the preprocessing and post-processing two sections.

2.1. Preprocessing

Finite element model of ECCDS are established by using ANSYS/LS-DYNA in this section, which includes the determination of 3D geometrical model, the selections of element and algorithm, the definition of material properties, meshing, the creation of PARTs and the definition of contact type. The geometrical model of impact simulation consists of two parts, the one is impact hammer and the other is specimen. The hammer is made up of a cylinder (radius = 10 mm, height = 10 mm) and a semi-sphere (radius = 10 mm). The specimen diameter is 100 mm, the other geometrical parameters and stacking sequences are shown in Table 1. After meshing the structure with hexahedral element, Fig. 1 shows a numerical finite element model. The circumferential boundary of ECCDS is clamped and the constraint nodes are in annular area between the inside diameter 81 mm and outside diameter 100 mm. The friction factor of contact parts between the steel hammer and specimen is 0.150 according to our experiment result and contact stiffness can be automatically calculated by LS-DYNA software according to geometric structure and material model in the impact analysis. But the factor FKN is input 0.8, which references LS-DYNA Theoretical Manual [18].

Three specimen models are established in detail, the one is fiber reinforced resin matrix composite specimen without damping layer, and the other two are ECCDS with 0.1 mm and 1 mm thickness damping layers embedded, respectively. In this paper, 8-node brick element SOLID164 is selected to use in this finite element analysis, and default Lagrangian algorithm is employed here. In the finite element model, three types of material models are selected, which are rigid material model, Chang-Chang composite failure model and incompressible Mooney-Rivlin rubber model. To save computing time in simulation, the hammer is defined as a rigid body and keyword MAT_RIGID is used to define the material of hammer [19]. In order to simplify model, the hammer in finite element analysis does not include the transducers and auxiliary devices, so the material density of hammer is modified to ensure that the mass of hammer model equals to the mass of actual hammer. The material parameters of hammer model are listed in Table 2.

The failure model developed by Chang and Chang in 1988 [20] is selected to simulate the composite failure in ECCDS, and keyword MAT_COMPOSITE_ DAMAGE is used to define the composite material. Three failure criteria are employed in this model, matrix crack failure, matrix compression failure and fiber breakage failure. Besides elastic moduli (EA, EB, EC), shear moduli (GAB, GAC, GBC) and Poisson's ratios (NUBA, NUCA, NUCB), five parameters are used in these criteria, which are shear strength (SC), longitudinal tensile strength (XT), transverse tensile strength (YT), transverse compressive strength (YC) and nonlinear shear stress (ALPH) [21,22]. All the parameters of composite material model are seen in Table 3.

The incompressible Mooney-Rivlin rubber model is used to simulate the viscoelastic material in ECCDS. The strain energy density function is defined as

$$W(I_1, I_2, I_3) = A(I_1 - 3) + B(I_2 - 3) + C(I_3^{-2} - 1) + D(I_3 - 1)^3 \qquad (1)$$

Where A and B are model constants, $C = \frac{1}{2}A + B$, $D = \frac{A(5\upsilon-2)(11\upsilon-5)}{2(1-2\upsilon)}$ and υ is Poisson's ratio. I_1, I_2 and I_3 are strain tensor invariants [21,22]. Table 4 shows viscoelastic material parameters.

ANSYS/LS-DYNA pre-processor can automatically generate different PARTs according to different materials. The contact type should be defined between different parts. Keyword CONTACT_ERODING_ SURFACE_TO_SURFACE is used to define the contact between hammer and specimen, which illustrates that this contact of the composite and damping layer with impact hammer is an eroding surface to surface contact. The material and element size of hammer are different from those of specimens, so the Soft Constraints Option (SOFT) is set to be 2. To make the calculation convergence fast, the Time Step Scale Factor (TSSFAC) has to be reduced to 0.1 [19,23]. In experiment, the delamination does not happen between the composite and viscoelastic materials or between composite layers. So the delamination is not considered in this simulation.

Table 1
Stacking sequences and geometric parameters.

	Specimen 1	Specimen 2	Specimen 3
Number of fiber glass prepregs (layer)	4	4	2
Thickness of viscoelastic material (mm)	0	0.1	1
Total thickness of specimen (mm)	0.8	0.9	1.4
Diameter of the cut specimen (mm)	Φ99.85	Φ99.92	Φ99.92
Stacking sequence	$[0_4]$	$[0_2/b]_s$	$[0_2/b]_s$

Where b denotes the viscoelastic layer.

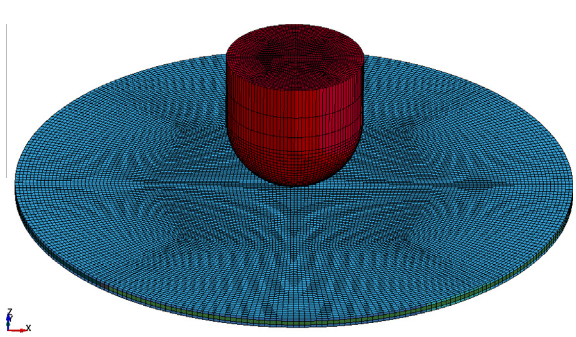

Fig. 1. The numerical finite element model.

Table 2
Material model parameters of the hammer.

Density, RO (kg/m^3)	1.461 × 106
Elastic modulus, E (GPa)	209
Poisson's ratio, PR	0.29

Table 3
Material parameters of fiber reinforced resin matrix.

Density, RO (kg/m^3)	2000
Elastic moduli, EA, EB, EC (GPa)	24.3, 24.3, 10.4
Poisson's ratios, PRBA, PRCA, PRCB	0.138, 0.4, 0.4
Shear moduli, GAB, GBC, GCA (GPa)	3.7, 3.7, 3.7
Shear strength, SC (MPa)	124
Longitudinal tensile strength, XT (MPa)	480
Transverse tensile strength, YT (MPa)	459
Transverse compressive strength, YC (MPa)	399
Nonlinear shear stress parameter, ALPH	0.25

Table 4
Viscoelastic material model parameters.

Density, RO (kg/m^3)	920
Poisson's ratio, PR	0.498
Coefficient A (MPa)	0.40
Coefficient B (MPa)	0.11