Ultrasonics 84 (2018) 187–200

Contents lists available at ScienceDirect

Ultrasonics

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

Simulation of guided-wave ultrasound propagation in composite laminates: Benchmark comparisons of numerical codes and experiment

Cara A.C. Leckey [a,*], Kevin R. Wheeler [b], Vasyl N. Hafiychuk [c], Halyna Hafiychuk [c], Doğan A. Timuçin [b]

[a] *NASA Langley Research Center, Hampton, VA 23681, United States*
[b] *NASA Ames Research Center, Moffett Field, CA 94035, United States*
[c] *Stinger Ghaffarian Technologies, Inc., Moffett Field, CA 94035, United States*

ARTICLE INFO

Article history:
Received 16 June 2017
Received in revised form 2 November 2017
Accepted 3 November 2017
Available online 4 November 2017

Keywords:
Ultrasound
Guided wave
Simulation
Composite
Delamination

ABSTRACT

Ultrasonic wave methods constitute the leading physical mechanism for nondestructive evaluation (NDE) and structural health monitoring (SHM) of solid composite materials, such as carbon fiber reinforced polymer (CFRP) laminates. Computational models of ultrasonic wave excitation, propagation, and scattering in CFRP composites can be extremely valuable in designing practicable NDE and SHM hardware, software, and methodologies that accomplish the desired accuracy, reliability, efficiency, and coverage. The development and application of ultrasonic simulation approaches for composite materials is an active area of research in the field of NDE. This paper presents comparisons of guided wave simulations for CFRP composites implemented using four different simulation codes: the commercial finite element modeling (FEM) packages ABAQUS, ANSYS, and COMSOL, and a custom code executing the Elastodynamic Finite Integration Technique (EFIT). Benchmark comparisons are made between the simulation tools and both experimental laser Doppler vibrometry data and theoretical dispersion curves. A pristine and a delamination type case (Teflon insert in the experimental specimen) is studied. A summary is given of the accuracy of simulation results and the respective computational performance of the four different simulation tools.

1. Introduction

In recent decades, the aerospace industry has seen a rapid growth in the use of composite materials since this class of materials can enable advanced lightweight aircraft and spacecraft designs. While the increased use of composites is expected to continue due to their weight benefit and tailorability, these materials also pose unique challenges for post-manufacture certification; as well as for in-service inspection. Common defect types that occur in composite materials include delamination damage, porosity, and microcracking [1,2]. Practical and reliable nondestructive evaluation (NDE) and structural health monitoring (SHM) methods for detection and quantification of such defects/damage are of key importance for enabling the certification and ensuring the safety of aerospace vehicles with composite parts.

Currently, ultrasonic methods constitute the leading physical mechanism used for NDE and SHM of aerospace composite materials such as carbon-fiber-reinforced polymer (CFRP) laminates. Computational ultrasound models (analytical, semi-analytical, and numerical) solve the equations of motion for a composite part with specified initial and boundary conditions. Numerical methods such as finite element (FE), spectral element (SE), and finite difference (FD) can incorporate detailed composite material properties and complex damage morphologies into ultrasound models. These high-fidelity ultrasonic wave propagation models can enable optimal NDE and SHM hardware, data processing tool designs, and inspection methodologies to provide the desired inspection accuracy, reliability, efficiency, and coverage for composite structures.

Within the last decade, a growing number of authors have reported the implementation of three-dimensional (3D) ultrasonic numerical simulations for composite materials. Ng et al. discussed the need for including three-dimensional (3D) damage representations in wave simulations of composite laminates, and used a 3D FE method to simulate guided waves in a quasi-isotropic laminate [3]. These authors modeled each individual layer in the quasi-isotropic laminate, using the assumption of a homogeneous orthotropic material properties for each ply. Simple circular-geometry delaminations of various radii were incorporated into the simulations and the FE results were then compared to analytical models. Singh et al. reported using a commercial FE code to simulate guided waves in a composite laminate with homogenized material proper-

---

* Corresponding author.
E-mail address: cara.ac.leckey@nasa.gov (C.A.C. Leckey).

https://doi.org/10.1016/j.ultras.2017.11.002
0041-624X/Published by Elsevier B.V.

ties through the thickness (*i.e.*, individual ply layers were not simulated) [4]. This team studied guided-wave interaction with a simulated cone-shaped defect representing impact damage. Leckey et al. used a custom 3D finite integration code to study guided-wave propagation in anisotropic composite laminates (simulating each ply layer), and incorporated a realistic damage geometry using X-ray computed tomography data of impact-induced delamination damage [5]. Murat et al. report using a custom FE code to simulate guided wave propagation in a cross-ply composite laminate, and specifically studied wave interaction with a square-shaped delamination [6]. More recently, Kudela et al. report using a custom graphics processing unit (GPU) based parallelized 3D SE code to study guided wave propagation in a crossply composite plate for both a homogenized ply case and a model case incorporating every individual ply layer. Kudela and colleagues found that modeling each ply layer is appropriate for guided wave simulations [7].

The intent of the simulation studies reported in this paper is to determine benchmark comparisons establishing the accuracy and the computational requirements of various numerical codes for simulating ultrasonic guided-wave propagation in composite laminates. Several considerations enter into the practical implementation of a simulation code, thereby rendering each code unique in its details. These include:

- represented spatial scale (fiber-, ply-, or plate-level specification of constitutive relationships, fine or coarse representation of defects);
- spatio-temporal discretization of governing equations of motion and boundary conditions (mesh shape, mesh density);
- spatio-temporal duration of simulation (localized *vs.* extended response, space-time *vs.* wavenumber-frequency domain computation);
- solver parameters (controlling stability, convergence, *etc.*).

The choices made in fixing these details for a particular problem must depend, to a large degree, on the experimental scenario that the numerical simulation is intended to represent. The chosen parameters essentially represent a trade-off between the accuracy and the stability of the code on the one hand, and its memory and computational runtime requirements on the other. While custom-developed codes can provide the user with significant flexibility in some of these details, taking proper advantage of such a capability requires a deep understanding of both the underlying physics and its numerical implementation on the part of the user. On the other hand, commercial software codes frequently "hard-wire" some of these details in order to provide easy access to a larger community of users. Proper validation of simulation tools is required for both custom and commercial codes in order to ensure that the simulation setup and implementation are appropriate for the physics experiment under study.

In making an informed decision about the choice and the use of a computational modeling tool, the availability of benchmark problems with associated experimental data sets and simulation studies is indispensable. In this paper, we report on two simulation case studies involving guided ultrasonic waves in (*i*) a pristine CFRP laminate, and (*ii*) a CFRP laminate containing a single delamination-type defect of known size and location. Guided wave simulations were performed for these simulation cases using four different simulation codes: the commercial finite element modeling (FEM) packages ABAQUS, ANSYS, and COMSOL, and a custom code executing the Elastodynamic Finite Integration Technique (EFIT). COMSOL, ANSYS and ABAQUS are implemented with an implicit time solution. Additionally, ABAQUS is also implemented in an explicit time-stepping mode, and EFIT is also explicit in time. For both CFRP simulation cases, comparisons are performed between the simulated guided wavefield results from the four different simulation tools and wavefield results from experiment. In addition, all wavefield results are compared with dispersion curve predictions.

In Section 2, the geometry and composition of the pristine and delaminated experimental specimens are documented, along with the experimental setup including the excitation source. Section 3 then gives a detailed description of the simulation tools used in this benchmarking study, focusing particularly on their resolution and stability requirements. Section 4 describes the experimental and simulation results for the pristine specimen, comparing codes on the basis of their wavenumber spectra and group velocity values. Section 5 reports on the experimental and simulation results for the delaminated specimen, showing time-domain wavefield images as well as wavenumber spectra for the various codes. Section 6 discusses the computational resource requirements of each simulation tool. Lastly, Section 7 summarizes the findings of this benchmarking study and discusses areas of future work.

2. Experimental setup

2.1. Composite specimens

In order to generate an example problem and representative data sets that anchor the simulation studies, two IM7/8552 CFRP test panels were fabricated at NASA Langley Research Center. IM7/8552 is a high-performance composite material used for aerospace applications. Table 1 lists elastic material properties for a single ply of IM7/8552 from values reported in the scientific literature [8–10]. It is noted that the material properties listed in the table are based on standard ASTM testing procedures to determine properties of composite materials. However, the materials properties of an as-manufactured composite component are affected by factors such as the age of the prepreg material, conditions during curing, level of compaction achieved during curing, etc. Therefore, it is expected that in any real scenario there will be variation between one as-manufactured composite specimen and properties acquired based on testing of another as-manufactured specimen. Methods for improved and rapid determination of the properties of as-manufactured composites without destructive testing are an area of active research [11].

The panels were made using eight plies of IM7/8552 material cured in a cross-ply layup of $[0_2/90_2]_s$ with an overall thickness of 0.92 mm and size of 38 cm × 38 cm. One of the panels was pristine while the other had a double-layered Teflon film in the shape of a 20 mm by 20 mm square inserted between the second and third ply layers. Fig. 1 shows the Teflon location through the sample thickness and a diagram representing the portion of the plate containing the Teflon and transducer. In the x-y plane of the plate (shown in the diagram), the center of the insert is located 12.7 cm from the left plate edge, 25.4 cm from the right edge, 25.4 cm from the top edge and 12.7 cm from the bottom edge. The Teflon insert served to mimic a delamination-type defect.

2.2. Excitation and measurement

For both specimens a GE Inspection Technologies Gamma Series (TCG-999) 0.5 MHz normal incidence contact piezo-electric transducer (PZT) was used to excite guided ultrasonic waves in the composite specimens. The transducer is a disk-shaped actuator with an overall diameter of 19 mm. The transducer was coupled to one side of the panel, see Fig. 1, and was driven by a 300 kHz 3-cycle Hann-windowed sine wave. The center frequency of the excitation signal was chosen to ensure that only two guided wave modes would be generated for the thicknesses of the experimental specimens. This