3pEA5. Comparison of finite element models simulating the interaction of ultrasonic guided waves with sites of disbonding in composites

Peter McKeon*, Slah Yaacoubi, Nico F. Declercq, Salah Ramadan and Weina K. Yaacoubi

*Corresponding author’s address: Georgia Institute of Technology, 2 rue Marconi, Metz, 57070, N/A, France, peter.mckeon@gatech.edu

Disbonding in composite structures is a serious defect which can dramatically reduce the structures’ life, and can lead to catastrophic failure. To avoid this, non-destructive testing or structural health monitoring techniques are needed. One such technique involves ultrasonic guided waves, which has recently found use in this field thanks to its ability to inspect in non-accessible areas and over long distances. Numerical models are often used since they can help understand experimental results and offer the ability to simulate different damage scenarios, predicting results with less cost than experiments. In this work, sites of disbonding between an orthotropic composite plate and an isotropic polyamide plate were modeled via the finite element method. Two simplistic methods for modeling the damage site are employed, and results will be compared with experimental data. Model types include introduction of a geometrical void at the interface boundary and addressing boundary conditions between adjacent layers. The amount of mode conversion after interaction with the damage site can be used to evaluate the validity of each model type. Results are discussed in terms of assumption made and physical accuracy.
INTRODUCTION

Techniques for non-destructive testing (NDT) or continuous structural health monitoring (SHM) utilizing ultrasonic guided waves have been applied to many industries in the past 20 years. Some examples of industries that have benefitted from such techniques include the railway and power generation industries [Papaelias et al. (2008)]. Recently, work is being carried out to extend these methods to more complex materials and geometries [D. Cerniglia (2010); Castaings and Hosten (2001)], in the attempt to identify, size and locate a range of defects. Composite materials, which are finding increased use in transportational technologies, are one such material that demonstrate a new range of difficulties due to their anisotropic nature. Often these materials are layed on top of thinner isotropic material layers to give form, connected by the same epoxy that binds the composite itself. Accordingly, a common defect type for this particular structure is disbonding between the two layers.

Modeling via the finite element method (FEM) has become a common practice, especially when analytical solutions are difficult or unattainable [Terrien et al. (2007); Alleyne and Cawley (1992); Singh et al. (2011)]. FEM offers the ability to model complex geometries and defect types either in the time domain or frequency domain. Often material symmetries can be exploited to simplify the model from three to two dimensions. However, the same damage type can often be modeled in many different ways. It is often appropriate to compare different approaches modeling damage in the context of FEM with experimental data to select the approach that best predict realistic behavior.

In this presentation, two simplistic methods will be presented to model a disbond between two layers, intended to be quickly and easily applied to existing FEM software. The layers consist of a thicker orthotropic carbon-epoxy layer, and a thinner isotropic polyamide layer. The two methods will be further simplified from three to two dimensions by modeling planar wave propagation along a principal direction, taking advantage of material symmetry. The first method consists of an elliptical void between the two materials layers. The second method consists of adjusting boundary conditions between the two layers at the site of the disbond. Both FEM approaches will be carried out in the frequency domain.

FINITE ELEMENT MODEL

The finite element model used in this work consists of two materials layered on top of each other. The thicker upper layer corresponds to a carbon-epoxy material, and the thinner lower layer corresponds to a polyamide. Material properties are withheld for confidentiality reasons.

Simulations are carried out in the frequency domain according to the formulation of the Helmholtz equation:

$$C_{ijkl} \frac{\partial^2 u_j}{\partial x_k \partial x_l} - \rho \omega^2 u_i = 0, \quad i,j,k,l = 1,2,3$$

which assumes a harmonic solution in time at a given frequency. The two-dimensional model assumes plane strain in the third dimension. Central frequency is chosen to be $f_c = 200$kHz. This $f_c$ is chosen for two reasons: it provides wavelengths of a similar size as the intended defect to be model, and 200 kHz centered transducers are readily available to the authors to ease experimental corroboration. The mesh elements are triangular and use quadratic interpolation. Mesh resolution is set at 6 elements per wavelength of the average modal wavelength, which experience has shown to be sufficient. A tighter mesh is used at and around the damage sites.

In order to suppress end reflections, viscoelastic absorbing boundary regions (ABR) are applied to each end of the plate [Hosten and Castaings (2005)]. For these regions, the components of the stiffness matrix are altered to the form

$$C_{ij}^{ABR} = C_{ij} \left( 1 + iA \left( \frac{x_{ABR}^3}{L_0^3} \right) \right), \quad i,j = 1,2$$

where $L_0$ is the length of the ABR, and $A$ is a constant for optimization. The independent variable $x_{ABR}$ is defined as being equal to zero at the ABR border and increasing in magnitude in the direction away from the propagation domain.
A schematic of the healthy waveguide (without damage) can be seen in figure 1. In real world experiments, out of plane motion on the surface of the plate is not only accessible, but also a common parameter monitored by already existing SHM methods [Michaels et al. (2011)]. Therefore, to distinguish between propagating modes, a series of inspection points are monitored on the top surface of the carbon-epoxy layer. These inspection points are equally spaced and record out-of-plane (y-direction) motion only. This method of investigating the results of the FEM is most convenient for comparison with experimental results.

The excitation is modeled as a force on the upper side of the plate (see figure 1). The excitation zone is chosen to be 1 cm long, for two reasons: 1 cm corresponds to roughly 1.5 wavelengths of Mode 2, and 1 cm diameter contact transducers are readily available to the authors for experimentation. The force is applied in the in-plane direction (x-direction).

In the following sections, two methods of modeling disbonding will be presented. The damage sites will be between the excitation zone and the inspection points. Therefore, the inspection points will be monitoring the transmitted wavefield, after the incident wave has passed through the damage zone.

### Disbond Modeling

A simple method for modeling disbonding is to create a geometrical void corresponding to the location of the disbond. A schematic of this model type is shown in figure 2A. Although in this model there are two parameters that can be adjusted, namely ‘h’ and ‘w’, ‘h’ is kept constant in this work (0.7 mm). Since Mode 2 is the primary mode excited and detected in the healthy waveguide, the values for ‘w’ are given in terms of actual length and percent wavelength of Mode 2 in table 1.

Each width is simulated and the results are analyzed based on the Fourier transform of the results collected from the inspection points, which allows for the separation of the modes via wavenumber. Although there are five guided wave modes present at this particular frequency-thickness, the highest two modes have similar wavenumbers and can not be resolved. The results from the elliptical void disbond model are shown in figure 3. It can be seen that the ratio of modal amplitudes vary greatly with the width. This is due to mode conversion after the incident wave interaction with the defect. One can imagine that the inverse problem has
Table 1: Dimensions of elliptical void width in meters and percent of Mode 2 wavelength

<table>
<thead>
<tr>
<th>w [mm]</th>
<th>% ( \lambda_2 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.85</td>
<td>10 %</td>
</tr>
<tr>
<td>1.28</td>
<td>15 %</td>
</tr>
<tr>
<td>2.13</td>
<td>25 %</td>
</tr>
<tr>
<td>2.84</td>
<td>30 %</td>
</tr>
<tr>
<td>4.27</td>
<td>50 %</td>
</tr>
<tr>
<td>6.4</td>
<td>75 %</td>
</tr>
</tbody>
</table>

Figure 3: Normalized modal amplitude in the transmission zone for the elliptical voids.

Potential for defect sizing.

A different approach involves altering the boundary condition for the defect region. Consider figure 2B. In this figure, four independent variables are used: \( u_1 \) and \( v_1 \) describe displacement in the upper layer, and \( u_2 \) and \( v_2 \) describe \( x \) and \( y \) displacement respectively in the lower layer. Accordingly the boundary conditions need to be established between the layers. Normally \( u_1 = u_2 \) and \( v_1 = v_2 \), but these relationships can be altered from their norm to model a disbond. In this work, each variable is treated as free in the defect zone.

Experimental Results

Experimental work to corroborate the two disbond modeling methods is planned for the immediate future. The experiments consist of materials that match those used in the finite element work, with similar thicknesses. Excitation is carried out with a shear contact transducer, pulsed with a central frequency of 200 kHz. Reception of the waveforms is accomplished with an air coupled transducer (centered at 200 kHz), attached to a robotic arm capable of 0.1 mm resolution. Disbands are created in a controlled environment by not applying the adhesive that binds the two layers over a specific area. Inspection will take place along the same principal direction modeled in the FEM work. Modal amplitudes will be compared with figure 3.

Conclusion

The two methods presented here are intended to be simple to apply. Both methods make assumptions in order to model a disbond defect between two layers. The elliptical disbond model takes away from physical geometry of the plate. That is to say, it may more accurately model certain disbands where are large gap occurs between the two layers, but it's inaccuracies stem from the physical change in plate thickness around the defect zone. Alternately, the method altering boundary conditions allows thicknesses to remain the same, but also allows for the two layers to occupy the same space, a physical impossibility. Experimental results will be able to verify the accuracy of these two simplistic models.
ACKNOWLEDGMENTS

This work is a part of H2E project which is supported in part by OSEO foundation. The authors would like to thank their partners (Air Liquide, EADS Composites Aquitaine,...) for their collaboration.

REFERENCES


