Experimental and numerical analysis of sandwich composite
Delamination behavior
Adikeshavan P*, Divyabarathi P, Syedhaleem M
Department of Aeronautical Engineering, Bharath University, Selaipur, Chennai-600073
*Corresponding author: E-Mail: adikeshavan.aero@bharathuniv.ac.in

ABSTRACT

The damage study performed numerically and experimentally using double cantilever beam (DCB) sandwich composite specimen under mode I loading. A Modified Compliance Calibration Method has been followed. This method incorporates the complex damaging phenomena of the fracture due to the material and geometric irregularity of the specimen and produce the values fracture energy and cohesive strength respectively. The experimental setup was numerically simulated using ABAQUS/Standard solver. Satisfactory agreement between the numerical and the experimental compliance vs. crack-length curve validates the accuracy of the ensued data reduction procedure.

KEY WORDS: Sandwich Composites; Delamination; Fracture Toughness.

1. INTRODUCTION

The use of sandwich composite materials for structural applications is widely common and advanced composite materials have been replacing traditional materials due to their better specific properties. At first, these materials were exclusively used in aerospace and aeronautical industries. Successively, these materials started to be used in several different applications with the technological developments and reduction of manufacturing costs which includes biomedical, automotive vehicles and small boats.

For the past twenty years, in the general industrial sector, an important increase in the use of composite materials particularly polymeric matrix materials reinforced with glass or carbon fibers along with light weight core has taken place, since they offer excellent stiffness to weight and strength to weight ratios, these properties make materials effective for many applications. For the extensive use and of composite materials, it requires a good knowledge of their structural behaviour and their failure mechanisms, which are quite complex and need to be studied carefully. For sandwich composites face/core interface delamination is the most critical forms of failure. Delamination can occur due to quasi-static or cyclic loading and different approaches are usually required for its study depending on the type of loadings. One of the most used current approaches for the study of the delamination under quasi-static loading is the cohesive surface interface modelling. One aim of this work is to examine the potential of cohesive surface interface for the prediction of delamination propagation under quasi-static mode I loading.

2. EXPERIMENTAL WORK

The experimental analysis was performed using a sandwich composite consisting of glass/epoxy face sheets over medium density PVC foam was selected. The face sheets consisted of Glass Fiber Chopped strand mat and Glass Fibre woven roving. The core material consisted on PVC Divinycell H100 foam with 0.1 g/cc density. Epoxy resin and hardener matrix was used for bonding face sheet and core material. The laminates, with dimensions of 400 mm × 400 mm, were fabricated following a 0° lay-up of Glass Fibre Chopped strand mat, Glass Fibre woven roving and PVC core, using vacuum resin infusion moulding technique and cured under plate press for 12 hours.

Material Properties: Properties of face sheet and core material given in Table 1.

Table 1. Material Properties

<table>
<thead>
<tr>
<th>Facesheet (E-Glass/Epoxy)</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>E₁</td>
<td>14540 MPa</td>
</tr>
<tr>
<td>E₂</td>
<td>11910 MPa</td>
</tr>
<tr>
<td>ν₁₂</td>
<td>0.23</td>
</tr>
<tr>
<td>G₁₂</td>
<td>2950 MPa</td>
</tr>
<tr>
<td>G₁₃</td>
<td>1600 Mpa</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Core (PVC Divinycell H100)</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>E</td>
<td>130 Mpa</td>
</tr>
<tr>
<td>ν</td>
<td>0.13</td>
</tr>
</tbody>
</table>

The DCB specimen was cut into 25 mm wide and 125 mm long from fabricates panel. Before fabrication process is started teflon film was introduced between face and core for ensuring initial debond. To achieve a natural crack, load was slowly applied to the specimen until the crack opened up over the region covered by the insert film. The DCB specimen was cut into 25 mm wide and 125 mm long from fabricates panel. Before fabrication process is started teflon film was introduced between face and core for ensuring initial debond. To achieve a natural crack, load was slowly applied to the specimen until the crack opened up over the region covered by the insert film. The DCB
test specimens were tested under displacement control at a speed of 1 mm/min, such a slow rate will allow monitoring the crack propagation with a traveling microscope.

**Fracture Energy Measurement**: Fracture Energy calculated experimentally using Modified Compliance Calibration Technique is tabulated below in Table.2.

### Table.2. Fracture Energy Values

<table>
<thead>
<tr>
<th>Pre-Crack length (mm)</th>
<th>Width b (mm)</th>
<th>Height H (mm)</th>
<th>Ult. Load Pc (N)</th>
<th>Ult. Load Deflec- tion δc (mm)</th>
<th>Fracture Energy (Glc) (Nmm/mm²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>50</td>
<td>25</td>
<td>35</td>
<td>174.2</td>
<td>3.695</td>
<td>1.043</td>
</tr>
<tr>
<td>50</td>
<td>25</td>
<td>35</td>
<td>167.2</td>
<td>3.432</td>
<td>1.060</td>
</tr>
<tr>
<td>50</td>
<td>25</td>
<td>35</td>
<td>190.9</td>
<td>3.443</td>
<td>1.123</td>
</tr>
<tr>
<td>50</td>
<td>25</td>
<td>35</td>
<td>164.1</td>
<td>3.185</td>
<td>1.066</td>
</tr>
</tbody>
</table>

**Numerical Implementation**: All the finite element analyses presented in this work are performed with the element finite code ABAQUS 6.10, of the ABAQUS Inc.

ABAQUS/Standard is utilized for numerically simulating the experimental work. It is a general-purpose solver using a traditional implicit integration scheme to solve finite element analyses which have been utilized in this project. FEM model for the DCB tests contains three parts: upper beam, lower beam and a cohesive surface in between. The upper beam consisting of Top Facesheet modeled based on orthotropic elasticity and lower beam consisting of Core and Bottom Facesheet modeled based on isotropic and orthotropic elasticity respectively. Model based on 2D continuum shell.

Continuum linear rectangular plane-stress 4-noded shell element (CPS4) was utilized to model the skin and core regions of the sandwich composite. Suitable meshing and refinement has been done to reduce error and obtain closer result. Figure.4, shows the meshing of the model. Pre-crack consist of coarsen mesh and crack propagation region have finer mesh density.

**Comparison of experimental work and Numerical simulation**: For Sandwich DCB Specimen with initial crack length, a= 50mm under Mode I loading condition and loading rate of 1mm/min, from the figure 4 and figure 5 it can be inferred that there is a satisfactory agreement between experimental and computational results. It is clear from the figure 5 that Compliance is increasing with the increasing crack length plus load delivery capacity reduce subsequent the deflection reaches the critical value as it can be seen in figure 4, this is due to the reduction of stiffness of the specimen with the propagation of crack.
CONCLUSION

The use of Modified Compliance Calibration as a data reduction method used for calculating fracture energy which is used as an input for experimental analysis. In experimental work load vs displacement and compliance vs crack length curves were produced. And acquired values were compared with numerical values obtained using ABAQUS/Standard solver. A good agreement on the values between the numerical and experimental curves validates the given methodology.

REFERENCES


Karthik B, Kiran Kumar TVU, EMI developed test methodologies for short duration noises, Indian Journal of Science and Technology, 6 (5), 2013, 4615-4619.


