Failure of Notched Laminates Under Out-of-Plane Bending, Phase V Update

2012 JAMS Meeting

J.P. Parmigiani, B.K. Bay, T.M. Froemming, and T.J. Wright
Oregon State University, Corvallis, OR, 97331

The design of aircraft structures made of composite materials is heavily influenced by damage tolerance requirements. Predicting failure in notched laminates has been the subject of numerous studies. In general, these investigations have focused on the response of laminates to in-plane tension, compression or shear. In spite of the fact that out-of-plane bending, twisting, or shear can be an important load situation, very little research has been devoted to this topic. To address this need, research has been conducted at Oregon State University through the AMTAS/JAMS FAA Center of Excellence program since 2007/2008 in a series of one-year phases. The overall objective of this research is to develop analysis techniques, typically with experimental validation, that are useful for the design of composite aircraft structures subjected to general out-of-plane loading. Tasks for the 2011/2012 phase consist of finite element modeling (using Abaqus) with experimental validation of out-of-plane shear (mode III) loading of notched laminate panels, an evaluation of the Abaqus plug-in Helius:MCT; and modeling of all-ninety-degree and all-zero-degree laminate panels under out-of-plane bending. The focus of this document is to provide an update on progress in the 2011/2012 phase and conclude the 2010/2011 phase.

I. Introduction

The motivation for this work is to develop analysis techniques that are useful in the design of composite aircraft structures under out-of-plane loading, specifically bending and shear. These loading cases have not been extensively covered in the literature but are relevant to aircraft. For example, out-of-plane bending and shear occur in the fuselage skin near reinforcing members such as frames and stringers. The objectives are to identify key failure modes and to evaluate the capabilities of current modeling techniques to predict and model damage. The approach is to use the commercial finite element program Abaqus to model progressive damage development and delamination and to predict behavior and key failure mechanisms. These models will be guided and validated using experimentation on composite panels representative of the materials used on aircraft.

Staffing for this project consists of personnel from Oregon State University (OSU), the Federal Aviation Authority (FAA), Boeing, and NSE Composites. From OSU, Professor John Parmigiani with graduate student Thomas Wright conduct the finite element work and Professor Brian Bay with graduate student Tyler Froemming conduct the experimental work. FAA technical monitors are Curt Davies and Lynn Pham, with Larry Ilciewicz also involved. From Boeing, Gerald Mabson provides guidance. Tom Walker from NSE Composites is also involved.

The overall project, part of the AMTAS/JAMS FAA Center of Excellence program, consists of a series of one-year phases beginning in the 2007-08 academic year. Phase I (2007-08) consisted of out-of-plane bending...
experiments with notched composite panels, shown in Fig. 1, and corresponding analytical analysis of notch stress-concentration and progressive damage modeling in Abaqus. Phase II (2008-09) consisted of Abaqus modeling including the capability of ply-buckling delamination at selected locations where delamination was observed in the experiments. Phase III (2009-10) consisted of adding additional ply-delamination capability to the Abaqus models. Phase IV (2010-11) included a number of tasks. The study of ply delamination for out-of-plane bending was completed, initiating-versus-propagating toughness values were evaluated, the feasibility of Abaqus explicit and XFEM for use if the out-of-plane bending models was evaluated, and a sensitivity study of Hashin damage parameters was conducted. Phase V (2011-12), the current phase, consists for three tasks. First, the study of mode III fracture (out-of-plane shear) of composite panels similar to those used in the out-of-plane bending work, however containing an edge notch instead of a center notch. This study consists of Abaqus modeling with experimental validation. Second, the evaluation of Helius:MCT for use in modeling the composite damage which occurs in the out-of-plane loading of this work. Helius:MCT is an Abaqus plug-in created by Firehole Composites, Inc. Third, the special cases of all-ninety-degree and all-zero-degree plies will be studied. It is expected that the study of such simple layups will provide insight into the fundamental behavior of composite panels under out-of-plane loading.

The focus of this paper will be on three topics. First, to provide background and recent results for the experimental portion of the out-of-plane shear study, second to give an update on the evaluation of Helius:MCT, and third to provide some summary comments on the evaluation of Abaqus Explicit.

II. Out-of-Plane Shear Study

The goals of the out-of-plane shear study are to (i) create an experimental set-up to load edge-notched laminate panels to failure via out-of-plane shear, (ii) measure the resulting load-displacement relationships and surface strains, and (iii) accurately model the loading in Abaqus. This section will discuss work-to-date on the first and second goals.

In creating the experimental set-up, the first step taken was a review of the literature to identify relevant prior work. This review revealed that there is currently little literature on the loading of plate materials to induce mode III fracture, and none specifically for laminate composite panels. Due to the lack of experimentation done on these materials, no coherent methodology has emerged as the standard. The following is a review of techniques that have been used to induce Mode III fracture in wide plates.

One of the earlier examples of mode III plate fracture is from an experiment performed by Jones and Subramonian\textsuperscript{4} in 1983. Out-of-plane shear was applied to a single edge-notched plate, using geometry similar to a compact tension specimen as shown in Fig. 2. The specimens were attached to the loading rails through connecting holes. Opposite ends of the loading rails were displaced in opposite directions. This particular testing procedure was performed on plexiglass, 2024-T3 aluminum, and wood. Birefringence was used to monitor the strain fields within the plexiglass specimen while load and displacement were recorded for all specimens.
Between 2007 and 2009, three papers from the University of South Carolina described a new method for tearing ductile metal (aluminum and steel) plates\(^5\). Pre-cracked specimens (again similar to compact tension specimens) were constrained to remove all rotational degrees of freedom from the gripping surfaces as shown in Fig. 3(a). The gripped surfaces were then displaced in opposite directions in order to apply both in-plane tensile loads and out-of-plane shear loads as shown in Fig. 3(b). Using a particular setting of their test setup, they were able to apply pure Mode III. In these studies, full field strain measurements were performed using Digital Image Correlation (DIC).

Frühmann et al.\(^8\) loaded wood specimens in a manner similar to that of Sutton et al.\(^5\) and Yan et al.\(^6,7\). While the wood specimens used here were less plate-like, the loading setup is relevant. To alleviate bending at the point of gripping, the grip points were rotated about an axis perpendicular to the plane of the fracture surface.

Departing from the general geometry of the compact tension specimen, Sutton et al.\(^9\) applied a mixed mode I / mode III loading to fatigue pre-cracked aluminum plates. Specimens were gripped on the ends of the plate (not on or opposite the cracked edge) and both a torsional displacement (twist) and tension was applied, as shown in Fig. 4. DIC was used to capture full field strain measurements of the specimen.
After reviewing the literature, the experimental set-up selected for this study consists of gripping the laminate panel along the plate-edge containing the edge notch. A grip on one side of the notch displaces in one out-of-plane direction, a grip on the other side displaces in the opposite out-of-plane direction. The grips are designed to allow free rotation along the edge of the specimen. A representation of this loading is shown in Fig. 5(a). A solid model of a conceptual test-machine fixture for applying this loading is shown in Fig. 5(b).

Specimen geometry was carefully selected to allow effective measurement of failure strain and loading. Preliminary modeling using Abaqus was used to predict regions of composite material damage. These results were used to specify panel planar dimensions and notch length to prevent panel edges or fixture grips from constraining the propagation of damage. Also, specimen geometry was specified to allow accurate digital image correlation (DIC) measurements of notch-tip strain fields. The specified specimen geometry is shown in Fig 6.
The testing plan consists of six lay-ups; three 20-ply lay-ups (10%, 30%, and 50% zero-degree plies) and three 40-ply lay-ups (10%, 30%, and 50% zero-degree plies). Three replicates of each lay-up will be tested. To-date one specimen of each lay-up has been tested. Results are given in Table 1. “Maximum load” is the largest applied load the panel was capable of supporting and was limited by damage accumulation emanating from the notch tip. “Work to Max” is the area under the load-displacement curve for the the loading of the specimen to maximum load. Both maximum load and work-to-maximum load appear to vary nearly linearly with percent-zero-degree plies, however additional replicates are needed to verify this trend.

Table 1: Current test results for out-of-plane shear study

<table>
<thead>
<tr>
<th>Layup</th>
<th>No. of Plies</th>
<th>% zero deg</th>
<th>Max Load (kN)</th>
<th>Work to Max (J)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>40</td>
<td>50</td>
<td>5.55</td>
<td>362.0</td>
</tr>
<tr>
<td>2</td>
<td>40</td>
<td>30</td>
<td>5.34</td>
<td>287.8</td>
</tr>
<tr>
<td>3</td>
<td>40</td>
<td>10</td>
<td>3.89</td>
<td>177.9</td>
</tr>
<tr>
<td>4</td>
<td>20</td>
<td>50</td>
<td>1.75</td>
<td>147.8</td>
</tr>
<tr>
<td>5</td>
<td>20</td>
<td>30</td>
<td>1.48</td>
<td>73.8</td>
</tr>
<tr>
<td>6</td>
<td>20</td>
<td>10</td>
<td>1.29</td>
<td>56.7</td>
</tr>
</tbody>
</table>
Surface strain fields were measured, using DIC, near the notch tip during all loading. A typical result, showing the expected notch-tip strain concentration, is shown in Fig. 7(a). A more unexpected result, an abrupt reduction in strain at the notch tip, is shown in Fig. 7(b). This variation in strain is thought to be evidence of sub-surface material damage. This, if verified by future results, is compelling because it occurs in the linear region of the load-displacement relationship and prior to any external evidence of material damage.

III. Helius: MCT Evaluation

Helius: MCT is an Abaqus plug-in created by Firehole Composites, Inc. It is marketed as superior to Abaqus built-in capabilities for progressive damage in composites and this is potentially relevant for use in this project. The evaluation plan is to use Helius and repeat portions of the out-of-plane bending analysis from earlier phases. Results using Helius will be compared to results obtained using Abaqus built-in capabilities in terms of run time, accuracy and ease of use. If the results are promising, Helius may be used for the out-of-plane shear & all-ninety-degree and all-zero-degree ply studies of Phase V.

IV. Abaqus Explicit

Exploration of the feasibility of ABAQUS Explicit as an alternative to ABAQUS Standard (i.e. implicit) was a task in Phase IV (2010-11). Prior to this study, Abaqus Standard had been used for all modeling in all phases. The hope was that Explicit would be faster and would significantly reduce the multi-day run times currently needed when using Abaqus Standard. Results of this work were presented at AMTAS 2011 Fall meeting. Questions and comments following the presentation raised some compelling points, so follow-up work was conducted in late 2011 and early 2012. This section will provide a brief recap of this task and the results of the additional work.

Explicit methods inherently include dynamic effects. If the total time-of-the-simulated-event is sufficiently long (deformation and motion sufficiently slow), kinetic energy is small and quasi-static events, such as those of this project, can be modeled. An advantage of explicit method is that it is unconditionally stable, convergence issues of ABAQUS Standard are gone. A disadvantage of Explicit is that the required time increment can be very small and model run times can thus become very long.

For the material properties of the specimens used in this project, if the actual conditions of the physical experiments were modeled using Explicit, run times would be extremely long (several months). However, methods exist to shorten Explicit run times and three were considered.

First, a method potentially relevant for this project is shortening the model time so it is much less than the actual physical experiment time. This can be acceptable if quasi-static conditions are still maintained in the shortened model. This requirement can be quantified by calculating and comparing internal vs. kinetic energy of the model. If kinetic energy is much less than internal energy, than quasi-static conditions can be assumed to exist. Implementing this method for this project and obtaining Explicit run time even close to implicit run times required reducing actual experimental run-times of minutes to model-simulation times of just a few seconds. The need for such a large deviation from the actual physical experiment in order to achieve run times using Explicit which are even close to those of Standad is not compelling and this method was not pursued further.

![Figure 7: Notch-tip strain fields measured with DIC](image-url)
A second method potentially relevant for this project is mass scaling. Mass scaling involves changing the material mass density of the model such that it is much greater than the actual material mass density of the actual physical specimen. The assumption is that if the phenomena is quasi-static then changing the material mass-density will not significantly change the calculated results. Pursuing this approach for the specimens of this study required increasing the material mass density of the model to 5200% of the actual material mass density in order to reach overall run times comparable to Abaqus Standard. Again, the need for such a large deviation from the actual physical experiment for no significant advantage in overall run time is not compelling and this method was not pursued further.

Finally, a third possibility is sub-modeling. This consists of running part of model in Explicit, part in Standard. This offers best of both worlds, however this approach can be problematic when changes in model material stiffness occur between regions modeled using Explicit and regions modeled using Standard. Given the extensive damage that occurs during loading, and the resulting large changes in stiffness, this approach was not pursued.

V. Conclusions

The experimental portion of the out-of-plane shear study has been carefully researched and designed and specimen testing is well underway. Strain fields obtained from digital image correlation together with load-displacement results will provide comprehensive data for validation of future Abaqus modeling. Furthermore, strain field results suggest internal material damage in a region of loading previously thought to be purely elastic. The evaluation of Helius:MCT is underway and may provide an attractive alternative to Abaqus built-in capabilities for the modeling required for other tasks of this phase. Finally, a wrap-up of an investigation of Abaqus Explicit shows it to not be an attractive alternative to Abaqus Standard.

References