Evaluation of Abaqus for Simulating Quasi-Static Mode III Shear of Edge Notched Carbon Fiber Reinforced Polymer Panels

By Imran Hyder
Candidate for Master of Science in Mechanical Engineering

Abstract

The ability to engineer stiffness and strength in any desired direction make composites an ideal material candidate for various applications when compared to their traditional isotropic counterparts. In spite of this, the ability to model the material response in composites has yet to be fully explored. Composite research largely focuses on in-plane conditions and research involving modeling Mode III (out-of-plane shear) is limited. Mode III occurs when adjacent sections of a plate are displaced in opposite out-of-plane directions, thus causing through-thickness tearing. Mode III can potentially lead to catastrophic failure for composite designs with inadequate out-of-plane transverse properties exposed to excessive out-of-plane loads. This can be countered by overdesigning a structure, but at the cost of sacrificing efficiency. Hence it is necessary to have models to appropriately capture material behavior during loading for design and analytical purposes. Commercial finite element (FE) packages are available for simulating various loading conditions, but there has not been an assessment of their applicability for composites enduring Mode III. This study aimed to evaluate the performance of a commercial finite element package, Abaqus, for modeling Mode III loading of edge notched Carbon Fiber Reinforced Polymer panels using previously conducted experiments as a metric. Six ply layups were considered and were composed of either 20 or 40 unidirectional plies. For each thickness, 10%, 30%, and 50% zero-degree panels were studied. Panels also included 45, -45, and 90 degree plies. This investigation was divided into two studies: Evaluation of finite element analysis (FEA) prior to visible damage initiation and evaluation of FEA for progressive failure simulation. The first study utilized strain fields obtained from Digital Image Correlation (DIC) and load versus displacement profiles retrieved from experiments to evaluate elastic based FEA conducted with Abaqus/Standard. Abaqus/Standard was able to simulate strain fields roughly within 30% with the exception of small regions near the notch tip and predict the maximum loads with a percent difference of 20%. The 50% zero-degree panels was an exception where in which large discrepancies occurred between experiments and FEA. The second study involved assessing Abaqus/Standard, Abaqus/Standard with the add-in Helius:MCT, and Abaqus/Explicit for simulating progressive failure analyses. Experimentally obtained load versus displacement profiles, damage paths, and maximum loads were used as a metric to evaluate the solvers. It was found that the solvers were not able to predict the complete damage paths. However, Abaqus/Standard and Abaqus/Explicit were able to predict the maximum loads with a percent difference of 20%. Helius:MCT experienced convergence failures using default settings. Although accuracy and predictive capabilities were limited, the solvers were able to provide reasonable approximations for the material behavior.

Monday, September 22, 2014
1:00 PM, Kelly 1007