ENABLING TECHNOLOGIES FOR PROGRESSIVE FAILURE ANALYSIS OF COMPOSITE STRUCTURES

Don Robbins, Distinguished Research Engineer

Autodesk, Inc., Laramie, Wyoming

KEYWORDS

Fiber-reinforced composites, progressive damage, global failure, material softening, geometric nonlinearity

ABSTRACT

Fiber-reinforced, laminated composite structures can exhibit significant stress concentrations that manifest on three different geometric scales. At the heterogeneous material level, there are stress concentrations that exist between the relatively stiff reinforcing fibers and the relatively compliant matrix material. These fiber/matrix stress concentrations cause damage accumulation in the fiber and matrix constituent materials and/or damage to the interface between the constituent materials, thus causing stiffness reduction of the composite material. At the ply level, there are stress concentrations at the interface between dissimilar composite plies that can cause damage to the composite ply or the bond between two adjacent plies. Finally, at the structural level, the laminate often contains holes, cut-outs or other geometric anomalies that cause in-plane stress concentrations that exist throughout the thickness of the entire laminate stack.

Due to the inherent multiscale heterogeneity and stress concentration of composite laminates, it is commonplace for laminated composite structures to begin accumulating damage at loads that are far below the ultimate load of the structure. In fact, in many cases, the laminated composite structure accumulates enough damage prior to ultimate load that the stress distribution at ultimate load is quite different from the stress distribution at load initiation. Consequently, the prediction of a laminated composite structure's ultimate load requires a progressive failure analysis (or PFA) where local damage evolution and associated local stiffness reduction are explicitly accounted for in the analysis.

Unfortunately, the material softening associated with damage evolution and material failure poses two serious difficulties for implicit finite element codes. First, the evolution of damage and material failure and the associated material softening makes it very difficult for the finite element code to determine a converged equilibrium solution for any given load increment in the specified load history (the greater the

damage evolution in a given load increment, the greater the difficulty of determining the converged solution). Second, if the analysis accounts for geometric nonlinearity, then those elements that have failed and softened will likely exhibit excessive distortion levels that cannot be tolerated by the FE code. If this excessive element distortion can't be resolved by reducing the load increment size (most often the case), then the analysis is arrested prematurely. These difficulties cause most analysts to utilize explicit FE codes to circumvent the requirement of obtaining a converged solution for each load increment.

Autodesk has effectively addressed these convergence and element distortion problems, thereby permitting PFA of composite structures to be performed very efficiently and reliably using implicit FE simulation. The Autodesk Helius PFA code is a type of User-Defined Material Subroutine (UMAT) that uses the proprietary Intelligent Discrete Softening (IDS) Method to eliminate convergence difficulties associated with material softening behaviour. Helius PFA can be used to enhance the performance of most commercial FE codes, and provides a multiscale material model for composite materials that independently predicts material damage of the matrix and fiber constituent materials using constituent average stress states. In addition, the excessive element distortion problem associated with damaged/softened elements in geometric nonlinear analyses has been eliminated by implementing an IDS-based element deletion feature in the Autodesk Nastran FE code. These enabling simulation technologies will be demonstrated for practical composite laminates that are progressively loaded to global failure.

Figures 1 through 4 and Table 1 demonstrate results that were obtained for an open-hole quasi-isotropic laminate under tension loading. In these simulations, Helius PFA was used to provide the multiscale progressive damage material response for the Abaqus Standard code and the Autodesk Nastran code. As seen in the figures, the complete load histories were successfully simulated despite the large amounts of ply damage and delamination that occurred during the loading.

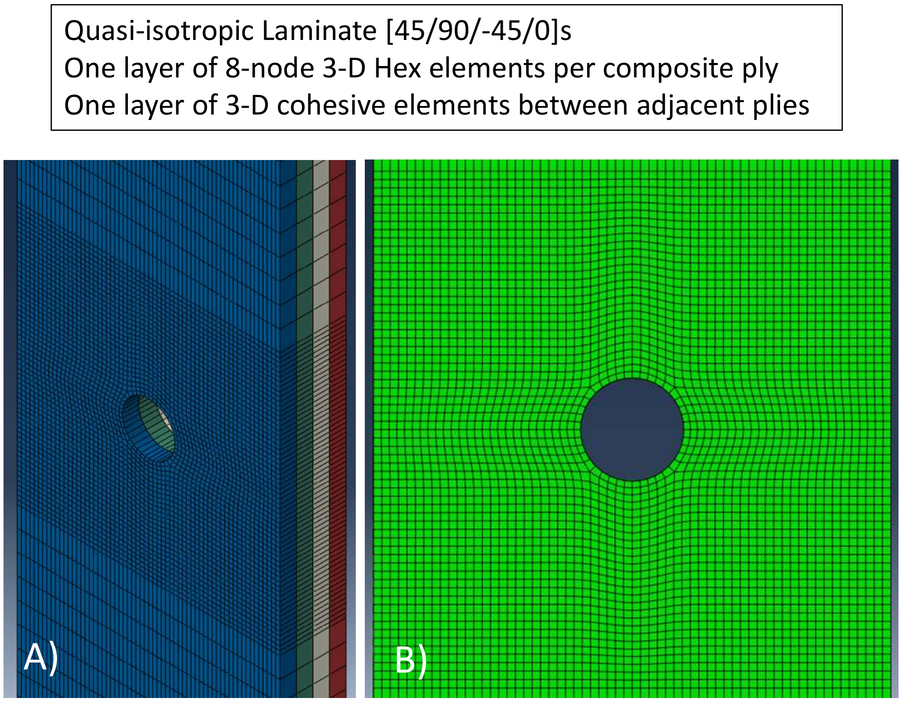

Figure 1. Finite element mesh of a laminated composite open-hole tension coupon. A) Oblique view showing the discrete layer mesh (one element layer per composite ply), B) close-up planform view of the refined mesh around the circular hole.

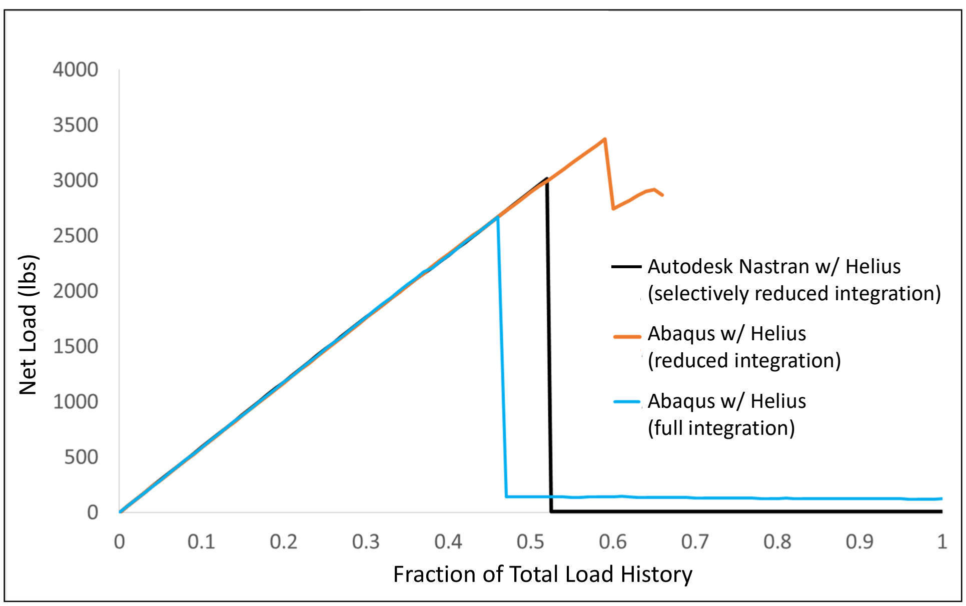

Figure 2. Comparison of the predicted response of the Open Hole Coupon using three different code combinations.

Table 1. Comparison of the total number of equilibrium iterations performed during the Open Hole Coupon simulation by three different code combinations

	Total Equilibrium Iterations at Max Load	Total Equilibrium Iterations after Global Failure Cascade
Autodesk Nastran w/Helius (selectively reduced integ.)	378	556
Abaqus w/ Helius (reduced integ.)	2285	3750
Abaqus w/ Helius (full integ.)	598	817

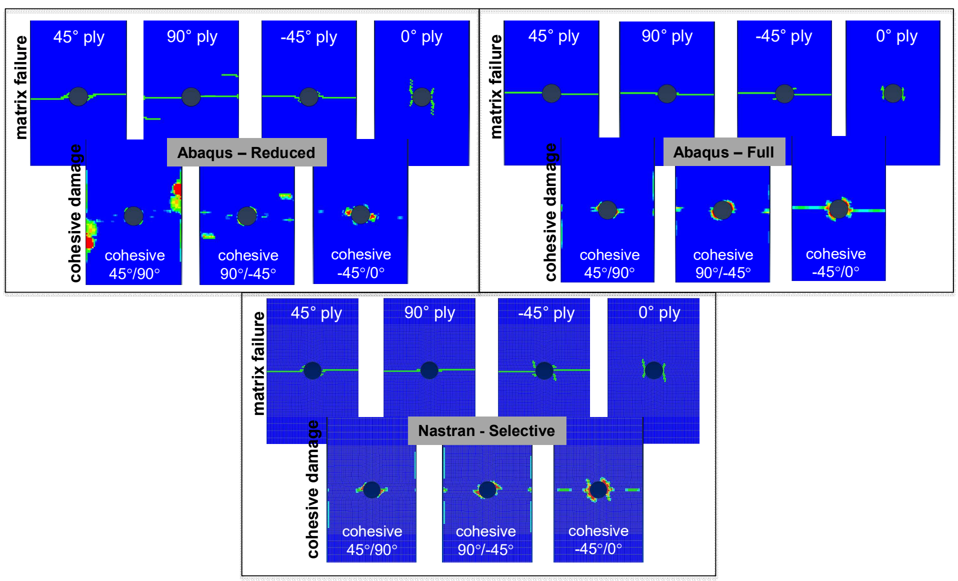

Figure 3. Comparison of the predicted intra-laminar and inter-laminar damage distribution at the critically damaged state (at max load) for the three different code combinations.

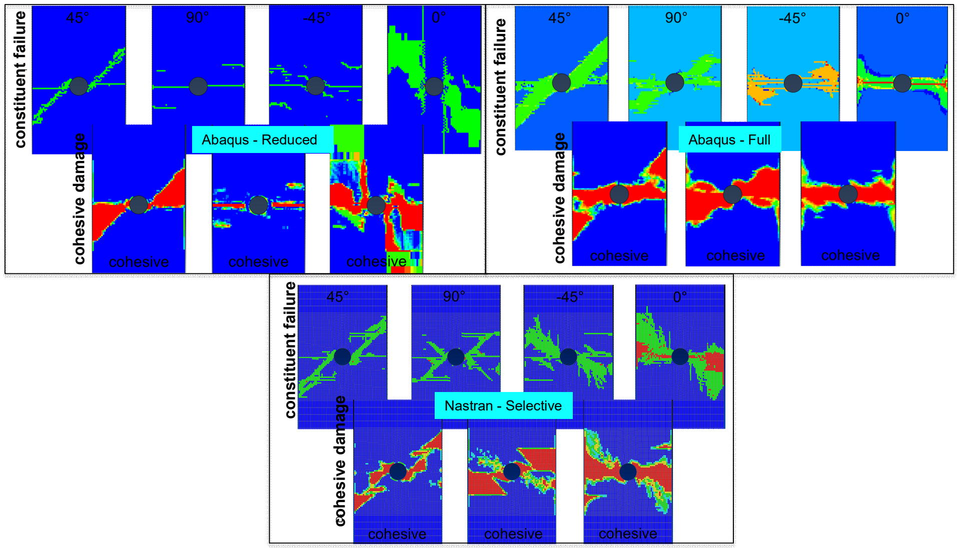

Figure 4. Comparison of the predicted intra-laminar and inter-laminar damage distribution immediately after the global failure cascade (large load drop) for the three different code combinations.