Failure of Notched Laminates Under Out-of-Plane Bending
Phase VII

Fall 2014 Meeting
Mitchell Daniels, Levi Suryan,
& John P. Parmigiani,
Oregon State University
Motivation and Key Issues

Develop analysis techniques useful in design of composite aircraft structures under out-of-plane loading (bending and shear)

Objective

Determine failure modes and evaluate capabilities of current models to predict failure

Approach

• Experiments: Out-of-plane shear (mode 3 fracture)
• Modeling: Progressive damage development and delamination (Abaqus) under Mode 3 fracture
Failure of Notched Laminates Under Out-of-Plane Bending. Phase VII

- **Principal Investigators & Researchers**
  - John Parmigiani (PI); OSU faculty
  - M. Daniels, L. Suryan; OSU grad students

- **FAA Technical Monitor**
  - Curt Davies
  - Lynn Pham

- **Other FAA Personnel Involved**
  - Larry Ilcewicz

- **Industry Participation**
  - Gerry Mabson, Boeing (technical advisor)
  - Tom Walker, NSE Composites (technical advisor)
Project Overview

Phase I (2007-08)
- Out-of-plane bending experiments w/composite plates
- Abaqus modeling with progressive damage

Phase II (2008-09)
- Abaqus modeling with buckling delamination added
- Sensitivity study of (generic) material property values

Phase III (2009-10)
- Abaqus modeling w/ more delamination interfaces
Project Overview

Phase IV (2010-11)
- Begin out-of-plane shear experiments
- Continue out-of-plane bending modeling

Phase V (2011-12)
- Complete out-of-plane shear experiments
- Begin out-of-plane shear modeling

Phase VI (2012-13)
- Continue out-of-plane shear modeling
  - Abaqus Standard
  - Abaqus Explicit
  - Helius MCT
Project Overview

Phase VII (2013-14)

- Comprehensive report on Phase VI work for Boeing
- Evaluation of solid vs. shell elements in modeling
- Improvement to Abaqus Explicit models
- Explore damage softening parameters in SCA (Simulation Composites Analysis, formerly Helius: MCT)
- Sensitivity study of material properties to explore possibility of inaccuracies in material properties
Today’s Topics

• Brief review of computational Model
• Evaluation of Solid Versus Shell Elements
• Improvements to Abaqus Explicit
• Exploration of SCA damage parameters
• Material Parameters Sensitivity Study
• Summary of Mode III Analysis
• Conclusions
Today’s Topics

• Brief review of Computational Model
• Evaluation of Solid Versus Shell Elements
• Improvements to Abaqus Explicit
• Material Parameters Sensitivity Study
• Future Work
• Summary of Mode III Analysis
• Conclusions
Computational Model

- Uses Hashin failure criteria
- Quasi-static analysis and non-linear geometry turned on
- Panel: Continuum shell, reduced integration elements (SC8R)
- Mesh defined around notch tip
- Grips: Continuum, 3-D, 8 node, reduced integration element (C3D8R)
- Boundary conditions implemented by grips
Today’s Topics

- Brief review of Computational Model
- Evaluation of Solid Versus Shell Elements
- Improvements to Abaqus Explicit
- Exploration of SCA damage parameters
- Material Parameters Sensitivity Study
- Summary of Mode III Analysis
- Conclusions
Solid Vs. Shell Elements

- **Task:** Explore the effect of using solid elements versus shell elements in Models
- **Approach:** Compared elements with Simulation Composites Analysis (SCA) (formerly Helius:MCT) because solid elements not available using Hashin damage criteria with Abaqus Standard or Explicit
- **Results:** Solid elements are not recommended
  - Fine mesh requirements to simulate laminates
  - Becomes computationally prohibitive before solution convergence
  - Limits accuracy of simulations
Today’s Topics

• Brief review of Computational Model
• Evaluation of Solid Versus Shell Elements
• Improvements to Abaqus Explicit
• Exploration of SCA damage parameters
• Material Parameters Sensitivity Study
• Summary of Mode III Analysis
• Conclusions
Improvements for Abaqus
Explicit-Increased Mesh Density

- **Task:** Reduce or eliminate noise from explicit solver by increasing mesh density
- **Approach:** Increased mesh density from 20 elements around the notch tip to 32 and 64 elements around notch tip
- **Results:**
  - Increasing mesh density not effective for reducing noise
  - Noise can lead to inaccuracies and requires filtering
  - Max load results discussed later in presentation
Improvements for Abaqus
Explicit- Mass Scaling

- **Task:** Reduce the time of Abaqus explicit solver analysis
- **Approach:** Mass Scaling
  - Conditionally stable solver yields long run times due to small time steps
  - Mass scaling specifies minimum time step
  - Scales mass to reach min time step
  - Value: 5x10-6 s
- **Results:** Drastically reduces solver time from ~230 hours to ~30 hours
- Quasi static condition considered valid: oscillations occur after maximum load
- Max load comparison on next slide
Improvements for Abaqus
Explicit- Max Load Results

- Compared mass scaling and increasing mesh density's effect on max load
- 40 ply layups or 64 element max load not considered due to computation time restraints and large file sizes making extracting max loads impractical
- Increased mesh density does not produce more accurate results
  - Hashin Damage mesh dependent in FEA solvers
  - Element characteristic length \((L_c)\) decreases strain to failure \((\delta_f)\) by
  \[
  \delta_f \downarrow = \frac{2G_{l,c}}{XL_{c}^\uparrow}
  \]
- Mass scaling with 20 elements show best agreement with experiments:
  - Mass scaling parameter of simulation, limits ability as a predictive tool
  - Mass scaling strongly suggested to reduce computation times

<table>
<thead>
<tr>
<th>Model</th>
<th>10 % 0°-20 Plies</th>
<th>30 % 0°-20 Plies</th>
<th>50 % 0°-20 Plies</th>
</tr>
</thead>
<tbody>
<tr>
<td>20 elements-Mass Scaling</td>
<td>10.9%</td>
<td>-1%</td>
<td>-19%</td>
</tr>
<tr>
<td>20 elements-No Mass Scaling</td>
<td>21.70%</td>
<td>23.80%</td>
<td>56%</td>
</tr>
<tr>
<td>32 elements-Mass Scaling</td>
<td>1.30%</td>
<td>-15.80%</td>
<td>-27.90%</td>
</tr>
</tbody>
</table>
Today’s Topics

• Brief review of Computational Model
• Evaluation of Solid Versus Shell Elements
• Improvements to Abaqus Explicit
• Exploration of SCA damage parameters
• Material Parameters Sensitivity Study
• Summary of Mode III Analysis
• Conclusions
Abaqus with SCA- Overview

• Autodesk Simulation Composite Analysis (SCA): Plug-in that applies damage criteria to fiber and matrix, formerly Helius:MCT

\[
\sigma^c = \phi_f \sigma^f + \phi_m \sigma^m
\]

Average stress of composite, fiber, and matrix respectively

• Task: Improve selection of instant stiffness degradation parameters for composite panels in Mode III shear
• Approach: Evaluate FEA results for convergence, maximum load, and damage trajectories
• Results: following slides
Abaqus with SCA- Instant Stiffness Degradation Parameters

- Instant degradation parameters $D_m$ and $D_f$ reduce matrix and fiber stiffnesses when damage criteria are met
- Parameters are user defined
- 8 combinations of $D_m$ and $D_f$ were evaluated, starting with those recommended by Autodesk
  - Shotgun approach for selecting combinations taught value of designed experiments
- Combinations effected convergence and load versus displacement results

<table>
<thead>
<tr>
<th>Run</th>
<th>Instant Degradation Values</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Matrix ($D_m$)</td>
<td>Fiber ($D_f$)</td>
</tr>
<tr>
<td>1</td>
<td>1.00E-01</td>
<td>1.00E-06</td>
</tr>
<tr>
<td>2</td>
<td>1.00E-01</td>
<td>1.00E-04</td>
</tr>
<tr>
<td>3</td>
<td>1.00E-02</td>
<td>1.00E-05</td>
</tr>
<tr>
<td>4</td>
<td>1.00E-03</td>
<td>1.00E-06</td>
</tr>
<tr>
<td>5</td>
<td>1.00E-02</td>
<td>1.00E-04</td>
</tr>
<tr>
<td>6</td>
<td>1.00E-03</td>
<td>1.00E-03</td>
</tr>
<tr>
<td>7</td>
<td>1.00E-01</td>
<td>1.00E-04</td>
</tr>
<tr>
<td>8</td>
<td>2.00E-01</td>
<td>1.00E-06</td>
</tr>
</tbody>
</table>
Abaqus with SCA- Convergence Results

- Convergence issues limited available softening parameter combinations (33 of 48 runs failed)
- No stiffness degradation parameter in study yielded convergence for all layups, meaning convergence is limiting accuracy of results

<table>
<thead>
<tr>
<th>Layup</th>
<th>Instant Stiffness Degradation Combination</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1</td>
</tr>
<tr>
<td>40P-50%</td>
<td>F</td>
</tr>
<tr>
<td>40P-30%</td>
<td>F</td>
</tr>
<tr>
<td>40P-10%</td>
<td>F</td>
</tr>
<tr>
<td>20P-50%</td>
<td>F</td>
</tr>
<tr>
<td>20P-30%</td>
<td>F</td>
</tr>
<tr>
<td>20P-10%</td>
<td>F</td>
</tr>
<tr>
<td>Success Rate</td>
<td>0.0</td>
</tr>
</tbody>
</table>

S: Maximum load reached (success)
F: No max load before model divergence (failure)
Abaqus with SCA- Maximum Load Results

- Maximum load of FEA model for each stacking sequence was compared to average maximum load of corresponding experimental group.
- Superior stiffness degradation parameter combination selected by maximum load accuracy.

<table>
<thead>
<tr>
<th>Layup</th>
<th>ISD Combo</th>
<th>Experimental Max Load (lb)</th>
<th>FEA Max Load (lb)</th>
<th>Percent Deviation</th>
</tr>
</thead>
<tbody>
<tr>
<td>40P-50%</td>
<td>6</td>
<td>1230</td>
<td>1115</td>
<td>9.8%</td>
</tr>
<tr>
<td>40P-30%</td>
<td>5</td>
<td>1283</td>
<td>1238</td>
<td>3.6%</td>
</tr>
<tr>
<td>40P-10%</td>
<td>4</td>
<td>921.9</td>
<td>1115</td>
<td>18.9%</td>
</tr>
<tr>
<td>20P-50%</td>
<td>3</td>
<td>403.6</td>
<td>296.5</td>
<td>30.6%</td>
</tr>
<tr>
<td>20P-30%</td>
<td>6</td>
<td>344.2</td>
<td>346.9</td>
<td>0.8%</td>
</tr>
<tr>
<td>20P-10%</td>
<td>3</td>
<td>282.9</td>
<td>299.8</td>
<td>5.8%</td>
</tr>
</tbody>
</table>

Average Percent Deviation: 11.6%
Abaqus with SCA- Damage Trajectory Results

- Damage trajectories indicate quality of FE analysis technique
- Two of three recurring damage trajectories successfully modeled:
  - Splitting damage (successful), from notch tip in perpendicular direction
  - Diagonal damage (successful), from notch tip in direction of surface ply fibers
  - Self-similar damage (unsuccessful), from notch tip in the direction of the notch
- Self-similar damage involved extensive delamination; not modeled by single element through thickness
Today’s Topics

• Brief review of Computational Model
• Evaluation of Solid Versus Shell Elements
• Improvements to Abaqus Explicit
• Exploration of SCA damage parameters
• Material Parameters Sensitivity Study
• Summary of Mode III Analysis
• Conclusions
Sensitivity Study Tasks

- Determine which Hashin damage parameters (listed below in table) have significant effect on max load during finite element simulations
- Using results from study determine if possible errors in nominal parameter values could explain errors in simulations and which values of parameters could yield experimental results

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>XT</td>
<td>tensile strength in 1-dir.</td>
</tr>
<tr>
<td>XC</td>
<td>compressive strength in 1-dir.</td>
</tr>
<tr>
<td>YT</td>
<td>tensile strength in 2-dir.</td>
</tr>
<tr>
<td>YC</td>
<td>compressive strength in 2-dir.</td>
</tr>
<tr>
<td>SL</td>
<td>longitudinal (in-plane) shear strength</td>
</tr>
<tr>
<td>SC</td>
<td>transverse shear strength</td>
</tr>
<tr>
<td>Gft</td>
<td>Energy required to fully damage a ply using fiber tension only</td>
</tr>
<tr>
<td>Gfc</td>
<td>Energy required to fully damage a ply using fiber compression only</td>
</tr>
<tr>
<td>Gmt</td>
<td>Energy required to fully damage a ply using matrix tension only</td>
</tr>
<tr>
<td>Gmc</td>
<td>Energy required to fully damage a ply using matrix compression only</td>
</tr>
</tbody>
</table>
Sensitivity Study - Approach

- 10 factor, 2 level partial factorial
  - 1/16th fractional factorial: $2^{10-6}$
  - Levels: ±20% from nominal values
  - Resolution IV: No first order interactions confounded with 2nd-order interactions
- Only 20 ply panels considered due to the large number of runs
- Results from implicit and explicit solvers considered
- Used normal probability plots used to determine significance
- Secondary factorial experiment used if confounding factors significant
- Regression modeling used to determine which parameter values yield experimental results
Sensitivity Study—Significance Results

• XT and XC (longitudinal tension and compression strength) shown to be the most significant parameters for mode III loading case
• Other parameters showed significance for certain layups but have small effects when compared to XT and XC
• Solver types yields different results
  • Explicit and implicit solvers different techniques yielding slightly different results
  • Solver tools like mass scaling can change results

<table>
<thead>
<tr>
<th>Layup</th>
<th>Analysis Method</th>
<th>Significant Parameters in Order of Significance</th>
</tr>
</thead>
<tbody>
<tr>
<td>20P/10%</td>
<td>Abaqus Standard</td>
<td>XT, XC, SL</td>
</tr>
<tr>
<td>20P/30%</td>
<td>Abaqus Standard</td>
<td>XT, XC, XT-XC Interaction</td>
</tr>
<tr>
<td>20P/50%</td>
<td>Abaqus Standard</td>
<td>XT, XC</td>
</tr>
<tr>
<td>20P/10%</td>
<td>Abaqus Explicit</td>
<td>XT, XC, SL, XT-XC Interaction, GFC, GFT</td>
</tr>
<tr>
<td>20P/30%</td>
<td>Abaqus Explicit</td>
<td>XT, XC, XT-XC Interaction, SL</td>
</tr>
<tr>
<td>20P/50%</td>
<td>Abaqus Explicit</td>
<td>XT, XC</td>
</tr>
</tbody>
</table>
Sensitivity Study - Regression Approach

- XT and XC main factors considered for regression due to significance
- Fitted model to sensitivity study data considering all ten factors and all second order interaction
- Optimization in statistics program for experimental max load with all factors considered, could not reach agreement between layups
- Varied XT and XC
  - compared results to experimental average maximum load
  - No agreement between different layups
  - Ran first ply failure analysis to calculate the strength of laminates
  - Used new strengths in regression to check for agreement
Sensitivity Study - Regression Results

- Figure shows variation of XT and XC with the lines representing the constant maximum load according to experimental results.
- Material parameter errors are not able to explain simulation errors:
  - Different layups do not agree on specified values of XT and XC.
  - Could not predict simulation results using nominal values.
  - Only accurately predicts FEA response for parameter values near levels in study (±20% Nom).
  - Suggests nonlinearity and other factors in FEA not captured by regression model.
Today’s Topics

- Brief review of Computational Model
- Evaluation of Solid Versus Shell Elements
- Improvements to Abaqus Explicit
- Exploration of SCA damage parameters
- Material Parameters Sensitivity Study
- Summary of Mode III Analysis
- Conclusions
### Summary of Mode III Analyses from Phase VI and VII

- Summary of all Mode III analyses from Phase VII (Explicit and SCA) and Phase VI (Standard)
- Hourglass stiffness scaling used in Standard to prevent large element deformations
- Explicit shows best agreement with experimental average maximum loads after mass scaling
- SCA and Standard had convergence difficulties
- Adding cohesive elements improved standard accuracy, but could not achieve convergence for explicit and SCA

<table>
<thead>
<tr>
<th>Panel Layup</th>
<th>Standard</th>
<th>Explicit</th>
<th>Standard with SCA</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1 Element Thick- HGSS</td>
<td>2 Element Thick- HGSS</td>
<td>3 Element Thick- HGSS</td>
</tr>
<tr>
<td>20-10% 0°</td>
<td>31%</td>
<td>15%</td>
<td>20%</td>
</tr>
<tr>
<td>20-30% 0°</td>
<td>DNC</td>
<td>19%</td>
<td>21%</td>
</tr>
<tr>
<td>20-50% 0°</td>
<td>-17%</td>
<td>-16%</td>
<td>-23%</td>
</tr>
<tr>
<td>40-10% 0°</td>
<td>40%</td>
<td>22%</td>
<td>18%</td>
</tr>
<tr>
<td>40-30% 0°</td>
<td>DNC</td>
<td>3%</td>
<td>-7%</td>
</tr>
<tr>
<td>40-50% 0°</td>
<td>-3%</td>
<td>-5%</td>
<td>-4%</td>
</tr>
<tr>
<td>Average</td>
<td>23%</td>
<td>13%</td>
<td>16%</td>
</tr>
</tbody>
</table>

HGSS: Hourglass Stiffness Scaling. DNC: Did Not Converge.
Today’s Topics

- Brief review of Computational Model
- Evaluation of Solid Versus Shell Elements
- Improvements to Abaqus Explicit
- Exploration of SCA damage parameters
- Material Parameters Sensitivity Study
- Summary of Mode III Analysis
- Conclusions
Conclusions from Phase VII

- Shell elements yield better results than solid elements
- Sensitivity study of material parameters showed longitudinal tension and compression strengths are most significant parameters
  - XT and XC shown to be most significant parameters for mode III
  - Regression models failed to predict desired property values
  - Material parameters considered can not predict discrepancy between experiment and simulation alone
- SCA is accurate and efficient, but has severe convergence issues
- Explicit yields best agreement with experiments
  - Refining mesh density does not achieve better agreement with experiments
  - Mass scaling drastically reduces computation time
  - Most recommended of current tools
  - Limited as predictive tool due to mass scaling being element of solver
Questions
Out-of-Plane Shear: Summary of Experimental Results

- Maximum applied load (failure load)

<table>
<thead>
<tr>
<th>Layup (#plies / % zero degree)</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>MEAN</th>
</tr>
</thead>
<tbody>
<tr>
<td>40/50%</td>
<td>5.552</td>
<td>5.345</td>
<td>5.122</td>
<td>6.103</td>
<td>5.395</td>
<td>5.321</td>
<td>5.473</td>
</tr>
<tr>
<td>40/30%</td>
<td>5.342</td>
<td>5.363</td>
<td>6.061</td>
<td>5.616</td>
<td>6.176</td>
<td>5.690</td>
<td>5.708</td>
</tr>
<tr>
<td>20/50%</td>
<td>1.751</td>
<td>1.859</td>
<td>1.929</td>
<td>1.691</td>
<td>1.740</td>
<td>1.801</td>
<td>1.795</td>
</tr>
<tr>
<td>20/30%</td>
<td>1.484</td>
<td>1.541</td>
<td>1.541</td>
<td>1.456</td>
<td>1.527</td>
<td>1.638</td>
<td>1.531</td>
</tr>
<tr>
<td>20/10%</td>
<td>1.290</td>
<td>1.215</td>
<td>1.258</td>
<td>1.254</td>
<td>1.198</td>
<td>1.336</td>
<td>1.259</td>
</tr>
</tbody>
</table>
Why Continuum Shell Elements vs. Solid Elements

• Solid elements can be laminated but max order of variation of the displacement is quadratic
  • Hence strain variation is at most linear
  • Insufficient to model variation of strain through thickness of laminate
• Potential Solution: stack solid elements at one element per lamina
  • In-plane dimensions can not be > 10x thickness
  • Requires a really fine mesh
• Alternate Solution: Use continuum shell elements
  • Does not have the same problems as a solid element
  • Can have multiple plies through the thickness
  • Also can be stacked for using with grips and delamination
• Laminate stacking sequence was constructed using Composite Layup in Abaqus – define material prop’ per ply
Scaling Hourglass Stiffness

- Default hourglass stiffness was scaled to prevent severe element deformation
- Pure stiffness approach was recommended for quasi-static analysis
- Three user defined scaling factors

<table>
<thead>
<tr>
<th>Factor</th>
<th>Description</th>
<th>Typical Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hour glass stiffness scaling factor for displacement degree of freedom</td>
<td>0.2 - 3.0</td>
<td></td>
</tr>
<tr>
<td>Hour glass stiffness scaling factor for rotational degree of freedom</td>
<td>0.2 - 3.0</td>
<td></td>
</tr>
<tr>
<td>Hour glass stiffness scaling factor for out-of-plane displacement degree of freedom</td>
<td>0.2 - 3.0</td>
<td></td>
</tr>
</tbody>
</table>
• Scaling $s^w$ caused solutions to fail prematurely
• Only scaled $s^s$ and $s^r$
• After scaling to the limits of the recommended value, not all stacking sequences converged
• After drastically increasing factors, convergence was achieved for most models
• Factors were selected based on a convergence study
Damage Path Model

1 Layer – No SSF

1 Layer – with SSF

2 Layer – VCCT

3 Layer – VCCT
## Results Table: Explicit and Helius:MCT

### Energy (Given)

<table>
<thead>
<tr>
<th>Combo</th>
<th>MCT (N)</th>
<th>Exp. (N)</th>
<th>% Diff</th>
<th>Converge</th>
</tr>
</thead>
<tbody>
<tr>
<td>F</td>
<td>2330.83</td>
<td>1188</td>
<td>65.0</td>
<td>Y</td>
</tr>
<tr>
<td>N</td>
<td>2377.34</td>
<td>1689</td>
<td>33.9</td>
<td>Y</td>
</tr>
<tr>
<td>P</td>
<td>2598.69</td>
<td>1472</td>
<td>55.4</td>
<td>Y</td>
</tr>
<tr>
<td>AN</td>
<td>9785.4</td>
<td>5111</td>
<td>62.8</td>
<td>N</td>
</tr>
<tr>
<td>FP</td>
<td>9278.33</td>
<td>4005</td>
<td>79.4</td>
<td>Y</td>
</tr>
<tr>
<td>AR</td>
<td>7394.08</td>
<td>5899</td>
<td>22.5</td>
<td>Y</td>
</tr>
</tbody>
</table>

### Instant Degradation (Given)

<table>
<thead>
<tr>
<th>Combo</th>
<th>MCT (N)</th>
<th>Exp. (N)</th>
<th>% Diff</th>
<th>Converge</th>
</tr>
</thead>
<tbody>
<tr>
<td>F</td>
<td>1296.9</td>
<td>1188</td>
<td>8.8</td>
<td>N</td>
</tr>
<tr>
<td>N</td>
<td>1184.99</td>
<td>1689</td>
<td>35.1</td>
<td>N</td>
</tr>
<tr>
<td>P</td>
<td>1388.43</td>
<td>1472</td>
<td>5.8</td>
<td>Y</td>
</tr>
<tr>
<td>AN</td>
<td>4989.86</td>
<td>5111</td>
<td>2.4</td>
<td>N</td>
</tr>
<tr>
<td>FP</td>
<td>5104.25</td>
<td>4005</td>
<td>24.1</td>
<td>N</td>
</tr>
<tr>
<td>AR</td>
<td>6528.27</td>
<td>5899</td>
<td>10.1</td>
<td>N</td>
</tr>
</tbody>
</table>

### Cohesive Zones (Given - Instant)

#### Instant Degradation (Default)

<table>
<thead>
<tr>
<th>Combo</th>
<th>MCT (N)</th>
<th>Exp. (N)</th>
<th>% Diff</th>
<th>Converge</th>
</tr>
</thead>
<tbody>
<tr>
<td>F</td>
<td>713</td>
<td>1188</td>
<td>50.0</td>
<td>N</td>
</tr>
<tr>
<td>N</td>
<td>996</td>
<td>1689</td>
<td>51.6</td>
<td>N</td>
</tr>
<tr>
<td>P</td>
<td>838</td>
<td>1472</td>
<td>54.8</td>
<td>N</td>
</tr>
</tbody>
</table>

### Abaqus/Explicit: Filter

<table>
<thead>
<tr>
<th>Combo</th>
<th>Explicit (N)</th>
<th>Exp. (N)</th>
<th>% Diff</th>
<th>Converge</th>
</tr>
</thead>
<tbody>
<tr>
<td>F</td>
<td>1291</td>
<td>1188</td>
<td>8.3</td>
<td>Y</td>
</tr>
<tr>
<td>N</td>
<td>928</td>
<td>1689</td>
<td>58.1</td>
<td>Y</td>
</tr>
<tr>
<td>P</td>
<td>1158</td>
<td>1472</td>
<td>23.8</td>
<td>Y</td>
</tr>
</tbody>
</table>

**Legend:** Y = Yes, N=No, Exp. = Experimental Values, MCT= Helius:MCT results

F = 10% zeros, 20 ply ; P = 30% zeros, 20 ply ; N=50% zeros, 20 ply ; FP = 10% zeros, 40 ply ; AR = 30% zeros, 40 ply ; AN = 50 % zeros, 40 ply
Helius: MCT Results – Boeing Parameters (Energy Degradation)

Fig 1. F Configuration

Fig 2. P Configuration

Fig 3. N Configuration

Fig 4. FP Configuration

Fig 5. AR Configuration

Fig 6. AN Configuration
Helius: MCT Results – Boeing Parameters (Instant Degradation)

Fig 1. F Configuration
Fig 2. P Configuration
Fig 3. N Configuration
Fig 4. FP Configuration
Fig 5. AR Configuration
Fig 6. AN Configuration
Helius: MCT Results – Default Parameters

Fig 1. F Configuration
Fig 2. P Configuration
Fig 3. N Configuration
Fig 4. FP Configuration
Fig 5. AR Configuration
Fig 6. AN Configuration
Cohesive Zones in Helius:MCT

- Cohesive zone runs do not converge
- Deformation in cohesive zone areas can be observed but it is difficult to discern if this deformation is delamination
Filtering Results

Application of filters with varying cut-off frequencies for F-configuration

*10% zeros, 20 plies

10 Hz cut-off filter compared to experimental results for F-configuration
Filtering Results

*50% zeros, 20 plies

Application of filters with varying cut-off frequencies for N-configuration

10 Hz cut-off filter compared to experimental results for N-configuration
More Multi-Layer Results

Multi-Layer Models: 30% zeros, 20 plies configuration

4 layer with varying degradation values: 30% zeros, 20 plies configuration
Abaqus/Explicit Solver Runtime

- Analyses are extremely long
  - the Explicit solver is only conditionally stable and requires an extremely small time step. Critical time step must considered:

\[
\Delta t \leq \frac{2}{\omega_{max}} \leq \Delta t_{cr}
\]

- Need to maintain a Quasi-static state: \( E \downarrow K \leq 0.1 \ E \downarrow I \)
<table>
<thead>
<tr>
<th>Layers (ct.)</th>
<th>Run Time (hr)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>354</td>
</tr>
<tr>
<td>4</td>
<td>672</td>
</tr>
<tr>
<td>8</td>
<td>585</td>
</tr>
</tbody>
</table>

**Table 1.** Run Times for Quasi-static models.
Evaluation of Abaqus Standard: Computational Model

- Solver basics:
  - Uses Newton-Raphson Technique to iterate to a converge solution for each time increment
  - Static equilibrium:
    \[
    [K][U] = [R]
    \]

- Uses Hashin failure criteria
- Quasi-static analysis and Non-linear geometry turned on
- Panel: Continuum shell, reduced integration elements (SC8R)
- Grips: Continuum, 3-D, 8 node, reduced integration element (C3D8R)
- Boundary conditions implemented by grips
- Mesh Selection – 20 elements around notch tip, based on a linear elastic convergence study
Evaluation of Abaqus Standard: Computational Model and Convergence Based Parameters

• Viscous Regularization Scheme (used in standard/explicit) helps with convergence
  § Viscosity coefficients for fiber compression, fiber tension, matrix tension, and matrix compression
  § Must be small with respect to the time increment,
  § Convergence trend at:

• Hourglass stiffness scaled to prevent severe element deformation (Standard only)
  § Three hourglass scaling factors for displacement degree of freedom rotational degree of freedom, and out-of-plane displacement degree of freedom
  § Scaling to recommended values didn’t yield converged solutions for some stacking sequences in Standard
  § Needed to drastically increase factors, most models and stacking sequences converged
  § Converging trend at
Evaluation of Abaqus Standard: Single Element Layer and 3 Element Layer Delamination with VCCT Results

- All models effectively captured linear region
- FE models have stiffness factors scaled high
- FE material response is similar, but 3-layer VCCT models capture experiment behavior better
- Not all models revealed a clear max
Evaluation of Abaqus Standard: Summary of Results

• Benefit: Standard predicts max load within 20% of experiments

• Major Challenges:
  § Implicit analysis fails to converge without excessive stiffness factors
  § After the use of excessive stiffness factors, some models still fail to converge

• Suggestion:
  § Accuracy can be improved by changing VCCT interfaces – but no rational for it
  § Modify convergence parameters

<table>
<thead>
<tr>
<th>Ply</th>
<th>% Zero</th>
<th>1 Element Layer - No Scaled Stiffness Factors (SSF)</th>
<th>1 Element Layer - with SSF</th>
<th>2 Element Layer VCT with SSF - Interface from Experiments</th>
<th>2 Element Layer VCT with SSF - Interface before 90° plies</th>
<th>3 Element Layer VCT with SSF - Interface from Experiments</th>
<th>3 Element Layer VCT with SSF - Interface before 90° plies</th>
</tr>
</thead>
<tbody>
<tr>
<td>20 PLY</td>
<td>10%</td>
<td>31%</td>
<td>15%</td>
<td>25%</td>
<td>20%</td>
<td>20%</td>
<td>-1%</td>
</tr>
<tr>
<td></td>
<td>30%</td>
<td>DNC</td>
<td>19%</td>
<td>21%</td>
<td>21%</td>
<td>DNC</td>
<td>18%</td>
</tr>
<tr>
<td></td>
<td>50%</td>
<td>-16%</td>
<td>-16%</td>
<td>-23%</td>
<td>-23%</td>
<td>DNC</td>
<td>DNC</td>
</tr>
<tr>
<td>40 PLY</td>
<td>10%</td>
<td>45%</td>
<td>22%</td>
<td>DNC</td>
<td>18%</td>
<td>DNC</td>
<td>17%</td>
</tr>
<tr>
<td></td>
<td>30%</td>
<td>DNC</td>
<td>3%</td>
<td>-2%</td>
<td>-7%</td>
<td>42%</td>
<td>22%</td>
</tr>
<tr>
<td></td>
<td>50%</td>
<td>-3%</td>
<td>-5%</td>
<td>-4%</td>
<td>-4%</td>
<td>DNC</td>
<td>DNC</td>
</tr>
</tbody>
</table>
Today’s Topics

- Experimental results: Out-of-plane shear
- Evaluation of Abaqus Standard results
- Evaluation of Helius MCT results
- Evaluation of Abaqus Explicit results
Abaqus/Standard with Helius:MCT

- Helius:MCT was utilized for its recognized convergence capabilities and fast solver algorithm for out-of-plane bending
- Solver basics: analyzes the composite based on its constituents as well as a whole:

\[ \sigma^c = \phi_f \sigma^f + \phi_m \sigma^m \]

Average stress of composite, fiber, and matrix respectively

- Method:
  - Adapt input file to include Helius:MCT solver
  - Use default parameters, instant degradation parameters, energy degradation parameters
  - Apply cohesive zones (CZ)
Abaqus/Standard with Helius:MCT

- Representative of all trials and configurations, including with CZ
- Benefits: fast solver: runtime < 10hrs
- Major challenges:
  - Convergence
  - Accuracy in certain situations
- Suggestions
  - Shows promise if convergence occurs, try different energy parameters or degradation values
  - Possible changes may occur in the future to better the solver: Autodesk ownership
Today’s Topics

- Experimental results: Out-of-plane shear
- Evaluation of Abaqus Standard results
- Evaluation of Helius MCT results
- Evaluation of Abaqus Explicit results
Abaqus/Explicit Analysis

• Why use explicit: implementation of element deletion and better convergence
• Solver basics:
  ▪ Analysis used an explicit, dynamic solver:
    \[ [M][\ddot{U}] + [C][\dot{U}] + [K][U] = [R] \]
  ▪ Central difference method for enhanced convergence: hope to overcome the issues present in Abaqus/Standard
  ▪ Hashin damage criteria
• Determination of quasi static state
  ▪ Varied total time until a majority of analysis was quasi static: kinetic energy < 10% internal energy
  ▪ Total time considered (seconds): 0.25, 0.50, 1.00, 2.00, 4.00, 6.00, 8.00
  ▪ 8 seconds chosen as total time increment
Abaqus/Explicit
Analysis: One Layer Results

- Benefits: convergence in most cases
- Major challenges:
  - Extreme amounts of noise
  - Extremely long runtime
- Suggestions
  - Filtering the data
  - Implementing more layers
• Dilemma: element deletion is not occurring
• Solution: create more layers so that the deletion criteria is met more readily
• Methods:
  ▪ Create 2, 4, and 8 layer models.
  ▪ Varying degradation coefficient: 1.0, 0.9, 0.8, 0.7
  ▪ Implement VCCT
Abaqus/Explicit Analysis: Implementation of Multiple Layers

- **Benefits**: eliminate distorted elements
- **Major Challenges**:
  - Convergence
  - Extremely long run times
- **Suggestion**: not much can be gained overall from implementing multiple layers
Conclusions for Explicit Analysis

- Analyses are extremely long.
- Analyses are inherently prone to noise during extreme deformations or accelerations. How do we appropriately filter this noise?
- Convergence is not guaranteed and element deletion may not always be something we can take advantage of.
Future Work - Explicit VCCT

- Revisited VCCT with explicit
- Showed better results for damage path (45° damage)
- Showed better stress strain curve shape
- Over predicts maximum load
- Longer solve time: ~90 hours
- Difficult convergence
- Future work: refine placement of cohesive layers
Future Work - Energy Sensitivity

- Previous work by Ludeman showed significance in energy parameters for quasi isotropic layups with a $2^{10-6}$
- First attempts at a thick quasi-isotropic layup yielded no energy significance
- All 90s showed significance of matrix energies
- Attempting to fully recreate Ludeman’s runs with $2^{10-4}$ fractional factorial
Edge-notched CF panels displaced to maximum load
- 20 and 40 lamina thick panels with three lay ups: 10%, 30%, & 50% 0° plies
- Metrics: Applied displacement and applied load

Notch size: 4” long (101.6 mm)  
End radius: 0.25” (6.35 mm)  
Panel size: 18” (457 mm)  
by 10” (254 mm)
Improvements for Abaqus
Explicit-Mesh Study

- Hashin criteria mesh dependent in Abaqus
- Mesh convergence study to determine most accurate mesh
- Ran simulations in linear elastic region to compare stress values of different meshes
- Selected coarsest mesh showing converged stress value
- Plot shows percent deviation from previous mesh
• Normal probability plot of 50% 0° explicit results shown to the right
• Deviation from linear line indicates significance
• Further deviations suggest more significant effects
• In example: $X_t$ shows more significance than $X_c$