Composites Seminar 2012 - Seattle

Sean Harvey
March 16, 2012
<table>
<thead>
<tr>
<th>Time</th>
<th>Agenda Items</th>
</tr>
</thead>
<tbody>
<tr>
<td>09:30 - 10:00</td>
<td>Registration</td>
</tr>
<tr>
<td>10:00 - 10:30</td>
<td>Incorporating Basic FEM Concepts in a Introductory Composites Analysis Course</td>
</tr>
<tr>
<td>10:30 - 11:15</td>
<td>Composites Failure, Fracture and Damage Mechanical Simulations</td>
</tr>
<tr>
<td>11:15 - 12:00</td>
<td>Composite Workflow Developments</td>
</tr>
<tr>
<td>12:00 - 13:00</td>
<td>Networking Lunch</td>
</tr>
<tr>
<td>13:00 - 14:15</td>
<td>Demonstrations:</td>
</tr>
<tr>
<td></td>
<td>• Workflow Developments</td>
</tr>
<tr>
<td></td>
<td>• Fracture/Damage</td>
</tr>
<tr>
<td></td>
<td>• Composite Radome Design</td>
</tr>
<tr>
<td>14:15 - 14:30</td>
<td>Closing Remarks and Future Work</td>
</tr>
</tbody>
</table>
Introduction

• Presenter: Sean Harvey
• Senior Technical Services Engineer
• Part of Aerospace and Defense Team
• Background in Composites, Non Linearity and Biomedical
• Composites is a very broad topic, and impossible to cover all aspects in this seminar
• Today’s focus is on laminated composites and the enhanced workflow at ANSYS R14
• Delamination, damage, and fracture will also be discussed.
Our Strategy
Simulation-Driven Product Development

Democratize Simulation Process Automation
Enable Best Practices
Focus on Engineering

Complete Systems
Simulated Environments
Multiphysics

Fluids Dynamics
Structural Mechanics
Explicit Dynamics
Low-Frequency Electromagnetics
High-Frequency Electromagnetics
Thermal Mechanics
Acoustics

Advanced Technologies
Virtual Prototyping

Process Compression
Democratize Simulation
Process Automation
Enable Best Practices
Focus on Engineering

Dynamic CAE Collaboration
Span Organizational and Geographic Silos
Share Engineering Insights
Better Decisions Faster
Composites in our World

Application Areas

- Aerospace
- Wind Energy
- Sports & Recreation
- Motorsport
- Construction
- Automotive
- Marine
- Defense
- Biomedical
- ... and more
Composite Progressive Damage and Fracture

What happens beyond first ply failure?

• Progressive Damage

How to model delamination and debonding?

• Cohesive Zone Model (CZM)
• Virtual Crack Closure Technique (VCCT)
What happens beyond first ply failure?

- Plotting failure criteria allow us to estimate the strength of the structure based on the first ply failure using one or multiple failure criteria (Puck, Hashin, Max Stress, etc.)

Progressive damage
- Prior to current release, progressive damage was not available
- Progressive damage generally refers to degradation of stiffness
- ANSYS has the new MPDG option along with the Microplane model (not covered here)
Composite Progressive Damage and Fracture

What happens beyond first ply failure?

Progressive Damage MPDG Model

• The damage initiation and propagation in fiber-reinforced composites can be simulated with a nonlinear solution process.
• The new capability allows you to estimate ultimate composite strength under complex stress states.
• The difference between first ply failure and ultimate is very dependent on the layup and loading. For some layups, the difference can be just a few percent, while with others, it can be much greater.

• Damage Initiation Criteria - This defines the criteria type for determining the onset of the material damage under loading. The available failure criteria are as follows:
  • Maximum strain, Maximum stress, Puck, Hashin, LaRc03, LaRc04, User-defined
• Damage Evolution Law - This defines the way material degrades following the initiation of damage
• Desire to correlate ANSYS R14 Damage capability TB,DMGI & TB,DMGE WITH published work of Camanho and Matthews to showcase the new ANSYS R14 capabilities to the Aerospace/Defense industry as well as other industries performing simulations and strength predictions of composite structures.

• Paper Title:

A Progressive Damage Model for Mechanically Fastened Joints in Composite Laminates

P. P. Camanho and F. L. Matthews

Centre for Composite Materials
Imperial College of Science, Technology and Medicine
Prince Consort Road
London SW7 2BY, UK

(Paper accepted for publication in the Journal of Composite Materials, 2000)

Journal of Composite Materials December 1999 33: 2248-2280,
Progressive Damage

• In this work, the Authors have experimental Load/Disp curves of composite coupons of Pin Loaded Net Tension, Shear-Out, and Bearing failure modes.

• Authors are using Abaqus Explicit with reduced integration and hourglass control.

• Hashin Failure on [0/90/-45/45]2s laminate made of T300/914.

• Simulation model is solid with one element per ply.

• Net Tension Failure experimental and Abaqus results shown below.
Progressive Damage

- ANSYS model built in Workbench Mechanical.
- Composite Setup in ACP.
- Failure and damage commands added manually to .dat file and solved.
- Layered composite SHELL181 with pin as rigid contact.
- Damage Evolution stiffness reduction per the cited paper

```
! Specify Hashin strength terms
TB,FCLI,my_mat_num,1,20,1
TBTEMP,71.6
TBDATA,1,1439.,-1318.,98.,-125.,98.,-125.
TBDATA,7,79.,79.,79.

! Specify damage initiation Hashin
TB,DMGI,my_mat_num,1,4,FCRT
TBTEMP,71.6
TBDATA,1,.93,.86,.8,.6

! Specify damage evolution Hashin
TB,DMGE,my_mat_num,1,4,MPDG
TBTEMP,71.6
TBDATA,1,.93,.86,.8,.6
```
Progressive Damage

- Specimen held ALL DOF at left edge, and at pin.
- Specimen displaced 1.2mm on right edge.
- Coupon dimension specified in this case to generate net tension failure, with some local bearing failure.
- Pin Contact is Normal Lagrange
ANSYS Results – 0 deg ply Sigma X

Normal Stress
Type: Normal Stress(X Axis) - Top/Bottom - Layer 1
Unit: MPa
Solution Coordinate System
Time: 2e-002
3/12/2012 9:14 AM

-3311.2 Max
-2359.3
1795.2
1039.2
281.8
-475.31
-1212.3
-1980.2
-2747.6
-3504.9 Min
ANSYS Results – Damage Animation

```
ELEMENT SOLUTION
STEP=1
SUB =2
TIME=.04
PDMGSTAT (NOAVG)
DMX =.03
```

progressive_failure_pin_loaded_Shell_ACP_4--Static Structural (B6)
ANSYS Shell Results Load/Disp Comparison

ANSYS R14 Beta

Experiment & Abaqus/Explicit
## ANSYS Load Comparison

### Abaqus

<table>
<thead>
<tr>
<th></th>
<th>Experimental</th>
<th>Experimental</th>
<th>FE</th>
<th>Error(^{(1)}) (%)</th>
<th>Error(^{(2)}) (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Shear</strong></td>
<td>6547</td>
<td>6547</td>
<td>6136</td>
<td>6.2</td>
<td>6.2</td>
</tr>
<tr>
<td><strong>Tension</strong></td>
<td>8020</td>
<td>7649</td>
<td>7284</td>
<td>9.2</td>
<td>4.7</td>
</tr>
<tr>
<td><strong>Bearing</strong></td>
<td>9803</td>
<td>8116</td>
<td>8524</td>
<td>13.0</td>
<td>-5.0</td>
</tr>
</tbody>
</table>

\(^{(1)}\) Maximum load sustained by the joint; \(^{(2)}\) first load drop-off

Table 4 - Comparison between experimental and predicted strengths

<table>
<thead>
<tr>
<th></th>
<th>ANSYS R14 beta</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max Load (N)</td>
<td>Error (%)</td>
</tr>
<tr>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>8468</td>
<td>5.6</td>
</tr>
<tr>
<td>--</td>
<td>--</td>
</tr>
</tbody>
</table>
Composite Progressive Damage and Fracture

How to model delamination and debonding?

Cohesive Zone Model - CZM is typically used to simulate:
- Interface delamination of composite structures
- Separation of adhesive joints
- Fracture process in a homogenous medium

- This approach introduces failure mechanisms by using the hardening-softening relationships between the separations and incorporating the corresponding forces across the interface

- An interface delamination and failure simulation is performed by first separating the model into two components or groups of elements. A cohesive zone is then defined between these two groups. ANSYS offers two options to model the interface:
  - Interface elements with CZM model
  - Bonded Contact elements & CZM model
Composite Progressive Damage and Fracture

How to model delamination and debonding?

Cohesive Zone Models - CZM

- Exponential
- Bilinear -
  - Consideration of different contribution of Shear component from Normal component
  - Consideration of irreversibility of CZM. Unloading behavior properly simulated (damage included).
- Support mode I, mode II/III, and mixed mode fracture

Damage = 0

Damage = 1
Composite Progressive Damage and Fracture

**CZM Examples**

- Two bonded strips
- Load Deflection Curve
Bilinear CZM Law

Objectives:

• Consideration of different contribution of Shear Component from Normal component
• Consideration of irreversibility of CZM
• Support mode I, mode II/III, and mixed mode fracture
• Usability of CZM for both interface elements and contact elements debonding.
Bilinear Cohesive Zone Modeling using Contact

• The bilinear cohesive zone material is based on the model proposed by Alfano and Crisfield
• Area Under the curve is the Normal Critical Fracture Energy $G_{cn}$ (units Energy/Area)
• Mode II/III Failure follows the same
• Debonding is complete when this equation is satisfied $\frac{G_n}{G_{cn}} + \frac{G_t}{G_{ct}} = 1$

• Options to specify
  **CBDD** -- Bilinear material behavior with linear softening characterized by maximum traction and maximum separation (valid for contact elements only).
  **CBDE** -- Bilinear material behavior with linear softening characterized by maximum traction and critical energy release rate (valid for contact elements only).

• User can specify normal, shear, or both
Bilinear Cohesive Zone Modeling Example – Skin -Stringer
Bilinear Cohesive Zone Modeling Example – T Joint

Delamination of a Stiffened Composite Panel under Compressive Load
Bilinear CZM Law using Interface Elements

- A bilinear traction-separation law provides the material behavior with TB,CZM,,,,,BILI
- Based on the work of Alfano and Crisfield
- Five or six material constants required
- Damage is included
- User inserts interface elements using czmesh command which splits the mesh at the nodes and inserts the interface elements.
Postprocessing INTER20x

Output of INTER20x elements is straightforward:

• The normal and shear tractions can be output in /POST1 or /POST26 using the SS,X and SS,XY (SS,XZ) values

• The normal and shear separations can be output in /POST1 or /POST26 using SD,X and SD,XY (SD, XZ) values

Output is evaluated with PLxSOL, PRxSOL, and ESOL commands.

Note example on left, showing normal stress of INTER20x elements. Failed elements show zero stress. Elements in compression (-13.3) are also present.
Composite Progressive Damage and Fracture

**How to model delamination and debonding?**

**Virtual Crack Closure Technique – VCCT:**
- Most commonly employed fracture mechanics approach to compute the debonding of composite structures.
- Based on the assumption that the energy needed to separate a surface is the same as the energy needed to close the same surface.
- VCCT was initially developed to calculate the energy-release rates of a cracked body and has been extended to crack propagation using interface elements.
- Crack growth occurs along a predefined crack path.
- The path is defined via interface elements.
- The analysis is quasi-static and does not account for transient effects.
- The material is linear elastic and can be isotropic, orthotropic or anisotropic.
How is \( G \) (strain energy release rate) computed for 3D VCCT?

For 3-D crack geometry with a low-order element mesh, the energy-release rate is defined as:

\[
\begin{align*}
G_I &= -\frac{1}{2\Delta A} R_Y \Delta v \\
G_{II} &= -\frac{1}{2\Delta A} R_X \Delta u \\
G_{III} &= -\frac{1}{2\Delta A} R_Z \Delta w
\end{align*}
\]

where:
- \( G_I, G_{II}, \) and \( G_{III} \) = mode I, II, and III energy-release rate, respectively
- \( \Delta u, \Delta v, \) and \( \Delta w \) = relative displacement between the top and bottom nodes of the crack face in local coordinates \( x, y, \) and \( z, \) respectively
- \( R_X, R_Y, \) and \( R_Z \) = reaction forces at the crack-tip node
- \( \Delta A \) = crack-extension area, as shown in the following figure:
Composite Progressive Damage and Fracture

How to model delamination and debonding?

Virtual Crack Closure Technique – VCCT:

- The crack can be located in a material or along the interface of the two materials.
- The fracture criteria are based on energy-release rates calculated using VCCT.
- Several fracture criteria are available are:
  - Linear Fracture Criterion
  - Bilinear Fracture Criterion
  - B-K Fracture Criterion
  - Modified B-K Fracture Criterion
  - Power Law Fracture Criterion
  - User-Defined Fracture Criterion

- Multiple cracks can be defined in an analysis
- VCCT doesn’t need a collapsed mesh as SIFS or JINT calculation
- Crack growth simulation is a nonlinear structural analysis
- Multiple cracks can grow simultaneously and independently
- Cracks can merge to a single crack when on the same interface
How to model delamination and debonding?

Virtual Crack Closure Technique – VCCT:
• Fracture occurs when the fracture criterion index is met, expressed as

\[ f \geq f_c \]

• In a crack growth simulation, a quantity of interest is the amount of crack extension. VCCT measures the crack extension based on the length of the interface elements that have opened, as expressed by the following equation and in the subsequent figure:

\[ \Delta a = \sum \Delta_i \]
VCCT-Based Crack Growth Simulation

VCCT Examples

End Notched Flexure (ENF) Specimen

Results from published paper

VCCT-Based Crack Growth Simulation

**VCCT Examples**

Mixed Mode Bending (MMB) Specimen

VCCT- Based Crack Growth Simulation

VCCT Examples

Mixed Mode Bending (MMB) Specimen

Composite Workflow Developments
Layered composites simulation requires:

- Orthotropic material stiffness terms (E’s, G’s, nu’s)
- Lamina (ply) thickness
- Fiber Orientation – draping
- Understanding of stacking sequence implications (ABD matrix coupling, bending and twisting during cure)
- Orthotropic strengths
- Understanding of numerous potential modes of failure and the various failure theories
Modeling Layered Composites Efficiently

Where does ANSYS stand with composite capabilities?

ANSYS is not new to modeling composites:

- Layered shell and solid elements for more than 2 decades
- Multi-material beams, with support for multiple layers and multiple section integrations
- Temp-Dep ortho material props with structural temps at each layer
- Composite PrepPost adds ease of use and results evaluation
- Integration into Workbench allows for rapid design studies
- VCCT and CZM, to characterize delamination and debonding for 7 years.
- Recent addition of progressive damage to characterize post first ply failure
Modeling Layered Composites Efficiently

What are some of the numerical approaches?

- Micro-Scale Approach – Detailed fiber matrix interactions

- Meso-Scale (Laminate Level) Approach – Layered Shells and Solids, extract displacements, modes, overall stiffness behavior, and detailed stresses and strains

- Macro-Scale Approach – Smeared or homogenous shell/solid, with no detailed ply stresses nor strains
Modeling Layered Composites Efficiently

What do we mean by Modeling Layered Composites Efficiently?

Highlight several focus area for this discussion:

- Allow for geometric design changes to model
- Make changes to layup, ply locations, and orientations
- Incorporate fluid/thermal loading directly from CFD
- Incorporate loading from external data or other simulations
- Solve structural simulations all off the same unified model
  - Static (linear & non-linear)
  - Buckling (linear & non-linear)
  - Transient Dynamic
  - Linear Dynamics
  - Explicit (Bird Strike, Drop Test, Crash and Impact, etc.)
- Ease of post-processing of composite results
Modeling Layered Composites Efficiently

How can recent advances in ANSYS facilitate composite simulations?

Let’s start with the workflow in ANSYS Workbench for incorporating composites in a typical design:

- Composite PrepPost integrated into Workbench workflow
- Upstream changes in design propagate through to the results
- In this example we might look at sensitivity of geometry size, shape or layups to deformation, natural frequencies, and failure margin
Composites PrepPost is used to define the layup. Here we see the part thickness, cross-sections, and fiber orientations.
### Modeling Layered Composites Efficiently

**Defining the layup with PrepPost**

<table>
<thead>
<tr>
<th>Material Data Fabrics/Stackups</th>
<th>Oriented Element Sets</th>
<th>Layup</th>
</tr>
</thead>
<tbody>
<tr>
<td>Layup Area</td>
<td>Layup Direction</td>
<td></td>
</tr>
<tr>
<td>Material Layer Thickness</td>
<td>Reference Direction</td>
<td></td>
</tr>
</tbody>
</table>
Modeling Layered Composites Efficiently

Make changes to layup, ply locations, orientations.

Here we change the width of the doubler shown in red.
Modeling Layered Composites Efficiently

Make parametric changes to ply dimensions
Modeling Layered Composites Efficiently

**Fiber orientation prediction and modification**

- Use internal draping calculations
- Interface with Vistagy’s FiberSim
- Modify the fiber orientation to something that is actually observed on the manufacturing floor.
- For this we use multiple rosettes (local coordinate systems) specifying the angle to match the measured at known locations, and let PrepPost interpolate in between.
Modeling Layered Composites Efficiently

Asymmetric laminate definitions

- Multiple overlapping oriented-element sets used to define asymmetric layups
Modeling Layered Composites Efficiently

Incorporate fluid/thermal loading directly from CFD (1-way and 2-way FSI)
Modeling Layered Composites Efficiently

Incorporate, transform, and validate loading from other simulations

Data Mapping

- Pressure
- Temperature
- Heat Transfer
- Thickness
- Displacements
Modeling Layered Composites Efficiently

Solve structural simulations all off the same unified model
Pressure loading, modal, and non-linear buckling example
Modeling Layered Composites Efficiently

*Solve structural simulations all off the same unified model*

Pressure loading, modal, and tip load non-linear buckling

- Change spar chord location, wing taper, rib position, add stiffener plies to skin
- Geometry, mesh, orientations, draping, properties, and results all updated

IRF - Distributed Pressure

2\textsuperscript{nd} Natural Freq = 174 Hz

IRF – 4” Tip Displacement

IRF - Distributed Pressure

2\textsuperscript{nd} Natural Freq = 241 Hz

IRF – 4” Tip Displacement
Perform what-if studies

Here we look at how the tip deflection and $2^{nd}$ natural frequency change with the $1^{st}$ rib spanwise location.

Create response surfaces for optimization
Modeling Layered Composites Efficiently

Ease of post-processing of composite results

Results Interrogation

• Worst case failure criteria over all layers shows

• $H_m(1)$ – Hashin matrix – Layer 1

• Pick element and see stress/strain and failure through the thickness
Modeling Layered Composites Efficiently

Ease of post-processing of composite results

Make Thru-Thickness Stress Plots and account for Interlaminar normal stresses with shells

- Below we look at the stress through the thickness in the bend
- Traditional shell approaches can not account for interlaminar normal stress (shown in blue curve)
- ANSYS Composite PrepPost can predict these base on the work of Roos, Kress & Ermanni
- Make more accurate assessments without the need for large 3D models
Modeling Layered Composites Efficiently

Ease of post-processing of composite results

Wrinkling

• Local buckling of a face sheet under compression
• Failure indicator available using shell modeling of sandwich

Core Failure

• Local failure of core in shear or tensile loading
• Failure indicator available using shell modeling of sandwich
Modeling Layered Composites Efficiently

Even more..

- 3D Models from shell models
- 3D Curves used to guide fiber orientation
- Flat pattern prediction
- Right click suppress parts or plies
- Map composite thickness from 3D CAD
- Build complex assemblies including contact
- Include mechanisms
- Customize layups using scripts and tables (Filament winding)
- 2-Way FSI with composites
Modeling Layered Composites Efficiently

Explicit dynamic model uses same composite setup

- Bird-Strike, Drop Test, Crash and Impact, etc.
- Simple setup for Eulerian mesh
Modeling Layered Composites Efficiently

Radome Case Study

Connections

Deformations from FSI

Failure Criteria
Modeling Layered Composites Efficiently

Radome Case Study – Bird Strike
Beam Modeling – Advanced Capabilities

- User Defined Sections
- Composite Sections
- Tapering
- Results (Disp, Strain, Stress) shown on Beam
Features for upcoming release

- PrepPost parameters available in Workbench
  - Layup definition as input parameters
  - Failure criteria as output parameters
  - Will allow for geometric and ply orientation design studies and even optimization
- Delamination/Interface layer defined as a named selection and automatically created in PrepPost when user picks solid model
- More integrated workflow for solid composites
- Fracture integrated into Mechanical
- XFEM at R15
- CDM based progressive damage
- Acoustic Simulation for Delamination Detection
Summary

- Composites simulations with ANSYS
  - Setup model one time, make changes to examine what if
  - Look at sensitivities of design changes
  - Investigate damage tolerance to flaws
  - Investigate post first ply failure
- Simple “Drag-and-Drop” Multiphysics
  - Easily incorporate thermal, fluids, and pressures
  - Additional capability to import 3rd party data
- Improve design fidelity by optimization
- One single framework – ANSYS Workbench
Thank You!