SYSTEMATIC APPROACH TO SIMULATING IMPACT FOR TRIAXIALLY BRAIDED COMPOSITES

A Dissertation
Presented to
The Graduate Faculty of The University of Akron

In Partial Fulfillment
of the Requirements for the Degree
Doctor of Philosophy

Brina J. Blinzler
May, 2012
SYSTEMATIC APPROACH TO SIMULATING IMPACT FOR TRIAXIALLY BRAIDED COMPOSITES

Brina J. Blinzler

Dissertation

Approved:

Advisor
Dr. Wieslaw K. Binienda

Committee Member
Dr. Robert K. Goldberg

Committee Member
Dr. Ernian Pan

Committee Member
Dr. Anil Patnaik

Committee Member
Dr. Xiaosheng Gao

Committee Member
Dr. Kevin Kreider

Accepted:

Department Chair
Dr. Wieslaw K. Binienda

Dean of the College
Dr. George K. Haritos

Dean of the Graduate School
Dr. George R. Newkome

Date
ABSTRACT

An analytical method has been developed to simulate the impact response of triaxially braided carbon fiber - epoxy composites, including the penetration velocity and impact damage patterns. Textile composites are being analyzed for their use in a variety of impact and containment situations. In particular, analysis methods that capture the architecturally dependent damage observed in impact tests in a computationally efficient manner are required. Before an impact simulation can be generated, all material input parameters must be found. The objective of this study is to develop an overall analysis method; creating a systematic method to determine the required input properties is just a portion of the work. In order to determine the input parameters, both the fiber and matrix properties and static tests conducted on standard coupon samples are used.

In the analytical model, the triaxial braid architecture is simulated by using four parallel shell elements, each of which is analyzed as a laminated composite. Specifically, non-shifted homogeneous subcells, and a new, more systematic method for impact property determination was developed. In order to determine the stiffness and strength properties required for the constitutive model used, a top-down approach for determining the strength properties is merged with a bottom-up approach for determining the stiffness properties. Combining these two types of calculations allows the material parameters to be established using a systematic approach. The model is correlated with the actual material properties using quasi-static coupon level tests for a number of representative triaxially braided composite materials.
The resulting model is then used to conduct a number of simulations. Specifically, the impact of a flat plate with a gelatin projectile, a flat plate with an aluminum projectile, and a composite tube pressurized with an elastomer are simulated and compared with experimental impact tests for a range of triaxially braided composite materials. These various simulations, using the systematic approach describe above, show promise in adequately capturing the macro level phenomenon seen during experimental testing.
DEDICATION

This dissertation is dedicated to my husband, and my parents for their support in pursuing my doctoral degree.
ACKNOWLEDGMENTS

I would first like to thank my advisor and mentor, Prof. Wieslaw Binienda, for the confidence, support, and guidance he has given me. I would also like to acknowledge Dr. Robert Goldberg for mentoring me during my time at NASA Glenn Research Center, and for the outstanding impact he has made on this research. I want to give a special thanks to Dr. Gary Roberts, Dr. Kelly Carney, and Dr. J. Michael Pereira from NASA Glenn Research Center for their time spent assisting me throughout my research.

I would also like to thank the other members of my Supervisory Committee, Prof. Ernian Pan, Prof. Anil Patnaik, Prof. Xiaosheng Gao, and Prof. Kevin Kreider for their support and advice. I am grateful to all my fellow graduate students and colleagues at NASA Glenn who have made my graduate study enjoyable. I would also like to thank The University of Akron for supporting me financially. Finally, I would like to thank NASA Glenn Research Center for supporting me through the Graduate Student Researchers Program during my doctoral studies through grant number NNXD9AL95H, technical monitor Dr. David Kankam.
TABLE OF CONTENTS

| LIST OF TABLES | xii |
| LIST OF FIGURES | xiv |

CHAPTER

I. MOTIVATION & INTRODUCTION ......................................................... 1

1.1 Motivation .................................................................................. 1

1.2 Introduction ................................................................................ 3

II. LITERATURE REVIEW ........................................................................ 10

2.1 Introduction ................................................................................ 10

2.1.1 Textile Polymer Composites .................................................. 10

2.1.2 Modeling Textile Polymer Composites .................................. 11

2.1.3 Multi-Scale Textile Composite Modeling Approaches .......... 15

2.1.4 Composite Failure under Impact Loading .............................. 18

2.1.5 Composite Energy Absorption in Impact Event .................... 20

2.2 Spalling ..................................................................................... 22

2.2.1 Epoxy Spalling ...................................................................... 23

2.3 Ongoing Research ...................................................................... 25
A.2.9  trans_t_LL.s............................................................................................ 228
A.2.10  trans_t_LLzero.m.................................................................................... 230
A.2.11  shear_LL.m............................................................................................ 232
A.2.12  writeInputCard.m.................................................................................. 234
A.2.13  writeInputExcel.m................................................................................ 236
A.2.14  input_calc.m.......................................................................................... 237
# LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1.2.1: Comparison of Composite Material Models within LS-DYNA</td>
<td>31</td>
</tr>
<tr>
<td>3.1.2.2: Comparison of Selected Material Models and Options</td>
<td>33</td>
</tr>
<tr>
<td>3.1.4.1: Input Parameters</td>
<td>42</td>
</tr>
<tr>
<td>3.1.6.4.1: Composite Strength Ratios</td>
<td>80</td>
</tr>
<tr>
<td>3.1.6.4.2: Composite Strength Ratios</td>
<td>80</td>
</tr>
<tr>
<td>4.1.3.1: Material property required for *Mat_58</td>
<td>91</td>
</tr>
<tr>
<td>4.1.4.1.1: Constituent Properties</td>
<td>94</td>
</tr>
<tr>
<td>4.1.4.1.2: Global Stress Values used in Calculations</td>
<td>95</td>
</tr>
<tr>
<td>4.1.4.5.1: Material property values used for the quasi-static coupon simulations</td>
<td>107</td>
</tr>
<tr>
<td>5.1.1: Mean, Standard Deviation, and Percentage Results</td>
<td>113</td>
</tr>
<tr>
<td>5.1.2: Interpolated Values at Strain Increments</td>
<td>114</td>
</tr>
<tr>
<td>5.1.3: Statistics for T700/PR520</td>
<td>115</td>
</tr>
<tr>
<td>5.1.4: Statistics for T700/E862</td>
<td>120</td>
</tr>
<tr>
<td>5.1.5: Statistics for T700/5208</td>
<td>124</td>
</tr>
<tr>
<td>5.1.6: Inputs for Homogenization Algorithm</td>
<td>128</td>
</tr>
<tr>
<td>5.1.7: PR520/T700 Notched Test Data and Statistics</td>
<td>129</td>
</tr>
<tr>
<td>5.2.1: Inputs for the Material Model</td>
<td>132</td>
</tr>
<tr>
<td>7.4.1: Tangent Moduli from Virtual Experiments – Axial Compression</td>
<td>175</td>
</tr>
<tr>
<td>7.6.1: Beta and Strain Rate Values for T700/PR520</td>
<td>178</td>
</tr>
</tbody>
</table>
8.2.1: Tie-Break Parameters ............................................................................................193
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.2.1: Composite Front Guards for a Motorcycle</td>
<td>4</td>
</tr>
<tr>
<td>1.2.2: Composite Wind Turbine Blades</td>
<td>4</td>
</tr>
<tr>
<td>1.2.3: Composite Turbojet Engine Case</td>
<td>5</td>
</tr>
<tr>
<td>1.2.4: Impact Damage Patterns</td>
<td>6</td>
</tr>
<tr>
<td>1.2.5: Triaxially Braided Composite Photograph</td>
<td>7</td>
</tr>
<tr>
<td>3.1.1.1: Impact Simulation for Baseline Method</td>
<td>29</td>
</tr>
<tr>
<td>3.1.3.1: Discretization of unit cell of triaxially braided composite for Braided Through the Thickness approach</td>
<td>39</td>
</tr>
<tr>
<td>3.1.3.2: Schematic of subcell discretization of six layer triaxially braided composite</td>
<td>39</td>
</tr>
<tr>
<td>3.1.3.3: Through Thickness optical image showing nesting</td>
<td>41</td>
</tr>
<tr>
<td>3.1.5.1: Simulation Schematics</td>
<td>44</td>
</tr>
<tr>
<td>3.1.5.2: Test vs. Simulation for T700/PR520 in Axial Tension</td>
<td>46</td>
</tr>
<tr>
<td>3.1.5.3: Test vs. Simulation for T700/PR520 in Transverse Tension</td>
<td>47</td>
</tr>
<tr>
<td>3.1.5.4: Test vs. Simulation for T700/3502 in Axial Tension</td>
<td>47</td>
</tr>
<tr>
<td>3.1.5.5: Test vs. Simulation for T700/3502 in Transverse Tension</td>
<td>48</td>
</tr>
<tr>
<td>3.1.5.6: Single Layer Through Thickness</td>
<td>50</td>
</tr>
<tr>
<td>3.1.5.7: Six Layer Stacked Through Thickness</td>
<td>50</td>
</tr>
<tr>
<td>3.1.6.1.1: Baseline Parametric Study T700/PR520 with Longitudinal Tensile Strengths Varied for the Axial Tension Simulation</td>
<td>52</td>
</tr>
<tr>
<td>3.1.6.1.2: Baseline Parametric Study T700/PR520 with Longitudinal Tensile Strengths Varied for the Transverse Tension Simulation</td>
<td>52</td>
</tr>
</tbody>
</table>
3.1.6.1.3: Baseline Parametric Study T700/PR520 with Transverse Tensile Strengths Varied for the Axial Tension Simulation

3.1.6.1.4: Baseline Parametric Study T700/PR520 with Transverse Tensile Strengths Varied for the Transverse Tension Simulation

3.1.6.1.5: Baseline Parametric Study T700/PR520 with In-Plane Shear Strengths Varied for the Axial Tension Simulation

3.1.6.1.6: Baseline Parametric Study T700/PR520 with In-Plane Shear Strengths Varied for the Transverse Tension Simulation

3.1.6.1.7: Baseline Parametric Study T700/3502 with Longitudinal Strengths Varied for the Axial Tension Simulation

3.1.6.1.8: Baseline Parametric Study T700/3502 with Longitudinal Strengths Varied for the Transverse Tension Simulation

3.1.6.1.9: Baseline Parametric Study T700/3502 with Transverse Strengths Varied for the Axial Tension Simulation

3.1.6.1.10: Baseline Parametric Study T700/3502 with Transverse Strengths Varied for the Transverse Tension Simulation

3.1.6.1.11: Baseline Parametric Study T700/3502 with In-Plane Shear Strengths Varied for the Axial Tension Simulation

3.1.6.1.12: Baseline Parametric Study T700/3502 with In-Plane Shear Strengths Varied for the Transverse Tension Simulation

3.1.6.2.1: Single Layer Comparison T700/PR520 for the Axial Tension Simulation

3.1.6.2.2: Single Layer Comparison T700/PR520 for the Transverse Tension Simulation

3.1.6.2.3: Single Layer Comparison T700/3502 for the Axial Tension Simulation

3.1.6.3.1: Single Layer Parametric Study T700/PR520 with Longitudinal Tensile Strengths Varied for the Axial Tension Simulation

3.1.6.3.2: Single Layer Parametric Study T700/PR520 with Longitudinal Tensile Strengths Varied for the Transverse Tension Simulation

3.1.6.3.3: Single Layer Parametric Study T700/PR520 with Transverse Tensile Strengths Varied for the Axial Tension Simulation
3.1.6.3.4: Single Layer Parametric Study T700/PR520 with Transverse Tensile Strengths Varied for the Transverse Tension Simulation .............................................................67

3.1.6.3.5: Single Layer Parametric Study T700/PR520 with In-Plane Shear Strengths Varied for the Axial Tension Simulation .................................................................68

3.1.6.3.6: Single Layer Parametric Study T700/PR520 with In-Plane Shear Strengths Varied for the Transverse Tension Simulation .........................................................69

3.1.6.3.7: Single Layer Parametric Study T700/3502 with Longitudinal Tensile Strengths Varied for the Axial Tension Simulation .............................................................70

3.1.6.3.8: Single Layer Parametric Study T700/3502 with Longitudinal Tensile Strengths Varied for the Transverse Tension Simulation .........................................................71

3.1.6.3.9: Single Layer Parametric Study T700/3502 with Transverse Tensile Strengths Varied for the Axial Tension Simulation .............................................................72

3.1.6.3.10: Single Layer Parametric Study T700/3502 with Transverse Tensile Strengths Varied for the Transverse Tension Simulation .........................................................72

3.1.6.3.11: Single Layer Parametric Study T700/3502 with In-Plane Shear Strengths Varied for the Axial Tension Simulation .............................................................73

3.1.6.3.12: Single Layer Parametric Study T700/3502 with In-Plane Shear Strengths Varied for the Transverse Tension Simulation .........................................................74

3.1.6.3.13: Effects of varying unidirectional transverse tensile strength on response of individual subcells for axial tensile simulation of T700/PR520 composite using single layer analysis model .................................................................76

3.1.6.3.14: Effects of varying unidirectional in-plane shear strength on response of individual subcells for axial tensile simulation of T700/PR520 composite using single layer analysis model .................................................................76

3.1.6.3.15: Effects of varying unidirectional transverse tensile strength on response of individual subcells for axial tensile simulation of T700/3502 composite using single layer analysis model .................................................................77

3.1.6.3.16: Effects of varying unidirectional in-plane shear strength on response of individual subcells for axial tensile simulation of T700/3502 composite using single layer analysis model .................................................................78

Figure 3.1.6.4.1: Axial tensile simulations of T700S/PR520 composite using shifted six layer, single layer and stacked six layer analysis models ........................................79

3.1.6.4.2: Transverse tensile simulations of T700S/PR520 composite using shifted six layer, single layer and stacked six layer analysis models ........................................79
3.2.1: Through Thickness View of Unit Cell .................................................................81
3.2.2: Highlighted by orientation ..................................................................................82
3.2.3: Impact Damage Pattern for a Homogenized Plate .............................................83
3.2.4: Impact damage patterns comparing laminated shell theory .................................84
4.1.2.1: Identification of the unit cell a) specimen photograph, b) four shell elements
(A,B,C,D), c) through thickness layers ......................................................................89
4.1.2.2: Schematic of subcell discretization of six layer triaxially braided composite .....89
4.1.4.1: Independently Homogenized Subcells Pseudocode ........................................93
4.1.4.3.1: Axial Tension Experimental Coupon Test T700/3502 Displaying Stiffness
Degradation Point (Point-of-Nonlinearity) .................................................................98
4.1.4.3.2: Illustration of the SLIMT1 (Stress Limiting Parameter in Axial Tension) ....100
4.1.4.3.3: Illustration of the Strength Correction ........................................................101
4.1.4.4.1: Image of the unit cell in transverse tension ..................................................104
5.1.1: Coupon Test Schematics .....................................................................................111
5.1.2: Raw Experimental Data from 2 Tests .................................................................113
5.1.3: Interpolated Values and Average with Error Bars .............................................114
5.1.4: Average Curve for T700/PR520 Axial Tension Test with Error Bars representing
Standard Deviation ....................................................................................................116
5.1.5: Average Curve for T700/PR520 Transverse Tension Test with Error Bars
representing Standard Deviation .................................................................................117
5.1.6: Average Curve for T700/PR520 Axial Compression Test with Error Bars
representing Standard Deviation ................................................................................118
5.1.7: Average Curve for T700/PR520 Transverse Compression Test with Error Bars
representing Standard Deviation .................................................................................119
5.1.8: Average Curve for T700/E862 Axial Tension Test with Error Bars representing
Standard Deviation ....................................................................................................120
5.1.9: Average Curve for T700/E862 Transverse Tension Test with Error Bars
representing Standard Deviation ................................................................................121
5.1.10: Average Curve for T700/E862 Axial Compression Test with Error Bars representing Standard Deviation ................................................................. 122

5.1.11: Average Curve for T700/E862 Transverse Compression Test with Error Bars representing Standard Deviation ...................................................................................... 123

5.1.12: Average Curve for T700/5208 Axial Tension Test with Error Bars representing Standard Deviation ........................................................................................................... 124

5.1.13: Average Curve for T700/5208 Transverse Tension Test with Error Bars representing Standard Deviation ...................................................................................... 125

5.1.14: Average Curve for T700/5208 Axial Compression Test with Error Bars representing Standard Deviation ...................................................................................... 126

5.1.15: Average Curve for T700/5208 Transverse Compression Test with Error Bars representing Standard Deviation ...................................................................................... 127

5.1.16: Notched Schematic .............................................................................................. 130

5.2.1: Shear Simulation Schematics ................................................................................. 131

5.2.2: Axial Tension Experimental vs. Computed T700/PR520 Composite Coupon Test ........................................................................................................................................ 134

5.2.3: Transverse Tension Experimental vs. Computed T700/PR520 Composite Coupon Test ................................................................................................................................... 134

5.2.4: Transverse Tension Experimental Straight Sided and Notched vs. Computed T700/PR520 Composite Coupon Test ................................................................. 135

5.2.5: Axial Compression Experimental vs. Computed T700/PR520 Composite Coupon Test ..................................................................................................................... 136

5.2.6: Transverse Compression Experimental vs. Computed T700/PR520 Composite Coupon Test ..................................................................................................................... 136

5.2.7: Shear Computed T700/PR520 Composite Coupon Test ......................................... 137

5.2.8: Axial Tension Experimental vs. Computed T700/E862 Composite Coupon Test 138

5.2.9: Transverse Tension Experimental vs. Computed T700/E862 Composite Coupon Test ..................................................................................................................... 139

Figure 5.2.10: Axial Compression Experimental vs. Computed T700/E862 Composite Coupon Test ..................................................................................................................... 140

5.2.11: Transverse Compression Experimental vs. Computed T700/E862 Composite Coupon Test ..................................................................................................................... 140
5.2.12: Shear Computed T700/E862 Composite Coupon Test .................................................. 141
5.2.13: Axial Tension Experimental vs. Computed T700/5208 Composite Coupon Test .......................................................................................................................................... 142
5.2.14: Transverse Tension Experimental vs. Computed T700/5208 Composite Coupon Test ..................................................................................................................... 143
5.2.15: Axial Compression Experimental vs. Computed T700/5208 Composite Coupon Test ................................................................................................................................... 144
5.2.16: Transverse Compression Experimental vs. Computed T700/5208 Composite Coupon Test ................................................................................................................................... 145
5.2.17: Shear Computed T700/5208 Composite Coupon Test .................................................. 146
5.2.18: Axial Tension Experimental vs. Computed T700/3502 Composite Coupon Test .......................................................................................................................................... 147
5.2.19: Transverse Tension Experimental vs. Computed T700/3502 Composite Coupon Test ..................................................................................................................... 148
5.2.20: Axial Compression Experimental vs. Computed T700/3502 Composite Coupon Test ................................................................................................................................... 149
5.2.21: Transverse Compression Experimental vs. Computed T700/3502 Composite Coupon Test ................................................................................................................................... 150
5.2.22: Shear T700/3502 Composite Coupon Test ......................................................................... 151
6.2.1.1: Simulated Al 2024 Projectile Dimensions ........................................................................ 152
6.2.1.2: Finite Element Mesh for Impact Simulations ...................................................................... 153
6.2.1.3: Finite Element Impact Simulation Setup ............................................................................ 154
6.2.2.1: Simulated Gelatin Projectile Dimensions .......................................................................... 155
6.2.2.2: Finite Element Mesh for Deflection Simulations ................................................................. 156
6.2.3.1: Composite Tube Simulation and Tube Test Sample .............................................................. 157
6.2.3.2: Tube Pressurization Simulation Diagram ............................................................................. 158
6.3.1.1: Simulated (left) and Experimental (right) Damage Patterns for T700/E-862 Composite in Impact .......................................................................................................................... 159
6.3.1.2: Simulated Damage Patterns for T700/PR520 Composite in Impact ................................. 160
| A.2.4: I.H.S. icanstr2_SAE.m Source Code | 217 |
| A.2.5: I.H.S. icanstr2_SI.m Source Code | 220 |
| A.2.6: I.H.S. subican.m Source Code | 223 |
| A.2.7: I.H.S. compliance.m Source Code | 224 |
| A.2.8: I.H.S. axial_t_LL.m Source Code | 226 |
| A.2.9: I.H.S. trans_t_LL.m Source Code | 228 |
| A.2.10: I.H.S. trans_t_LLzero.m Source Code | 230 |
| A.2.11: I.H.S. shear_LL.m Source Code | 232 |
| A.2.12: I.H.S. writeInputCard.m Source Code | 234 |
| A.2.13: I.H.S. writeInputExcel.m Source Code | 236 |
| A.2.14: I.H.S. input_calc.m Source Code | 237 |
CHAPTER I
MOTIVATION & INTRODUCTION

1.1 Motivation

Polymer matrix composites are becoming popular material choices for aerospace and other advanced structural components, including impact resistant applications. Polymer matrix composite materials are utilized for a variety of aerospace and other structural impact applications because of their reduced weight, and versatile material properties. Composite structures may undergo a variety of loading conditions during service life. In order to be used as protective structures, composites are required to resist ballistic impact loading. Triaxially braded composites with a [+60°/0°/-60°] layup are being used for their optimal combination of static and impact properties. Some of the benefits of using braided composites instead of laminated composites include a greater cost savings and inter-laminar strength (Swanson & Smith, 1996). Braided composites also exhibit a higher impact resistance when compared to unidirectional laminated composites (Pandey & Hahn, 1994). Impact conditions, it is necessary to have a model that accurately simulates both ballistic limit and damage patterns for the impact region. The ballistic limit of a structure is the point at which a projectile travels completely through the material but the projectile’s nominal residual velocity equals zero. The ballistic limit is generally a range determined from a full array of impact experiments. The ballistic limit and damage mechanisms are highly dependent on the material
constituents, the structure’s layup, the textile architecture, and the shape of the projectile, and every composite material behaves differently under impact loading conditions. Specifically, in a braided composite, numerous micromechanical phenomena occur contributing to both the ballistic limit and the damage patterns. This study focuses on quasi-isotropic triaxial braids with a large unit cell size. As can be seen in Figure 1.2.5, the unit cell has dimensions 17.8mm by 5.1mm. Because of this large unit cell size, the materials experience strain and damage more like structures, rather than homogenized materials. For this reason, the high velocity impact damage patterns are highly dependent on the material architecture and projectile shape (Littell, 2008). Classical homogenized material analysis methods where fully smeared composites properties are computed cannot account for these architecture and shape effects.

There has been a significant amount of research conducted in textile composite modeling and analysis. In most of the previous research, homogenized material properties were computed and used as material properties within a finite element analysis. For example, both Tanov and Tabiei, as well as Bednaryck and Arnold used a representative volume cell approach which assigned homogenized material properties to the model elements (Tanov & Tabiei, 2001), (Bednaryck & Arnold, 2003). Another type of analysis was conducted by Jenq and Mo which also assumed similar fully homogenized element properties (Jenq & Mo, 1996). These types of approaches do not directly account for the textile architecture in the finite element model and work well for small yarn sizes but lack the ability to capture damage in a composite with larger yarn sizes. Failure propagating along fiber directions has been noted in experimental impact testing in some textile composites with large yarn sizes (Roberts, et al., 2002). In order to
simulate the unique failure propagation along the fiber direction, the patterns in the textile architecture should be accounted for in the finite element model. This study will discuss a method that has been developed to systematically capture architecturally dependent damage.

1.2 Introduction

There are a number of applications for impact resistant triaxially braided composite materials. One application is as protective structures in vehicles such as the front guards shown in Figure 1.2.1. Another application for these composites is as wind turbine blades used for power generation systems (Figure 1.2.2). These blades are frequently subjected to bird strike impact. Current estimates indicate an average of three bird strikes occur per wind turbine every year in the United States (Manville, 2005). A third application for triaxially braided composite materials is for use in turbojet engine fan blade containment systems. Similar to the wind turbines for power generation, avian ingestion can in some cases lead to a blade out event in turbojet engines. In order to optimize structural design for these three examples, the failure, damage, and deformation of the composite material needs to be simulated with the use of explicit finite element codes. A design tool is required to capture the structural effect of the braid and damage along fiber bundles, while modeling the entire structure. In order to balance these needs, a macro scale finite element simulation technique was developed. For these impact resistant applications, the model needed to accurately simulate the textile composite under impact loading situations.
Figure 1.2.1: Composite Front Guards for a Motorcycle (Motorcycle Superstore, 2012)

Figure 1.2.2: Composite Wind Turbine Blades (Colorado Wind Power, 2012)
Due to the large unit cell of the triaxially braided composites currently under study, the impact damage patterns seen in these composite structures are dependent on the material architecture, the target shape, and can also very slightly depending on the boundary conditions. Examples of this can be seen in Figure 1.2.4. The image in a) shows impact failure in a triaxially braided composite with a fixed square boundary and impacted with a gelatin projectile where the failure is a butterfly shape (Figure 1.2.4). The image in b) shows impact failure in a similar composite with a circular boundary and impacted with an aluminum projectile where failure occurs in a more irregular pattern with much more fiber pull out (Figure 1.2.4). Finally, the image in c) displays impact failure in a curve composite panel where initial failure occurs across the hoop direction of the material (Figure 1.2.4).
An analysis method was developed previously by Cheng and Littell to model the material response of braided composites (Cheng, 2006), (Littell, 2008). In the current work, their method is modified to improve the prediction of the high velocity impact damage patterns and strain rate dependence of the material response. To incorporate the material response, a systematic approach was developed for calculating the material properties required by the finite element material model utilizing experimental results. The focus of this paper is to fill this need for quasi-isotropic triaxially braided carbon-epoxy composites, but the methodology which will be presented is applicable to any braided or woven composite.

A design tool is required to capture the structural effect of the braid and damage along fiber bundles while modeling the impact response of a structure. In order to balance these needs, a macro scale finite element simulation was developed employing an advanced continuum damage mechanics material model in the finite element code LS-DYNA® (Hallquist, 2007). For these aerospace applications, the model needed to accurately simulate the textile composite under both static loading and impact loading.
situations. In particular, a triaxially braided composite with a \([+60^\circ/0^\circ/-60^\circ]\) layup was investigated. An image can be seen in Figure 1.2.5.

![Triaxially Braided Composite Photograph](image)

Figure 1.2.5: Triaxially Braided Composite Photograph

The materials investigated in the following chapters were composed of a two-dimensional triaxially braided preform and a 177°C cured epoxy resin. Four different braided composites are discussed. All composites were constructed using the same standard modulus carbon fiber. TORAYCA T700S fibers (Toray Carbon Fibers America, Inc.), a high strength standard modulus carbon fiber was used. For the first material, CYCCOM PR 520 (Cytec Industries, Inc.), a one part toughened resin was used (PR520). For the second material, EPICOTETM Resin 862/EPIKURETM Curing Agent W system (Hexion Specialty Chemicals) (E862), a two part low viscosity system was used. Cytec’s 5208 resin and Hexcel’s 3520 resin were used for the remaining two material systems. These four materials range in resin systems from toughened to brittle: PR520, E862, 5208, and 3502.

For the composites under consideration, six layers of a \([+60^\circ/0^\circ/-60^\circ]\) braided preform were stacked on top of each other. The axial fibers were 24k flattened tows
while the bias fibers were 12k flattened tows. The global fiber volume fraction for all of
the triaxially braided composites was determined to be approximately 56 percent.

Each of the following chapters details a different portion of this analysis. This
begins with an in depth literature survey in order to collect knowledge on the topic of
textile polymer composite finite element simulation in Chapter II. Next in Chapter III, an
effort to examine the capabilities of the material model being used to simulate triaxially
braided composite materials is described. In this chapter, a number of parametric studies
are conducted in order to obtain a deeper understanding of the sensitivity of the material
model. Chapter III illuminates a need for a systematic method to characterize the
material model. The Independently Homogenized Subcells program was developed in
order to meet this need for a methodology to systematically calculate the input
parameters required for the material model exclusively from experimental and
manufacturing data and to develop an approach to capture both the ballistic limit and
architecturally dependent damage in the finite element analysis. This program is fully
described in Chapter IV. In this program, a top-down approach was merged with a
bottom-up approach to determine the required input parameters. The top-down portion
used global strengths obtained from macro-scale experimental quasi-static coupon tests to
characterize the material strengths. The bottom-up portion used micro-scale fiber and
matrix stiffness properties to characterize the material stiffness. The analytical tool
Independently Homogenized Subcells was used to characterize the four composites being
investigated. In Chapter V, the characterization and calibration was accomplished by
using quasi-static properties from experimental testing. Statistics are performed on the
data to create average curves that are then used to calibrate the required input properties
to be used in the finite element analysis. Next, a set of representative flat panel impact tests are simulated using the analysis approach in Chapter VI. Because the input material parameters are correlated based on the coupon level tests, these simulations are purely predictive. Chapters VII and VIII investigate the expansion and further modification of the core analysis approach presented in this paper. In Chapter VII, the methodology is expanded to incorporate the strain rate dependence of the material response. Due to the lack of experimental data at high strain rates, results obtained from a series of virtual experiments conducted by Liu is utilized in order to characterize the parameters which control the strain rate dependence of the material within the constitutive model (Liu, 2011). A series of flat panel impact simulations are then carried out using a representative braided composite. In Chapter VIII, the approach is modified to account for delamination between the layers of braid. For ballistic impact loading types of applications, it may be important to have a model that accurately simulates not only the ballistic limit and damage patterns in an impact region, but also the effect of barely visible impact damage (BVID). Finally, Chapter IX presents a summary of the concepts discussed in this paper and a discussion of future work.
CHAPTER II
LITERATURE REVIEW

2.1 Introduction

The current trend in material mechanics analysis, the finite element method, is frequently employed to analyze fabric composite materials. This literature review will give a brief synopsis of impact analysis for fabric composites with regard to the finite element method and review other methods such as fracture and energy. First, textile composite modeling techniques involving finite element analysis will be presented. Then, elements of spalling and fracture pertaining to braided composites will be explored. Finally, ongoing research in macro-scale 2D triaxially braided composite impact analysis will be discussed.

2.1.1 Textile Polymer Composites

Textile composite materials absorb energy through a variety of mechanisms from matrix cracking and powdering, fiber breakage, and internal friction (Janapala, et al., 2008) to fiber jamming, in-plane twisting, and other architectural effects (Beard, et al., 2002). All the micromechanical phenomena contribute to the overall behavior of the textile composite and the most significant phenomena should be accounted for in order to model the material with accuracy (Janapala, et al., 2008). Recently, many assumptions have been made, and many algorithms have been developed in order to feasibly model textile composites.
Only a few textile composite models consider failure behavior. Progressive failure in composites, especially fabric (woven and braided) composites, is highly complex and requires much consideration. It is understood that the progressive failure behavior in braided composites is a product of the material nonlinearity of the matrix combined with the geometrical nonlinearity of the fibers reorienting, the damage accumulation accounting for stress concentrations, and the interaction between the fibers and the resin (Tabiei, et al., 2004).

Simulating an impact event involving braided composites poses many challenges. Impact modeling is dependent on many dynamic mechanical properties such as: the properties of the target, the target geometry, the shape, the size, mass, and velocity of the projectile (Naik, et al., 2004), (Warrior, et al., 2004). Impact results and modeling are also dependent on the type of test. Impact tests include tube crush, vertical drop and ballistic gas gun tests (Naik, et al., 2004), (Warrior, et al., 2004). Not only do the properties listed above affect the simulation results, but the accuracy of the projectile material model will also have a significant effect on the overall impact simulation results (Littell, 2008). Due to the fact that the research efforts described in this paper involve the simulation of gas gun impact tests with gelatin or metallic projectiles, this paper will look specifically at these tests.

2.1.2 Modeling Textile Polymer Composites

Woven and braided composites are becoming lower in cost and have shorter fabrication times than conventional laminated composites (Beard, et al., 2002). Due to these recent innovations, woven and braided composites are being used in traditional aerospace and defense applications, and also in low volume niche applications (Pickett, et
al., 2006). The expanding composite material market is creating a need for quick and accurate finite element simulations and the need to improve composite material models. However, there are a number of difficulties in accurately modeling fabric composites. Woven and braided composites carry load and dissipate energy in an entirely different fashion than unidirectional laminates. Fabric composites also pose problems for obtaining in-situ properties (Warrior, et al., 2004). Because of these difficulties, a number of simplifications and solutions have been proposed to fill the growing need.

One type of textile composite is a 2D triaxially braided composite. Like other textile material models, the theories for modeling triaxially braided composite developed for uniaxial composites are derived from older methods. Some of the early textile composite models used classical lamination theory with constant stress and constant strain assumptions. One example is given by Ishikawa and Chou (1983). Their mosaic-like one-dimensional model combines classical lamination theory with assumptions of constant stress and constant strain to yield elastic properties of plain weave composites. These classical laminate theory models were then extended by Naïk and Shembekar (1992), and Ganesh and Naik (1992). In these studies, two-dimensional analyses of plain weave composites were conducted. Through this process the geometry was more accurately modeled and the effects of the woven geometry on the elastic properties were more accurately captured. However, strength properties were only calculated in the fill yarn direction of loading. They divided the composite into cells and through-the-thickness slices, and used different failure criteria for the fill strand, the warp strand, and pure matrix regions (Ganesh & Naik, 1992). Another classical lamination theory derived approach was conducted by Mital et al. (1996). Mital used the micromechanics method
from the stress analysis code ICAN to obtain equivalent stresses in the warp and fill regions (Murthy & Chamis, 1986). These stresses were then homogenized to predict the elastic properties and microstresses for plain weave composites.

A repeating unit cell (RUC) was developed and used to discretize a composite into repeatable sections (Masters, et al., 1996). This also came to be known as the representative volume cell (RVC) or representative volume element (RVE) (Tabiei, et al., 2004). The mesomechanical RUC, RVC, or RVE is simply the smallest unit in the fabric composite that is repeatable such that many of those units could be joined to form the whole composite structure. Almost all fabric composite finite element models emanate from identifying an RUC in the structure (Masters, et al., 1996). Some examples of research utilizing RUCs include: Aggarwal et al. (2000), Beard et al. (2002), Quek et al. (2004), Flesher (2005), and Carey et al. (2005). Tanov and Tabiei (2001) used an RUC to discretize a plane weave woven composite into cells. To increase fidelity these cells were then divided into four subcells. A through-the-thickness homogenization process was used to obtain the elastic properties of each subcell. A number of recent methods have been developed to capture failure in composite fabrics. The majority of these methods focus on woven composites, but a few focus on braided composites. The TEXCAD software developed by R. Naik (1995) made use of three dimensional unit cells. TEXCAD models continuum damage mechanics by implementing an empirical stiffness reduction method. In a later work, Tabiei implemented a micromechanical failure criterion to determine the stiffness degradation of the material in order to simulate progressive failure of a woven composite material (Tabiei, et al., 2004).
Around the same time, a method was developed to calculate stiffness and strength properties in all directions for textile composites. Schweizerhof (1998) implemented Matzenmiller’s (1994) approach of continuum damage mechanics for composites in finite element analysis and expanded it for use in modeling homogenized fabric composites. Matzenmiller had previously developed a constitutive model for anisotropic damage of fibrous composite materials with non-ductile matrices (Matzenmiller, et al., 1994). This method was specifically designed for laminated composites. The method used continuum damage mechanics theory in the material axis system to approximate damage initiation and ultimate material failure. Each lamina was assumed to be a unidirectional composite and any nonlinearity was assumed to be due to damage mechanisms. The development of damage was completely dependent on the stress and strain states of the individual unidirectional lamina. In this method, Matzenmiller utilized the Hashin failure criteria which determined a failure envelope based on the five strength properties of the unidirectional lamina: longitudinal tension, longitudinal compression, transverse tension, transverse compression, and shear strengths (Hashin, 1980).

Schweizerhof implemented this approach in finite element analysis and expanded it for use in modeling laminated composites and homogenized fabric composites. Schweizerhof’s elastic damage model assumes the deformation of the composite introduces microcracks and cavities in the material (Schweizerhof, et al., 1998). A smeared type composite model can undergo sudden brittle failure. To alleviate this occurrence, Schweizerhof’s material model can be implemented as layered shell elements. There are two commonly used versions of this material model: one for woven composites, and one for arbitrary composites. The model version for woven composites
has smooth failure surfaces and the failure is the same in the axial (11) and transverse (22) directions. The model version for the arbitrary composite, where all stresses are treated in an uncoupled fashion during damage evolution, can used for braided composites.

2.1.3 Multi-Scale Textile Composite Modeling Approaches

Multi-scale composite modeling approaches are currently being developed and used to better characterize large scale composites with important micromechanical details. These approaches typically include a combination of two or more of the following: micro-scale, meso-scale, and macro-scale composite models. Micro-scale models generally simulate one fiber within a finite volume of matrix and the interface around the fiber. Meso-scale models are mostly small one or two mesomechanical unit cell models that explicitly simulate the fiber, matrix, and sometimes the interface of the composite. Macro-scale models typically contain many mesomechanical unit cells and use a form of smeared material properties. At the macro-scale, one can simulate, and analyze, whole composite structures; and one can simulate impact and large scale deformation. Each scale has advantages and limitations. For example, fiber matrix separation will not be viewable in a macro-scale model. The fidelity needed does not exist. If fiber matrix separation is vital for the model, a meso or micro-scale model should be simulated and analyzed. Complex composites, such as woven or braided composites, may need to be modeled in two or three scales to fully capture the material behavior.

Micromechanical finite element models account for the fiber, the matrix, and sometimes the fiber-matrix interface of small composite specimens. The micro-scale
models simulate the individual fibers and resin pockets. When individual fibers are modeled, events such as fiber splitting and micro-buckling can be observed. When resin pockets are modeled, events such as craze formation can be observed (Benzerga et al. 2009). Perhaps most important, micro-scale models, in general, include the area where the fibers and matrix meet, commonly referred to as the fiber-matrix interface. The fiber-matrix interface is included to more accurately define the composite material response. The interface typically has a strong effect on strength and failure modes in the composite specimen. One of the first high fidelity micro-scale finite element models was the Integrated Composite Analyzer ICAN (Murthy & Chamis, 1986). The ICAN program computed stiffness properties using a more realistic geometry where one cell contains pure resin and another cell contains a fiber-resin combination. Sun and Chen (1991) extended the micromechanics method used in ICAN to include separate compliance matrices for the pure resin and fiber-resin combination regions. Subsequently the Generalized Method of Cells (GMC) was developed (Paley & Aboudi, 1992). GMC divided the micromechanical model further into four cells, three with pure resin and one with fiber. Goldberg et al. (2005) also extended the ICAN micromechanical model. The slice micromechanics model divides the single fiber matrix unit cell into a number of horizontal slices. Each slice is then separated into the fiber and matrix portion yielding much more accurate equivalent stiffness properties (Goldberg, et al., 2005). Primarily, semi-analytic micromechanics methods have been discussed, but finite element analysis can also be used to calculate equivalent stiffness properties. An example of a recent high fidelity micro-scale finite element simulation can be observed in Benzerga et al. (2009).
Meso-scale finite element models are one or two mesomechanical unit cell models that explicitly simulate the fiber, matrix, and typically the interface of the composite. An example of a high fidelity mesomechanical model can be found in Quek et al (2004). Here Quek analyzes and models a representative unit cell (RUC) of a triaxially braided composite material. It should be noted that all scales are relative and are defined differently in each situation. In this paper, Quek’s model is defined as a meso-scale approach while in Quek et al (2004) the model was defined as a micro-scale approach. Another example of a mesomechanical model can be found in Aggarwal et al (2000). Aggarwal’s paper demonstrates meso-scale modeling of a diamond (biaxially) braided composite. Li’s model provides a third example of a mesomechanical model (Li, 2008). Li’s model provides a high fidelity finite element approach by modeling the individual fiber, matrix, and fiber-matrix interface of an RUC of triaxially braided composite. A meso-scale modeling approach or an approach that simulates one or two mesomechanical RUCs is the most common approach to analyzing braided and woven composites.

Macro-scale finite element models are large multi RUC models of smeared elements. Effective properties are typically obtained by using analytical micromechanics methods such as those found in Goldberg et al (2005). The ply properties can then be used to calculate overall material properties using the geometry of the mesomechanical RUC and Classical Laminated Plate Theory (CLPT) (Jones 1999). The composite material models incorporated into most finite element codes generally use some type of homogenized or smeared material stiffness and strength properties such as the moduli, Poisson’s ratios, and stress at failure. Examples of macromechanical material models for
triaxially braided composites primarily follow this logic. 2D triaxially braided composite materials are frequently modeled as a shell element mesh because the material is generally thin in comparison to the structure size. Lomov et al. (2004) used a fabric assembly program WiseTex to generate homogenized RUC stiffness properties from the mesomechanical model TexComp. The homogenized RUC properties were then implemented in a structural finite element model. Failure of the material was not included (Lomov, et al., 2004). McGregor et al. (2008) expanded CODAM, a continuum damage based macromechanical model. CODAM was designed for modeling crushing in braided composite tubes where the tubes are simulated as layers of shell elements. Cheng et al. (2008) used a braided-through-the-thickness approach to model triaxially braided composites materials. Cheng used shell element RUCs with integration layers at orientations corresponding to the fiber plys of the braid. Littell (2008) used Cheng’s braided-through-the-thickness approach and merged it with Schweizerhof’s material model.

2.1.4 Composite Failure under Impact Loading

Woven and braided composites are commonly used in situations of impact loading. There are three categories of impact loading: low velocity, high velocity, and hyper velocity. This paper will focus on high velocity impact also known as a ballistic impact. A ballistic impact response is governed by the local material behavior and is independent of boundary conditions (Naik, et al., 2004). These large scale high strain rate loading situations are most efficiently simulated with macro-scale models. However, impact events can also be simulated with meso-scale models, but these models are not typically computationally efficient. The fidelity of the model depends upon which
composite characteristics are desired for a particular application. Applications can range from ballistic armor to component containment. Generally attributes desired for composite impact simulations include: accurate deflection, ballistic limit (point at which the projectile penetrates the target but has no residual velocity), loss of stiffness or strength, and damage patterns (Bogetti, et al., 2003), (Cheng, et al., 2008), (Iannucci, et al., 2006), (Kim, et al., 1999). Another characteristic of fabric composites is the change in damage patterns as the velocity of the projectile increases. This phenomenon can be seen in Wu et al. (1995) where a woven composite’s damage pattern changes from a circle, to an oval, and then to a diamond as the velocity of the projectile increases. Similar transitions occur in most fabric composites.

A mesomechanical model may be required to simulate impact in order to capture large amounts of composite and local fiber elongation. For example, Bogetti uses a mesomechanical model to more accurately simulate a compliant ballistic armor (Bogetti, et al., 2003). Gu also uses a mesomechanical model with explicitly defined fibers modeled as solid elements undulating through the material (Gu, 2004). Gu’s mesomechanical model simulated a woven composite fabric. In a later paper, however, Gu switched to a macromachanical model with three dimensional RUCs in order to simulate a triaxially braided material (Gu, et al., 2005).

In general, macromechanical impact models are sufficient because most composites have a brittle matrix. Composites with a brittle matrix undergo mechanisms such as microcracking and fiber splitting under stress rather than ductile deformation. Kim developed a macromechanical model to simulate hail ice impact on a composite fabric panel in order to investigate barely visible impact damage (Kim, et al., 1999).
resultant based shell element marcomechanical model was developed by Schwer. Schwer’s model was used to simulate the impact of a woven composite with a gelatin projectile. This simulation was an approximation of a bird impact on an airplane wing (Schwer, et al., 1999). Shell element formulations such as Schwer’s are even more computationally efficient than macromechanical models with solid element RUCs. Iannucci also uses the shell element approach for a fabric composite. However, Iannucci’s composite was modeled as shell elements with layers of thin resin elements (interface elements) in between to capture delamination of the woven composite in an impact event (Iannucci, et al., 2006). This type of model has the ability to capture delaminations while maintaining some efficiency. In order to increase fidelity while maintaining efficiency, Cheng (2008) and Littell (2008) divided the shell element RUC into four subcells. These subcells were divided by the transitions in fiber undulation (Littell, 2008). The subcells were then divided into layers. Each layer represented a local fiber orientation of the triaxially braided composite material, and was modeled as an integration point in the shell element (Cheng, et al., 2008), (Littell, 2008). They then utilized Schweizerhof’s arbitrary composite material model (see above) to simulate impact of a triaxially braided composite plate with a gelatin projectile. The layered shell element approach allows the impact simulation to capture both delamination and velocity damage pattern progression. However, the transition of damage pattern with increasing velocity was not captured.

2.1.5 Composite Energy Absorption in Impact Event

There are a vast number of high strain rate and impact test setups, in this paper the focus will be on a gas gun impact test setup. The gas gun impact test is used most
predominately in high velocity (ballistic) impact research because of the relative repeatability of the test (Pereira, et al., 2010). However, this test is highly projectile dependent and the size, weight, and shape of the projectile must be considered when comparing different tests. For a gas gun impact test, energy loss can be calculated by balancing the kinetic portion of the conservation of energy equation.

The total energy loss in impact can be summarized in a simple equation where $E_L$ is the total energy loss, $V_S$ is the strike velocity, and $V_R$ is the residual velocity (Equation 2.1.5.1) (Morye, et al., 2000), (Cantwell, et al., 1999).

$$E_L = \frac{1}{2} m (V_S^2 - V_R^2) \quad (2.1.5.1)$$

This equation is expanded in all the energy characterization methods. There are two primary methods to characterize the energy loss in impact of woven composite laminates.

The first method to characterize energy loss is the sum method. In the sum method, energy loss is calculated for all the components of the composite and the energy losses for these components are summed to calculate the total energy loss (Zee, et al., 1993). The components accounted for are the energy absorption due to the following: fiber failure, delamination, and friction. The energy absorption due to the fiber is calculated as the energy density multiplied by the volume of fibers in the fracture zone. The energy absorption due to delamination is equivalent to the total energy loss minus the energy loss in stacked sheets of the same composite with no bond between the lamina. The energy absorption due to friction is calculated by multiplying the normal force by the material coefficient of friction. The sum method had many assumptions that were unrealistic for polymer matrix composites. Carbon fiber composites in particular
appeared to have artificially high friction absorption due to this method. Specifically carbon fiber composites and many other composites tend to transform into debris during an impact event. When the materials are transformed into debris, the area of contact between the materials decreases and so the energy absorbed by friction also decreases.

The second method, the primary/secondary yarn method, captures fiber pull-out and deformation of the composite. This method divides the composite into two types of areas, the primary yarns and the secondary yarns. A depiction of the primary and secondary yarns can be seen in Naik et al (2006). The primary yarns are defined as the yarns in the fabric that are directly impacted by the projectile. All other yarns are defined as secondary yarns. During an impact event, the primary yarns are strained to tensile failure and the secondary yarns experience elastic deformation. This method uses the geometry of the cone-shaped wave to calculate stress and strains in the primary and secondary yarns (Naik, et al., 2006). The energy absorption is calculated by summing the energy loss due to tensile stress in the primary yarns, the energy loss due to elastic deformation in the secondary yarns and the energy loss due to the kinetic energy of the cone-shaped wave (Naik, et al., 2004), (Naik, et al., 2006). The primary/secondary yarn method is the typical method used to calculate energy absorption in fabric composites.

2.2 Spalling

There is a frequently overlooked physical phenomenon which may increase the amount of energy a target can absorb under impact loading. This phenomenon is known as matrix spalling or spall fracture. The spall fracture of metals and alloys has been widely studied for many years, but like other mechanical phenomena the algorithms developed for metals cannot be directly used for composites. There is very little previous
research in composite matrix spalling. Spall fracture in metals has been shown to absorb up to 80 percent of the total impact energy (Antoun, et al., 2003). It is likely that composites can be designed to absorb an even larger fraction of total energy.

2.2.1 Epoxy Spalling

Neither the sum method, nor the primary/secondary yarn method, examined previously, account for matrix spalling. These methods do not neglect matrix spalling directly, but they try to capture matrix spalling using other mechanisms such as matrix cracking energy. Matrix cracking energy cannot accurately account for matrix spalling because the flying particles are not only absorbing fracture energy; they are also absorbing propelling kinetic energy. In reality, both mechanisms need to be accounted for to accurately compute total energy loss. There are many methods for metal spall fracture. These typically use a fracture based process that determines the critical impact velocity that produces spall for a given impactor thickness and a given target thickness. For example, one such process uses the critical impact velocity and the experimental parameters to determine the energy loss through equating the balance of energy and momentum (Antoun, et al., 2003). In this equation, $v$ is the impact velocity, $h_i$ is the thickness of the impactor, and $h_t$ is the thickness of the target (Equation 2.2.1.1). The energy loss is calculated and is the upper bound for the work of fracture in the material.

$$EL = \frac{\rho h_i v^2}{2} \left(1 - \frac{h_i}{h_t}\right)$$  \hspace{1cm} (2.2.1.1)

If the wave distributions are known, the initial velocity of the flying particles can be estimated. A graphical analysis may be useful for calculating the initial velocity of the flying particles in composite matrix spalling.
During an impact event the matrix can spall causing small bits of resin to fly off the back side of the composite target. If matrix spalling is significant, it is possible that only the fiber fabric will remain in the impact zone. An experimental example of spall fracture in metals can be seen in Antoun et al. (2003). Complete matrix spalling in a centralized region of impact has been seen in experimental testing with laminated fabric composites. The fiber fabric remaining in the impact zone is mostly undamaged. This phenomenon is important for a variety of reasons. The energy loss due to matrix spalling may not be negligible. The total energy loss equation could be thrown off due to neglecting the matrix spalling energy loss. If most of the matrix has spalled, the target will yield more, due to the unrestrained progressive reorientation of fiber fabric after the matrix has spalled. This effect would lead to a higher ballistic limit.

The energy methods for calculating energy absorbed by matrix spalling described here open up an area of future work in optimization. While initial optimization can be designed with energy methods, further accuracy will likely require some type of fracture mechanics. This may prove to be a difficult task because it was noted that spall fracture in metals actually sits on the line between classical fracture mechanics and microstatistical fracture mechanics (Antoun, et al., 2003). The processes that control macrocrack propagation are similar to those observed in spall fracture, but in general microstatistical fracture mechanics approaches are used to describe spall fracture. Microstatistical fracture mechanics alternatively relies on quasi-simultaneous nucleation and growth of many microcracks. In essence, spall fracture behaves similar to microcracks and macrocracks at the same time. Perhaps this calls for a combination theory for fracture mechanics that includes both of these methods, or more likely a
unique type of element for finite element analysis that has the capability for fracture spall.

Another phenomenon that could be increasing the energy absorption capability of a composite might be the occurrence of a high-wave-velocity. Materials with both a high modulus and low density will have a high-wave-velocity during a ballistic impact event. These materials disperse the strain wave more rapidly away from the point of impact, distributing the energy over a wider area and therefore preventing large strains from accumulating at the point of impact (Bogetti, et al., 2003).

2.3 Ongoing Research

Previous research has not accurately characterized triaxially braided composite material response. Micromechanical and mesomechanical models use a bottom up approach. A bottom up approach is where the material properties are calculated based on the individual constituent (fiber and matrix) properties. Macromechanical models use either this approach with many assumptions, or a top down approach. A top down approach is where the material composite properties are obtained from full composite specimen data. The top down approach yields much better strength correlation. In order to adequately model a braided composite, the bottom up and top down approaches should be merged. The bottom up approach should be used to calculate stiffness properties, while a top down approach should be used to calculate strength properties. The final impact simulation should be modeled in the macro scale, with homogenized strength and stiffness properties computed from a combination of a micromechanical model, a mesomechanical model, and a fabric geometry assembly program derived from Classical Lamination Theory (Jones, 1999).
Previous research has also not identified a systematic approach to modeling fabric composites. The systematic approach should be applicable to any fabric composite with any orientation of fibers. A straightforward algorithm should be developed to approximately characterize any woven or braided composite.

A material orientation dependent impact failure shape cannot be obtained with current analysis methods. A method should be developed that can capture failure shape. Current analysis methods focus on the failure shape of woven (not braided) composites, which experimentally appears as a diamond at high velocities (Wu, et al., 1995). If the macromechanical models properties are uniformly homogenized only a diamond impact failure shape will occur. For braided composites this diamond shape is not always realistic. A method should be developed to capture the various impact failure shapes of braided composites.

Strain Rate dependence has not been addressed in previous research. It has been noted in experiments that composites with strain rate dependent matrices have some overall strain rate dependence (Littell, 2008). This may be an important characteristic of the composite material for an impact simulation. A composite model should be developed that incorporates strain rate dependence. For impact simulations, this may prove a difficult task because the composite panels undergo a whole array of strain rates during an impact event. In order to incorporate this effect properly, an intrinsic algorithm needs to be developed to iteratively pull the strain rate data for each strain rate observed by each individual element in the simulation.

Alternative energy absorption phenomena, such as matrix spalling and high-wave-velocity, have not been included in previous fabric composite research. These alternative
energy absorption phenomena are important for increasing the ballistic limits of composites while maintaining a light weight. A method should be investigated to characterize these phenomena, so they can, at some point, be optimized in the design process.
CHAPTER III
METHOD INTRODUCTION & EXPLORATION

3.1 Investigation of Mat_58 Input Parameters

Previous research concerning the analysis of triaxially braided composite panels under impact loading was conducted by Cheng and Littell (Cheng, 2006), (Littell, 2008). The analytical method used discretized the braided composite into a series of parallel shell elements. Each element in this series was modeled as a laminated composite. This allowed for architecturally dependent damage to be modeled while maintaining a large unit cell which increases efficiency. This model was based on the identification of the repeating unit cell within the triaxial braid architecture seen in Figure 3.1.3.1(a). The unit cell was further divided into four subcells: ‘A’, ‘B’, ‘C’, and ‘D’ noted in Figure 3.1.3.1(b). These four elements were the building blocks of the model and were repeated in both in-plane directions until the desired size was created. The subcells divided the braid architecture into four areas that were individually approximated as uniaxial laminated composites. This can be seen in Figure 3.1.3.1(c). The braided composite is composed of six layers which can be seen in Figure 3.1.3.2. In this figure, the subcell shifting through the thickness of the unit cell was done to account for the nesting effect noted in the experimental samples (Littell, 2008).
3.1.1 Introduction

The motivation for the investigation of Littell’s approach was to determine which parameter adjustments control the damage shape in impact simulations. Figure 3.1.1.1 displays the damage patterns observed in impact simulations conducted using Littell’s approach. The actual damage shape for this impact setup, seen in experimental testing, is a butterfly pattern (Roberts, et al., 2002). One of these impact tests can be seen in Figure 1.2.4 a). The butterfly damage pattern is clearly not reflected in the simulation, and an extensive parametric investigation was conducted to determine why the damage patterns were not captured.

![Impact Simulation for Baseline Method](image)

Figure 3.1.1.1: Impact Simulation for Baseline Method

3.1.2 Comparison of Composite Material Models

Before Littell’s approach could be investigated and improved, the optimum finite element solver and material model was identified. The finite element software used in the analysis is LS-DYNA® (Hallquist, 2007). This solver is widely accepted by both
academia and industry and was found to be optimal for this study. LS-DYNA® is a commercially available transient dynamic finite element solver, which contains contact algorithms for ballistic impact modeling (Hallquist, 2007). This solver also contains a number of material models for simulating composites.

There are a number of models available for composites in the commercial finite element solver LS-DYNA®. These composite material models are listed in Table 3.1.2.1. These material models have been analyzed for their ability to model impact failure and damage patterns for the triaxially braided composite materials being discussed in this paper (Tabiei, 2008). Many of these material models have the ability to model impact and failure, these include; *Mat_22, 54, 55, 58, 59, 158, 161, and 162 (Day, 2011). Of these models all but one has the capability to model non-linear material behavior, which is necessary to capture damage in the triaxially braided material. *Mat_54, because of the type of element used in the model, is only suitable for very thin shells (Hallquist, 2007). Because the six layer braided composites are 6.35 mm thick and the in-plane dimensions of the elements are 4.45 mm by 5.1 mm, the elements cannot realistically be approximated as a very thin shells. *Mat_161, and *Mat_162 are proprietary models developed by an external company (MSC) and were not available for this study (Materials Sciences Corporation, 2012).
### Table 3.1.2.1: Comparison of Composite Material Models within LS-DYNA®

<table>
<thead>
<tr>
<th>#</th>
<th>Name</th>
<th>Element</th>
<th>Laminated Shell Theory</th>
<th>Impact Modeling</th>
<th>Failure Criteria</th>
<th>Linear/Non-linear</th>
</tr>
</thead>
<tbody>
<tr>
<td>22</td>
<td>Mat Composite Damage</td>
<td>both</td>
<td>x</td>
<td>x</td>
<td>Chang-Chang</td>
<td>Linear</td>
</tr>
<tr>
<td>54</td>
<td>Mat Enhanced Composite Damage</td>
<td>thin shell</td>
<td>x</td>
<td>x</td>
<td>Chang-Chang</td>
<td>Non-linear</td>
</tr>
<tr>
<td>55</td>
<td>Mat Enhanced Composite Damage</td>
<td>solid</td>
<td>x</td>
<td>x</td>
<td>Tsai-Wu</td>
<td>Non-linear</td>
</tr>
<tr>
<td>58</td>
<td>Mat Laminated Composite Fabric</td>
<td>shell</td>
<td>x</td>
<td>x</td>
<td>modified Hashin</td>
<td>Non-linear</td>
</tr>
<tr>
<td>59</td>
<td>Mat Composite Failure</td>
<td>both</td>
<td>x</td>
<td></td>
<td>Chang-Chang</td>
<td>Non-linear</td>
</tr>
<tr>
<td>114</td>
<td>Mat Layered Linear Plasticity</td>
<td></td>
<td>x</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>116</td>
<td>Mat Composite Layup</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>117</td>
<td>Mat Composite Matrix</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>118</td>
<td>Mat Composite Direct</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>158</td>
<td>Mat Rate Sensitive Composite Fabric</td>
<td>shell</td>
<td>x</td>
<td>modified Hashin</td>
<td>Non-linear</td>
<td></td>
</tr>
<tr>
<td>161</td>
<td>Mat Composite MSC</td>
<td>solid</td>
<td>x</td>
<td>MSC</td>
<td>Non-linear</td>
<td></td>
</tr>
<tr>
<td>162</td>
<td>Mat Composite DMG MSC</td>
<td></td>
<td>x</td>
<td>DMG MSC</td>
<td>Non-linear</td>
<td></td>
</tr>
<tr>
<td>221</td>
<td>Mat Orthotropic Simplified Damage</td>
<td>solid</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>235</td>
<td>Mat Micromechanics Dry Fabric</td>
<td>shell</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Further investigation was required for the remaining material models which were both capable of modeling failure due to impact and capable of modeling material nonlinearity. The material models, listed in Table 3.1.2.2, were analyzed in more depth by comparing their options. These options dictate the equations used from the overall failure criteria to create the failure surface. *Mat_58 option ‘a’ was developed to be a quick approximation of the modified Hashin failure criteria, and due to this was not
found optimal for a macro scale simulation of braided composites (Schweizerhof, et al., 1998). *Mat_58 option ‘b’ uses a smooth failure surface where the strengths and failure strains in the longitudinal direction are combine with the strengths and failure strains in the transverse direction to achieve mixed mode failures (Schweizerhof, et al., 1998). Because of the way the triaxially braided composite is broken up into subcells which have vastly different strengths and failure strains in the two in-plane directions with some interaction but not the type of smoothing described by Schweizerhof et al., this model was determined to be unsuitable for this analysis. The *Mat_58 option ‘c’ model proved to be useful because of the faceted failure surface’s ability to model arbitrary composite fabrics and arbitrary laminates, proving its usefulness for this study (Schweizerhof, et al., 1998). An additional benefit of the *Mat_58 material options is that there is the ability to use laminated shell theory. Laminated shell theory changes the type of element used from a traditional Reissner-Mindlin element to an element that can model a non-linear through thickness strain. Both options of *Mat_59 were determined to be unsuitable for this analysis due to the symmetric in-plane failure definitions. The *Mat_158 options are exactly the same as the *Mat_58 options and therefore can be evaluated in the same way. However, it should be noted that the laminated shell theory is not available in *Mat_158. The benefit of *Mat_158 is that it incorporates strain rate. *Mat_158 will be discussed in greater detail in Chapter VI.
<table>
<thead>
<tr>
<th>#</th>
<th>Failure Criteria</th>
<th>Failure Surface</th>
<th>Recommendation</th>
</tr>
</thead>
<tbody>
<tr>
<td>58a</td>
<td>modified Hashin</td>
<td>smooth failure surface</td>
<td>laminates and unidirectional composites</td>
</tr>
<tr>
<td>58b</td>
<td>modified Hashin</td>
<td>unidirectional failure surface</td>
<td>laminates and symmetric woven composites</td>
</tr>
<tr>
<td>58c</td>
<td>modified Hashin</td>
<td>facetted failure surface</td>
<td>laminates, woven, and braided composites</td>
</tr>
<tr>
<td>59a</td>
<td>Chang-Chang</td>
<td>faceted failure surface</td>
<td>symmetric woven composites</td>
</tr>
<tr>
<td>59b</td>
<td>Chang-Chang</td>
<td>ellipsoidal failure surface</td>
<td>symmetric woven composites</td>
</tr>
<tr>
<td>158a</td>
<td>modified Hashin</td>
<td>smooth failure surface</td>
<td>laminates and unidirectional composites</td>
</tr>
<tr>
<td>158b</td>
<td>modified Hashin</td>
<td>unidirectional failure surface</td>
<td>laminates and symmetric woven composites</td>
</tr>
<tr>
<td>158c</td>
<td>modified Hashin</td>
<td>facetted failure surface</td>
<td>laminates, woven, and braided composites</td>
</tr>
</tbody>
</table>

After careful evaluation of the material models available, *Mat_58 with option ‘c’ was chosen to represent the material systems under investigation. This is a laminated composite material model derived from Matzenmiller’s method (*Mat_58).

Matzenmiller developed a constitutive model for anisotropic damage of fibrous composite materials with non-ductile matrices (Matzenmiller, et al., 1995). The method employed continuum damage mechanics theory in the material axis system to approximate damage initiation and ultimate material failure. Each element was assumed to be a laminated composite, and any nonlinearity was assumed to be due to damage mechanisms. The development of damage was completely dependent on the stress and strain states of the individual unidirectional lamina. Matzenmiller adopted the Hashin failure criteria in this approach, which determines a failure envelope based on the five strength properties of the unidirectional lamina: longitudinal tension, longitudinal...
compression, transverse tension, transverse compression, and shear strengths (Hashin, 1980). Equations 3.1.2.1, 3.1.2.2, 3.1.2.3, and 3.1.2.4 are used to describe this envelope. In these equations, $e_f$, $e_c$, $e_m$, and $e_d$ are failure criteria parameters; the axial stress, transverse stress, and shear stress are $\sigma_{11}$, $\sigma_{22}$, and $\tau$ respectively; and the axial tensile strength, axial compressive strength, transverse tensile strength, transverse compressive strength, and shear strength are $X_t$, $X_c$, $Y_t$, $Y_c$, and $S_c$. It should be noted that continuum damage mechanics for non-linear shear terms is only included in *Mat_58 option ‘c’, but not in options ‘a’ or ‘b’.
Tensile fiber mode I: $\sigma_{11} \geq 0$
\[
e_f^2 = \left(\frac{\sigma_{11}}{\bar{X}_t}\right)^2 - 1 \left\{\begin{array}{ll}
\geq 0 & \text{failed} \\
< 0 & \text{elastic}
\end{array}\right\}
\]
(3.1.2.1)

Compressive fiber mode II: $\sigma_{11} < 0$
\[
e_c^2 = \left(\frac{\sigma_{11}}{\bar{X}_c}\right)^2 - 1 \left\{\begin{array}{ll}
\geq 0 & \text{failed} \\
< 0 & \text{elastic}
\end{array}\right\}
\]
(3.1.2.2)

Tensile matrix mode III: $\sigma_{22} \geq 0$
\[
e_m^2 = \left(\frac{\sigma_{22}}{\bar{Y}_t}\right)^2 + \left(\frac{\tau}{\bar{S}_c}\right)^2 - 1 \left\{\begin{array}{ll}
\geq 0 & \text{failed} \\
< 0 & \text{elastic}
\end{array}\right\}
\]
(3.1.2.3)

Compressive matrix mode IV: $\sigma_{22} < 0$
\[
e_d^2 = \left(\frac{\sigma_{22}}{\bar{Y}_c}\right)^2 + \left(\frac{\tau}{\bar{S}_c}\right)^2 - 1 \left\{\begin{array}{ll}
\geq 0 & \text{failed} \\
< 0 & \text{elastic}
\end{array}\right\}
\]
(3.1.2.4)

Failure Criterion for Hashin Envelope

The equations used in the LS-DYNA® material model *MAT_58 reflect the Matzenmiller method and assumptions for continuum damage mechanics (Hallquist, 2007). Below are Equations 3.1.2.5 - 3.1.2.18 describing damage criteria and how they are used to reduce the modulus during damage and failure progression (Matzenmiller, et al., 1995). Omega is the damage variable. It is initially set to zero and it increases from zero to one as damage increases. Once an omega has reached the value one, the material is fully damaged in the corresponding failure direction. The strengths in the x and y directions are represented by $\sigma_x$ and $\sigma_y$. The strains in the x, y, and x-y shear directions are represented by $\varepsilon_x$, $\varepsilon_y$, and $\gamma$. The elastic strains for these directions are represented by $\varepsilon_{xe}$, $\varepsilon_{ye}$, and $\varepsilon_{se}$. True strain for the various directions is represented by $e$. The elastic modulus for the x, y, and x-y shear directions is represented by $E_x$, $E_y$, and $G_{xy}$. The Poisson’s ratio for the x and y directions is represented by $\nu_x$, and $\nu_y$. The stiffness matrix terms which are used are defined as $C_{11}$, $C_{12}$, and $C_{22}$. A partial elastic modulus in the 12
and 21 directions are indicated by $E_{12}'$ and $E_{21}'$. The remaining elastic modulus terms are $E_{11}$, $E_{12}$, $E_{21}$, and $E_{44}$. The formulas for terms $b_x$, $b_y$, $b_z$, and $D$ are included in Equations 3.1.2.7-4.1.9, and 3.1.2.13. The symbol $\alpha$ is an empirically based variable that generally is determined to be around 0.1 (Matzenmiller, et al., 1995). The variable $m$ is a failure variable determined by the equations of $\alpha$, if $\alpha$ is constant then $m$ is set equal to 1 (Matzenmiller, et al., 1995). For arbitrary composites (*Mat_58 option ‘c’) the facetted failure surface is used. Equations 3.1.2.19 – 3.1.2.22 seen below show that there is almost no coupling of the failure equations for this type of failure surface (Schweizerhof 1998). However, when used for laminated composites with multiple fiber orientations coupling will occur in the overall composite (Chapter III). In this way, the Hashin failure criteria was modified to allow for the modeling of arbitrary composites, which may have vastly different strengths in the five defined orientations.

\[
\begin{align*}
\sigma_x &= c_{11}\varepsilon_x + c_{12}\varepsilon_y \\
\sigma_y &= c_{12}\varepsilon_x + c_{22}\varepsilon_y \\
b_x &= \frac{E_x}{\alpha X_{e,t}} \\
b_y &= \frac{E_y}{\alpha Y_{e,t}} \\
b_z &= \frac{G_{xy}}{\alpha S_c} \\
\omega_1 &= 1 - \exp\left(-\frac{1}{me}(b_x \varepsilon_{se})^m\right) \\
\omega_2 &= 1 - \exp\left(-\frac{1}{me}(b_y \varepsilon_{ye})^m\right) \\
\omega_3 &= 1 - \exp\left(-\frac{1}{me}(b_z \varepsilon_{se})^m\right)
\end{align*}
\]
\[ D = 1 - (1 - \omega_{11c,t})(1 - \omega_{22c,t})v_{12}v_{21} = 0 \quad (3.1.2.13) \]

\[ E_{11} = c_{11} = \frac{E_s}{D^2} \left[ \varepsilon_x + (1 - \omega_2)\varepsilon_y \right] \frac{d\omega_1}{d\varepsilon_x} \]

\[ E'_{12} = c_{12} = \frac{(1 - \omega_2)\varepsilon_y}{D^2} \left[ (1 - \omega_1)\varepsilon_x + \varepsilon_y \right] \frac{d\omega_2}{d\varepsilon_y} \quad (3.1.2.14) \]

\[ E'_{21} = c_{12} = \frac{(1 - \omega_2)\varepsilon_y}{D^2} \left[ \varepsilon_x + (1 - \omega_1)\varepsilon_y \right] \frac{d\omega_1}{d\varepsilon_y} \]

\[ E_{22} = c_{22} = \frac{E_y}{D^2} \left[ (1 - \omega_1)\varepsilon_x + \varepsilon_y \right] \frac{d\omega_2}{d\varepsilon_y} \quad (3.1.2.15) \]

\[ E_{44} = c_{44} = G_{xy} \frac{d\omega_x}{d\gamma_{xy}} \]

\[ E_{12} = E_{21} = \frac{E_{12} + E_{21}}{2} \quad (3.1.2.16) \]

Equations implemented in LS-DYNA® *Mat_58

The final failure criteria equations used in the material model are shown in Equations 3.1.2.20-3.1.2.22. The failure criteria parameters in these equations are called \( f_\parallel, f_\perp, f_s \). The damage thresholds, which are determined by the size of the elastic region, are denoted as \( r_{\parallel c,t}, r_{\perp c,t}, \) and \( r_s \) (Schweizerhof, et al., 1998).

\( f_\parallel = \frac{\sigma_{\parallel t}^2}{(1 - \omega_{11c,t})^2 x_{\parallel t}^2} - r_{\parallel c,t} = 0 \quad (3.1.2.20) \)

\( f_\perp = \frac{\sigma_{\perp t}^2}{(1 - \omega_{22c,t})^2 y_{\perp t}^2} - r_{\perp c,t} = 0 \quad (3.1.2.21) \)

\( f_s = \frac{\gamma_s^2}{(1 - \omega_{12})^2 s_e^2} - r_s = 0 \quad (3.1.2.22) \)

(For Facetted Loading Surface *Mat_58 option ‘c’)

3.1.3 Approach

The braid architecture was incorporated in this model by using equivalent unidirectional composite properties based on laboratory experiments. The stiffness and
strength of the equivalent zero degree layers are required by the constitutive model. This is because the constitutive model is based on the approximation of the individual subcells as unidirectional laminated composites. The method as developed by Cheng and Littell utilizes the capability in *Mat_58 to model layered shell elements. Because the lamina fiber directions are defined by fiber angles input at the integration point level, only unidirectional properties, of an equivalent unidirectional ply, are required by the material model. The unidirectional lamina properties are not known for a braided composite and cannot be directly measured (Roberts, et al., 2002). This is due to the complex way load is transferred through a braided composite. It would be extremely difficult to achieve realistic equivalent unidirectional properties even if the fiber undulation of one direction of fiber bundles was maintained while eroding the other bundles. This discrepancy is due to the lack of resistance from the eroded bundles. Therefore, the equivalent unidirectional properties seen in Table 3.1.3.1 were back calculated from the experimental results by Littell (Littell, 2008). Further explanation of the equivalent unidirectional properties formulation can be found in Littell et al. (Littell, et al., 2008). There are several advantages for this analytical model. One advantage is that the constitutive model being used is a continuum damage mechanics model. The use of a continuum damage mechanics model is important for impact analysis where ultimate failure of the material and element deletion is desired. Another advantage is that the experimental data required for the model comes only from the triaxially braided composite samples; no additional inputs are required. In addition, the model is computationally efficient, and it can be used to simulate large components.
The shell element formulation allows each integration layer to have a weight factor and thickness. The braided material consists of [+60°/ 0°/-60°] degree fibers. The
zero degree fiber bundles are referred to as axial fiber bundles and the plus and minus sixty degree fiber bundles are referred to as bias fiber bundles. In this case all layers were assigned the same thickness but the zero degree fibers were assigned twice the weight factor of the bias fibers because the zero degree fiber bundles contain two times the amount of carbon fibers as the bias bundles. Two simulation sets were conducted in axial tension, and transverse tension. One set had all fiber bundles weighted the same and the other set had the fiber bundles weighted according to fiber count. After analyzing the outcomes of the two simulation sets, it was determined that the effects of changing the thickness of the layer were negligible for this particular configuration. This insensitivity to the weight of ply layers was the first clue that something unusual was occurring with this particular configuration and that further investigation was needed to determine the cause.

Each subcell in the unit cell is assumed to have an equal thickness with a constant fiber volume fraction. This assumption is inaccurate. In the single layer model as shown in Figure 3.1.3.1 (c), the plies in subcells ‘B’ and ‘D’ should have lower overall fiber volume fractions due to the lack of axial fibers. These subcells have only two fiber plies while subcells ‘A’ and ‘C’ have three plies. However, this assumption simplifies the process of determining the material properties which are used as inputs for the model. The unit cell can be approximated to be a symmetric composite, but each individual subcell is not a symmetric composite. However, normal-bending coupling is assumed to be negligible since the whole unit cell is a symmetric composite.

The braided composites examined here have six layers of braided fibers. The axial (0°) fibers in each layer are aligned vertically (Figure 3.1.3.2), but the lateral
position of axial fibers in each layer may be shifted right or left. As a result, the axial fibers in the six layers may not be located directly on top of each other. Typically these fibers are shifted in a random way relative to the axial fibers in layers above and below, referred to here as fiber shifting. As displayed in Figure 3.1.3.3 the nesting of the axial fiber bundles is random and varies greatly among samples. In the figure, the axial fiber bundles show up as the lighter ellipses and the darker regions are the undulating bias fibers.

![Figure 3.1.3.3: Through Thickness optical image showing nesting (Kohlman, 2012)](image)

The fiber shifting was approximated in an extreme sense for purposes of bounding the problem by shifting each layer of fibers by one subcell to the left in the full model in order to generate the finite element model for the full six layer composite (Figure 3.1.3.2).

### 3.1.4 Material Model

The method was implemented utilizing LS-DYNA® (Hallquist, 2007). The constitutive model resident within LS-DYNA®, which was used for the analysis, is *Mat_58. This model is investigated in detail earlier in this chapter.
### Table 3.1.4.1: Input Parameters

<table>
<thead>
<tr>
<th>Material Parameter Name (LS-DYNA® name)</th>
<th>Value (unit)</th>
<th>Value (unit)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Modulus (EA)</td>
<td>51.37 (GPa)</td>
<td>51.37 (GPa)</td>
</tr>
<tr>
<td>Transverse Modulus (EB)</td>
<td>25.03 (GPa)</td>
<td>25.03 (GPa)</td>
</tr>
<tr>
<td>In Plane Shear Modulus (GAB)</td>
<td>18.96 (GPa)</td>
<td>18.96 (GPa)</td>
</tr>
<tr>
<td>In Plane Poisson Ratio (PRBA)</td>
<td>0.071</td>
<td>0.071</td>
</tr>
<tr>
<td>Axial Tensile Failure Strain (E11T)</td>
<td>0.0216</td>
<td>0.0151</td>
</tr>
<tr>
<td>Axial Compressive Failure Strain (E11C)</td>
<td>0.018</td>
<td>0.01</td>
</tr>
<tr>
<td>Transverse Tensile Failure Strain (E22T)</td>
<td>0.0168</td>
<td>0.01</td>
</tr>
<tr>
<td>Transverse Compressive Failure Strain (E22C)</td>
<td>0.011</td>
<td>0.068</td>
</tr>
<tr>
<td>In Plane Shear Failure Strain (GMS)</td>
<td>0.024</td>
<td>0.02</td>
</tr>
<tr>
<td>Axial Tensile Stress at Failure (XT)</td>
<td>1044.59 (MPa)</td>
<td>607.45 (MPa)</td>
</tr>
<tr>
<td>Axial Compressive Stress at Failure (XC)</td>
<td>377.09 (MPa)</td>
<td>363.37 (MPa)</td>
</tr>
<tr>
<td>Transverse Tensile Stress at Failure (YT)</td>
<td>361.99 (MPa)</td>
<td>68.95 (MPa)</td>
</tr>
<tr>
<td>Transverse Compressive Stress at Failure (YC)</td>
<td>344.75 (MPa)</td>
<td>243.39 (MPa)</td>
</tr>
<tr>
<td>In Plane Shear Stress at Failure (SC)</td>
<td>307.31 (MPa)</td>
<td>224.09 (MPa)</td>
</tr>
<tr>
<td>Stress Limiting Parameter for Axial Tension (SLIMT1)</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Stress Limiting Parameter for Transverse Tension (SLIMT2)</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Stress Limiting Parameter for Axial Compression (SLIMC1)</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Stress Limiting Parameter for Transverse Compression (SLIMC2)</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Stress Limiting Parameter for Shear (SLIMS)</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

The inputs for the material model are listed in Table 3.1.4.1. The material inputs required by this model are unidirectional strength and stiffness properties. The full procedure for the determination of the equivalent unidirectional material properties based on the results of coupon tests on the braided composite is described in detail in Littell et al. (Littell, et al., 2008). There are five additional terms. The SLIM inputs, which are not related to mechanical properties, are known as stress limiting factors. These stress limiting factors define the amount of additional stress an integration layer can take after the ultimate strength has been reached. A SLIM of one would allow no further stress in
the defined direction after reaching ultimate strength and cause the material to act like a perfectly plastic material. SLIM values less than one simulate the material carrying a percentage of stress after hitting ultimate strength. In the model used in this paper a SLIMT2 of one is used. The SLIMT2 parameters in *Mat_58 is the stress limiting factor in the local ply transverse tension direction. This effectively simulates a perfectly plastic situation when a fiber bundle in the composite reaches ultimate tensile strength in the transverse direction.

3.1.5 Simulations

Static simulations were conducted to accurately characterize the material model. The axial tension and transverse tension simulations, seen in Figure 3.1.5.1, were developed to mimic experimental coupon tests conducted by Littell (Littell, 2008). In this figure the blue regions represent the griped region of the specimen. Only the gauge region is meshed for these simulations. The entire gauge region is meshed in order to allow for the capability to model damage initiation and failure and eventually model the damage progression seen in tests. For these quasi-static simulations, the rows of nodes at the top of the coupon simulations were fixed. Displacement in the coupon simulations was applied to the nodes at the bottom in the downward direction while motion for these nodes was constrained in the other two coordinate directions. The whole coupon specimens were modeled in order to allow for a sufficient amount of attenuation length from the fully constrained top and bottom edges to simulate the bias tearing seen in some of the coupon tests (Littell, 2008).
The dimensions of both axial tension and transverse tension simulations are 30.48 cm (12.00 in) by 3.58 cm (1.40 in) with a 20.32 cm (8.00 in) long gauge section. The axial tension simulation is composed of 480 elements, 8 across and 60 long. The transverse tension simulation is composed of 476 elements, 7 across and 68 long. The difference in these two is due to the fact that the elements are not perfectly square due to the geometry of the unit cell. The stress-strain curves for these simulations were defined by averaging the elemental stresses and strains for the bottom six rows of elements in each model. The bottom six rows are used because this section provides a large enough representative sample such that the stress-strain curve of the entire coupon is equivalent.
to the stress-strain curve of the sample but this process is not as costly as post-processing the entire coupon. The dimensions of both axial compression and transverse compression simulations are 15.24cm (6.00in) by 3.58cm (1.40in) with a 2.54cm (1.00in) long gauge section. The axial compression simulation is composed of 40 elements, 8 across and 5 long. The transverse compression simulation is composed of 35 elements, 7 across and 5 long. The stress-strain curves for these simulations were defined by averaging the elemental stresses and strains for all of elements in each model.

The materials T700/PR520 and T700/3502 were selected for this study to bound the range of resin systems. In the constitutive model, both materials are assumed to have the same elastic properties, but the overall strengths are different. Stress-strain curves comparing the baseline simulation with the experimental results are displayed in Figure 3.1.5.2, Figure 3.1.5.3, Figure 3.1.5.4, and Figure 3.1.5.5 (Littell, 2008). These show the curves for both axial and transverse tension for both the T700/PR520 material and the T700/3502 material. In the axial and transverse test for both materials, the correlation of the computed modulus to the actual modulus was relatively good. The axial tension simulation for the T700/PR520 material was linear, similar to the actual linear response of the specimen, and the axial tension simulation for the T700/PR520 material was able to correlate with the nonlinearity exhibited in the test.

The data for the transverse tension tests was not used to correlate due to the decrease in load capacity due to edge effects. These edge effects are caused by the lack of total fiber gripping. Because the axial fiber tows run horizontally, none of the axial fiber tows in the gauge section are gripped, and this is to be expected. However, there are also bias fiber tows that terminate at the free edges in the gauge section. Many of these bias fiber
bundles are un-gripped at either one end or the other and there are a few bias fiber bundles in the center of the transverse tension specimens that remain fully un-gripped on both ends (Littell, 2008). The un-gripped regions create stress concentrations in a zig-zag like pattern along the edges of the coupon specimens. These stress concentrations cause the specimens to fail at a lower stress than the material would if it were fully bounded.

Due to the edge effects seen in the transverse tension testing, it may be better to compare the transverse tension simulations to the axial tension tests. The axial tension tests can be employed for an approximate comparison due to the quasi-isotropic nature of the material. The transverse tension simulations compare relatively well to the axial tension test for both materials.

![Figure 3.1.5.2: Test vs. Simulation for T700/PR520 in Axial Tension](image)

Figure 3.1.5.2: Test vs. Simulation for T700/PR520 in Axial Tension
Figure 3.1.5.3: Test vs. Simulation for T700/PR520 in Transverse Tension

Figure 3.1.5.4: Test vs. Simulation for T700/3502 in Axial Tension
At this point damage patterns in the simulation were not capturing the realistic material response and changes in simulated ply weight yielded an unexpected insensitivity. This insensitivity was particularly concerning for the axial tension simulation case, where the axial strength should be highly sensitive to the weight of the axial fiber bundles. Therefore, in order to gain a better understanding of how the input parameters were affecting the simulation results, parametric studies were conducted. These studies were also used to aid in modifying previous assumptions and developing a method of modeling based on experimental data that will capture the differences seen in two of the triaxially braided composites tested. The damage patterns in the impact simulations conducted by Littell using this method were not predicted correctly (Littell, 2008). In order to improve on Littell’s approach the stiffness and strength of the various input properties were adjusted systematically. The two composites studied, which bound the range of material properties, were T700/PR520 and T700/3502. In order to completely understand the model and to accurately predict failure damage patterns and
ballistic limit a complete understanding of how the material inputs effect the simulation results had to be achieved.

In order to examine how each of the strength and failure strain properties affected the failure of the baseline (shifted) model, a full diagnostic was conducted. Axial and transverse tension simulations from Figure 3.1.5.1 were used in this parametric study. The five strengths were varied along with their corresponding failure strains (XT, E11T, XC, E11C, YT, E22T, YC, E22C, SC, and GMS from Table 3.1.5.1). For example, when shear strength was increased 25 percent, shear failure strain was also increased 25 percent to keep the elastic modulus constant. Both the original six layer and a new single layer models were examined with strength values varied up and down at increments of 25 percent. The strengths and failure strains were increased 25 and 50 percent, and decreased 25 and 50 percent for each variation. From this diagnostic, the primary and limiting parameters were identified. The parametric study was first conducted for the baseline simulation, which was the full six layer shifted model (Figure 3.1.4.2). However, some of the results from this study led to more questions about the shifted model and the interaction of ply level properties. In order to investigate the shifted approach more thoroughly, a simplified single layer simulation (Figure 3.1.5.6) was developed to help verify the results. The single layer model allowed for the investigation of subcell properties both individually and as a complete unit cell. There were some unanticipated conclusions from the simulation of the single layer model which warranted further parametric studies. Following this, a parametric study was conducted with both the single layer model and a six layer stacked model (Figure 3.1.5.7). The six layer
staked model was used to verify the single layer results and also compare with the previous study.

![Unit Cell](image1)

**Figure 3.1.5.6: Single Layer Through Thickness**

![Unit Cell](image2)

**Figure 3.1.5.7: Six Layer Stacked Through Thickness**

3.1.6 Results

As explained above, in order to capture the unique failure mechanisms of the triaxially braided composites some of the model assumptions needed to be modified. The model sensitivity had to be fully understood to scientifically modify these assumptions. A parametric study was conducted to describe the model sensitivity in detail. There were three sets of simulations run during the course of the parametric study, which was
conducted in order to find the mechanisms that contribute to damage patterns: the
baseline (shifted) parametric study, the single layer parametric study, and the stacked
comparison. The baseline parametric study was conducted first. Below are the stress-
strain graphs which show the effects of the changes to the input parameters in the
baseline model. During each of these studies, primary failure drivers were identified for
the different inputs for the composite materials. The primary failure driver for these
studies was defined as the uniaxial input strength which created the largest change in
global output strength and stiffness.

3.1.6.1 Baseline Parametric Study

The test data noted in the figures found in this chapter is from experimental
composite coupon tests (Roberts, et al., 2002). There has been a reduced strength and
stiffness noted in the test data for the transverse tension tests for both composites. It has
been proposed that there is a significant amount of artificial nonlinearity resulting from
ungripped bias fibers (Roberts, et al., 2002). Due to preliminary alternatively designed
coupon tests results the experimental transverse tension strength and stiffness should be
similar to the strength and stiffness of the axial tension test (Kohlman, 2012).

For this study, first the unidirectional longitudinal tensile strength and
corresponding longitudinal tensile failure strain were increased 50% and then decreased
50% for the baseline input values. Then, the unidirectional transverse tensile strength
and corresponding failure strain were increased 50% and then decreased 50%. Uniaxial
longitudinal strength and failure strain adjustments had a slight effect on the strength of
the axial tension simulation and very little effect on the transverse tension simulation
(Figure 3.1.6.1.1 & Figure 3.1.6.1.2).
Uniaxial transverse strength and failure strain adjustments had only a slight effect on the strength of the axial tension simulation (Figure 3.1.6.1.3). An increase in strength
and failure strain only had a slight effect on the transverse tension simulation, but a decrease had a significant reduction in strength (Figure 3.1.6.1.4).

Figure 3.1.6.1.3: Baseline Parametric Study T700/PR520 with Transverse Tensile Strengths Varied for the Axial Tension Simulation

Figure 3.1.6.1.4: Baseline Parametric Study T700/PR520 with Transverse Tensile Strengths Varied for the Transverse Tension Simulation
Uniaxial compression strength and failure strain variations for both the longitudinal and transverse directions had no differentiable change in the result for the axial and transverse tension simulations.

Some of the results obtained from the parametric studies were not intuitive. Uniaxial shear strength was identified as the primary driver for both axial tension specimen failure and transverse tension specimen failure for the T700/PR520 composite simulation (Figure 3.1.6.1.5 & Figure 3.1.6.1.6). The increase in shear strength and failure strain resulted in some increase in strength and the decrease resulted in a significant reduction in strength for the axial tension simulation. This was unexpected because the composite axial tensile strength should primarily be driven by the axial tensile strength of the longitudinal fibers. The cause for this anomaly is not exactly clear but may be due to the interactive way the equations controlling initiation and propagation of damage behave or the orientation of fiber bundle layers. Overall the increase in shear strength and failure strain resulted in a slight increase in strength, and the decrease resulted in a significant reduction in strength and a slight reduction in stiffness for the transverse tension simulation.
The same parametric studies were conducted for the T700/3502 composite material. Uniaxial longitudinal tension strength was identified as the primary failure driver for the axial tension specimen failure for the T700/3502 composite simulation.
(Figure 3.1.6.1.7). This was different from the T700/PR520 composite and was due to the difference in the ratio of longitudinal strength to other strengths. The adjustment of uniaxial longitudinal tension strength and failure strain affected the strength of the axial tension simulation significantly. The adjustment of uniaxial longitudinal tension strength and failure strain only somewhat changed the strength of the transverse tension simulation (Figure 3.1.6.1.7).

Figure 3.1.6.1.7: Baseline Parametric Study T700/3502 with Longitudinal Strengths Varied for the Axial Tension Simulation
The changes in the uniaxial transverse tensile strength and failure strain had no effect on the T700/3502 axial tension simulation (Figure 3.1.6.1.9). Adjustments in the uniaxial transverse tensile strength and failure strain had only a slight effect on the stiffness of the T700/3502 transverse tension simulation (Figure 3.1.6.1.10).
Uniaxial compression strength and failure strain variations for both the longitudinal and transverse directions had no differentiable change in the result for the T700/3502 axial and transverse tension simulations.
Uniaxial shear strength was identified as the primary driver for transverse tension specimen failure for the T700/3502 composite simulation (Figure 3.1.6.1.11). This was the same primary failure driver for the T700/PR520 composite material. The increase in shear strength and failure strain resulted in a similar increased stiffness, and the decrease in shear strength and failure strain resulted in a significant decrease in stiffness and a slight decrease in strength for the T700/3502 transverse tension simulation. There was only a slight effect in the stiffness of the axial tension simulation due to changes in the shear strength and failure strain (Figure 3.1.6.1.12).

Figure 3.1.6.1.11: Baseline Parametric Study T700/3502 with In-Plane Shear Strengths Varied for the Axial Tension Simulation
3.1.6.2 Baseline and Single Layer Comparison

After the baseline parametric study was completed the single layer verses baseline comparison was conducted. The single layer model (Figure 3.1.5.6) is one layer of the composite and is described in more detail in the previous section. The comparison was originally done to validate the baseline parametric study, but resulted in prompting further research due to unexpected results. Overall the single layer axial tension simulation of T700/PR520 was less stiff and weaker than the six layer simulation (Figure 3.1.6.2.1). Subcells ‘A’ and ‘C’ followed the six layer curves and subcells ‘B’ and ‘D’ were significantly weaker. The resulting unit cell average had a much lower stiffness and strength than the six ply simulations. Littell’s approach assumed equal fiber volume ratio in each of the subcells (Littell, 2008). This was valid due to the subcell shifting. However, removing shifting may invalidate this assumption. The calculation of subcell volume fraction will be addressed in the following chapter. The single layer transverse
tension simulation of T700/PR520 (Figure 3.1.6.2.2) was also less stiff than the six layer simulation. The curve of subcells ‘A’ and ‘C’ were closer to the stiffness of the six layer simulation than the average and the curves of subcells ‘B’ and ‘D’ were far weaker. The stiffnesses of the individual subcells were investigated due to the large difference in the stiffness between the shifted and un-shifted single layer simulations. These differences in the single layer simulations were attributed to the behavior of the zero degree fibers. Because of the shifted nature of the six layer model, all subcells contained the same amount of zero degree fibers and had the same ply thickness. However, in the single layer simulations this was not the case. Subcells ‘B’ and ‘D’ contained no zero degree fibers in the single layer simulations and the plies in these layers were thicker comparatively. Due to the lack of strength from zero degree fiber layers, the single layer axial tension simulation had a decreased stiffness and strength and the single layer transverse tension simulation had a decreased strength and a slight decrease in the point of nonlinearity.
The single layer axial tension simulation of T700/3502 (Figure 3.1.6.2.3) was also less stiff than the baseline simulation. Again, the curve for subcells ‘A’ and ‘C’ was in line with the six layer simulation but the curve for subcells ‘B’ and ‘D’ was much less
stiff. The single layer transverse tension simulations of T700/3502 (Figure 3.1.6.2.4) had the same stiffness but were weaker than the six layer simulations. The point where the stress-strain curve becomes non-linear is much sooner for T700/3502 than it is for T700/PR520 for all the sets of curves. This aspect proves useful when correlating with non-linear test data in Chapter IV.

Figure 3.1.6.2.3: Single Layer Comparison T700/3502 for the Axial Tension Simulation
3.1.6.3 Single Layer Parametric Study

After the differences were noted in the single layer comparison, a single layer parametric study was conducted to characterize the effects of changing the strengths on subcells with and without zero degree fiber layers. This study was performed to further investigate the results of eliminating subcell shifting in the unit cell. During the single layer parametric study, the T700/PR520 composite was analyzed first. For this analysis, the uniaxial longitudinal tension strength and failure strain were varied first. When the uniaxial longitudinal tension strength and failure strain (XT & E11T) were varied, the axial tension simulation was affected significantly while the transverse tension simulation was not. During the axial tension simulation, when the uniaxial longitudinal tension strength and failure strain were increased, the global strength increased and the global strength decreased when they were decreased (Figure 3.1.6.3.1). However, during the transverse tension simulation, there was no significant change (Figure 3.1.6.3.2).
was only a small decrease in the global strength when the uniaxial longitudinal strength and failure strain were decreased by 50 percent. This was considerably different from the shifted transverse tension simulation for this variation (Figure 3.1.6.1.2), which displayed far more sensitivity to the changes in longitudinal tensile strength.

Figure 3.1.6.3.1: Single Layer Parametric Study T700/PR520 with Longitudinal Tensile Strengths Varied for the Axial Tension Simulation
Next, the uniaxial transverse tension strength and failure strain were varied. When the uniaxial transverse tension strength and failure strain (YT & E22T) were varied, both the axial tension and the transverse tension simulations were affected. During the axial tension simulation, the curve followed a similar path to the baseline, but as the uniaxial transverse tension strength and failure strain decreased the global strength decreased and the stress at which the transition to nonlinearity occurs also decreased (Figure 3.1.6.3.3). During the transverse tension simulation, when the uniaxial transverse tension strength and failure strain increased the global strength and stiffness of the curve increased and when the uniaxial strength and failure strain decreased the global strength and stiffness decreased (Figure 3.1.6.3.4). This response is not intuitive and most likely due to the highly multiaxial stress state of the bias layers in this simulation.
Finally, the uniaxial shear strength and failure strain were varied. When the uniaxial shear strength and failure strain (SC & GMS) were decreased both the axial tension and transverse tension simulations were significantly affected. There was less change when the uniaxial shear strength and failure strain were increased. During the
axial tension simulation, when the uniaxial shear strength and failure strain were decreased the global strength decreased and when the uniaxial strength and failure strain were increased the global strength increased (Figure 3.1.6.3.5). During the transverse tension simulation, when the uniaxial shear strength and failure strain decreased the global strength and stiffness decreased and when the uniaxial strength and failure strain increased the global strength and stiffness increased (Figure 3.1.6.3.6).

Figure 3.1.6.3.5: Single Layer Parametric Study T700/PR520 with In-Plane Shear Strengths Varied for the Axial Tension Simulation
The T700/3502 composite was analyzed second for the single layer parametric study. For this analysis, the uniaxial longitudinal tension strength and failure strain were varied first. When the uniaxial longitudinal tension strength and failure strain ($X_T$ & $E_{11T}$) were increased and decreased, both the axial tension and transverse tension simulations were affected. There was some change when the uniaxial longitudinal tension strength and failure strain were increased. During both simulations when the uniaxial strength and failure strain decreased, the global strength decreased significantly (Figure 3.1.6.3.7 & Figure 3.1.6.3.8). While the T700/PR520 axial simulation had very little stiffness change for the longitudinal variation, the T700/3502 simulation had a relatively significant amount of stiffness change.
Figure 3.1.6.3.7: Single Layer Parametric Study T700/3502 with Longitudinal Tensile Strengths Varied for the Axial Tension Simulation
Next, the uniaxial transverse tension strength and failure strain were varied. When the uniaxial transverse tension strength and failure strain (YT & E22T) were varied, both axial and transverse tension simulations were affected. During the axial tension simulation when the uniaxial transverse tension strength and failure strain increased, global stiffness increased, and when the uniaxial strength and failure strain decreased the global stiffness decreased (Figure 3.1.6.3.9). During the transverse tension simulation when the uniaxial transverse tension strength and failure strain increased, global strength and stiffness increased, and when the uniaxial strength and failure strain decreased the global strength and stiffness decreased (Figure 3.1.6.3.10). The stiffness again is significantly more sensitive in this study for the T700/3502 material compared with the same study performed on the T700/PR520 material.
Finally, the uniaxial shear strength and failure strain were varied. When the uniaxial shear strength and failure strain (SC & GMS) were varied, both axial and
transverse tension simulations were affected. During the axial tension simulation, when
the uniaxial shear strength and failure strain increased the global strength increased and
when the uniaxial strength and failure strain decreased the global strength decreased
(Figure 3.1.6.3.11). During the transverse tension simulation, when the uniaxial shear
strength and failure strain increased the global strength and stiffness increased
significantly and when the uniaxial strength and failure strain decreased the global
strength and stiffness decreased significantly (Figure 3.1.6.3.12).

Figure 3.1.6.3.11: Single Layer Parametric Study T700/3502 with In-Plane Shear
Strengths Varied for the Axial Tension Simulation
Comparing all the variations of the single layer parametric study, primary failure drivers were found for the transverse tension and axial tension tests for both composites. Uniaxial shear strength was identified as the primary driver for both axial tension specimen failure and transverse tension specimen failure for the T700/PR520 composite (Figure 3.1.6.3.5 & Figure 3.1.6.3.6). These results were similar to those noted in the parametric study involving the six ply laminate. The uniaxial longitudinal strength was identified as the primary failure driver for the axial tension specimen failure for the T700/3502 composite simulation (Figure 3.1.6.3.7). The uniaxial shear strength was identified as the primary driver for transverse tension specimen failure for this composite (Figure 3.1.6.3.12). These results are also similar to those noted in the parametric study involving the six ply laminate.

There were two surprising results from this parametric study. First, there was a profound effect on the overall simulation results from adjustments of the shear
parameters. This was not expected. While it follows, from the ply architecture, that the shear properties should somewhat effect the transverse results, the effect on the axial results should have been minute. Second, there was some effect on the axial simulation results from adjustments of the transverse properties. To assist in understanding these effects and the discrepancy between the baseline and single layer results, stress-strain curves have been plotted for the individual subcells (elements) of the material (Figure 3.1.6.3.13 - Figure 3.1.6.3.16). The curves are labeled by subcell ‘A’, ‘B’, ‘C’, and ‘D’. Curves for subcells ‘A’ and ‘C’ were the same so only one curve is shown. Similarly, curves for subcells ‘B’ and ‘D’ were the same so only one curves is displayed. For the T700/PR520 material, in Figure 3.1.6.3.13 it can be seen that decreasing the transverse strength and failure strain induces a plateau into the curve for subcells ‘B’ and ‘D’, which are the subcells with no axial fiber bundles. This greatly affects the stiffness of the average curve. In Figure 3.1.6.3.14 it can be seen that the altering the shear strength had a significant effect on the strength of all the individual subcells, but only affects the stiffness in subcells ‘B’ and ‘D’. This is most likely due to the abundance of bias fiber bundles in subcells ‘B’ and ‘D’. Theoretically these bias fiber bundles should be sensitive to changes in shear due to the fiber orientation direction.
Figure 3.1.6.3.13: Effects of varying unidirectional transverse tensile strength on response of individual subcells for axial tensile simulation of T700/PR520 composite using single layer analysis model.

Figure 3.1.6.3.14: Effects of varying unidirectional in-plane shear strength on response of individual subcells for axial tensile simulation of T700/PR520 composite using single layer analysis model.

For the T700/3502 material, in Figure 3.1.6.3.15 it can be seen that decreasing the transverse strength and failure strain decreases the point of nonlinearity for subcells ‘B’ and ‘D’. This has an overall decreasing effect on the stiffness of the average curve. In
Figure 3.1.6.3.16 it can be seen that the altering the shear strength has almost no effect on the strength of the individual subcells, and has no effect on their stiffness. Because of the difference in material properties used as inputs for the material model composites T700/PR520 and T700/3502 have vastly different responses due to transverse and shear parameter adjustments. In both materials a decrease in subcells ‘B’ and ‘D’ lowers the strength in those subcells. However, in the T700/PR520 material the stiffness changes from linear to highly non-linear, but in the T700/3502 material this decrease causes only a slight change in nonlinearity.

Figure 3.1.6.3.15: Effects of varying unidirectional transverse tensile strength on response of individual subcells for axial tensile simulation of T700/3502 composite using single layer analysis model
Figure 3.1.6.3.16: Effects of varying unidirectional in-plane shear strength on response of individual subcells for axial tensile simulation of T700/3502 composite using single layer analysis model

3.1.6.4 Stacked Comparison

After the single layer parametric study, six layer non-nested simulations were run to validate the data from the single layer study. While the single layer simulations were a good first step in assessing the sensitivity of the material inputs, some of the change in sensitivity might have been due to changes in the thickness of the layers. In order to assess a set of simulations with an equivalent thickness, a six layer stacked approach was established. This approach was simply the single layer approach stacked vertically in six layers. The six layer non-shifted (stacked) model behaved very similar to the single layer model. For all four of the T700/PR520 and T700/3502 composite simulations the stress strain curves were very similar for the six layer non-nested and the single layer models.
In order to identify why the different materials had different simulation responses and different input sensitivity, a number of strength ratios were calculated. In-situ uniaxial strengths were compared in the form of tables of ratios (Table 3.1.6.4.1 & Table 3.1.6.4.2). This showed that the in situ shear and transverse strengths of the T700/PR520
composite were similar while the axial strength was much larger (Table 3.1.6.4.1). This meant that the specimens could be near failure in either the shear or transverse directions and small changes in uniaxial strength could easily tip the scales. This also helped explain why shear strength and stiffness plays such a significant role in failure in both the axial and transverse tension simulations for the T700/PR520 composite material. This was not the case for the T700/3502 composite. The transverse strength is much lower and the shear strength is much higher in comparison. The uniaxial strengths for the two different composites were also compared by ratios (Table 3.1.6.4.2). This comparison makes clear the large relative difference in the uniaxial transverse tensile strengths and the small relative difference in the uniaxial shear strengths. This relative difference is important because uniaxial shear strength appears to play a large role in the failure of both composites.

**Table 3.1.6.4.1: Composite Strength Ratios**

<table>
<thead>
<tr>
<th>Ratio</th>
<th>T700/PR520</th>
<th>T700/3502</th>
</tr>
</thead>
<tbody>
<tr>
<td>(YT/XT)</td>
<td>0.35</td>
<td>0.11</td>
</tr>
<tr>
<td>(SC/XT)</td>
<td>0.29</td>
<td>0.37</td>
</tr>
<tr>
<td>(SC/YT)</td>
<td>0.85</td>
<td>3.25</td>
</tr>
</tbody>
</table>

**Table 3.1.6.4.2: Composite Strength Ratios**

<table>
<thead>
<tr>
<th>Ratio</th>
<th>T700-3520/T700-PR520</th>
</tr>
</thead>
<tbody>
<tr>
<td>(YT)</td>
<td>0.58</td>
</tr>
<tr>
<td>(SC)</td>
<td>0.73</td>
</tr>
</tbody>
</table>
3.2 Smearing

In the single layer and stacked methods there is a clear definition between the subcells with axial fiber bundles and the subcells without axial fiber bundles. However, in the baseline method there is no definition between subcells. All the subcells appear to have the same response curves in the simulations. Due to this finding, the baseline (shifted) method was investigated in detail.

A true view of the baseline unit cell with the integration points noted by their ply orientations can be seen in Figure 3.2.1. When the ply orientations are highlighted by color the issue becomes clearer (Figure 3.2.2). From this figure it can be seen that there are exactly three zero degree plys in each subcell, six +60 degree plys in each subcell, and six -60 degree plys in each subcell.

![Unit Cell Diagram](image)

Figure 3.2.1: Through Thickness View of Unit Cell
Due to the way the elemental stresses are calculated, the subcells are essentially computed as homogenized materials. So the extreme fiber shifting (baseline) method is interpreted as smeared elements by the finite element solver. This artificial smearing is confirmed by impact analysis. Figure 3.2.3 displays the impact damage patterns for a homogenized material in a finite element analysis. The same impact shape is seen in the simulation of the extreme fiber shifting method (Figure 3.1.1.1). Impact testing and simulation will be described in further detail in Chapter VI. The actual damage shape for this impact setup is a butterfly pattern, which is seen in experimental testing (Roberts, et al., 2002). In fact, the parametric studies conducted on the baseline method had no effect on damage patterns.
A benefit of the *Mat_58 material model discussed earlier is the ability to use laminated shell theory. However, the smearing seen in the shifted model, persist even when invoking the laminated shell theory for the material systems currently under investigation (Figure 3.2.4). Because developing a material model that will adequately simulate laminated shell theory for the triaxially braided composites under study is outside of the scope of this work, an approach to calculate input properties using laminated shell theory is developed in the following chapter.
3.3 Conclusion

An effort was discussed above to characterize the material model being used to simulation triaxially braided composite materials. The material properties utilized for the analysis were determined based on experimental tests conducted on the braided composite. During this effort to characterize the sensitivity of input parameters of the material model used, steps were taken to ensure a thorough set of studies were investigated. The various material strength parameters that are input into the model have a significant effect on the overall results of the simulations, sometimes in non-intuitive ways. Through this effort, a deeper understanding of the sensitivity of the macromechanical model has been obtained. The parametric studies are a crucial first step in the definition of a quasi-empirical analytical method to capture material failure.

Accounting for extreme fiber shifting has a number of advantages and disadvantages. First, since in the extreme fiber shifting method each subcell has fifteen
plies (layers of fiber bundles), the overall fiber volume fraction within each subcell can be assumed to be constant. However, since each subcell has an equal number of $0^\circ$, $+60^\circ$ and $-60^\circ$ fibers, capturing the architecture dependent damage under impact may not be possible with the extreme fiber shifting method. Overall the original method explored above has some limitations and a method to address these limitations and improve the approach will be discussed in detail in the following chapters.
CHAPTER IV
HOMOGENIZATION ALGORITHM

4.1 Homogenization Algorithm

This chapter presents a systematic way of calculating material properties in order to simulate architecturally dependent impact damage, while maintaining computational efficiency using only experimental and manufacturer properties. This algorithm is called Independently Homogenized Subcells, and the full source code can be found in the Appendix.

4.1.1 Motivation

While *Mat_58 option ‘c’ has been shown to be the best option for these materials, a vast quantity of material parameters must be computed or assumed in order to simulate an impact event. A systematic way of calculating material properties was needed to simulate architecturally dependent impact damage, while maintaining computational efficiency using only experimental and manufacturer properties. The Independently Homogenized Subcells algorithm was developed to fill this need.

4.1.2 Approach

Previous research in modeling architecturally dependent damage in braided composites conducted by Cheng and Littell (Cheng, 2006), (Littell, 2008) was presented in the previous chapter. As was mentioned before, Littell attempted to approximate the ply shifting that has been observed in braided composites by his discretization method.
There were some difficulties implementing the discretization method, and the irregular
damage patterns of the braided composites could not be captured.

In the current study, to improve the simulation tool, a top-down approach for
determining the strength properties was merged with a bottom-up approach for
determining the stiffness properties. In order to resolve discretization issues,
independently homogenized shell elements were used, where each element is modeled as
a smeared continuum with a set of homogenized stiffness and strength properties. Each
subcell in the model is defined to have different homogenized properties, which allows
for the simulation of architecturally dependent damage. In the top-down approach, all the
information needed to characterize the material strengths was taken from the global
strengths obtained from macro-scale experimental quasi-static coupon tests. This is
necessary because, as described previously in section 3.1.3, the equivalent unidirectional
properties must be back-calculated from global tests. In the bottom-up approach, the
information needed to characterize the material stiffness was taken from micro-scale fiber
and matrix stiffness properties.

A disadvantage of this homogenization approach is the inability to model through
thickness damage. A layered shell approach that worked well with the lamination theory
described in the previous chapter would be preferable. However, as noted in Chapter III
the lamination theory option does not appear to work well for these materials. Therefore,
simulation of through thickness damage using a model based on the homogenization
approach is investigated in Chapter VIII.
To apply the analysis approach, the braided fiber architecture is idealized. As a first step in this process, a schematic of the top view of the fiber architecture is shown in Figure 4.1.2.1(b). As shown in the figure, the unit cell is divided into four parallel subcells. Next, each subcell is approximated to be a laminated composite composed of a stack of fiber tows at various orientations that are determined by the braid architecture (Figure 4.1.2.1c). Subcell A is computed in the homogenization algorithm as a [+60°/0°/-60°] composite (bottom layer listed first), and so forth for all the subcells. For subcells A and C, the fact that the 0° fiber tows have twice as many filaments per tow as the +60° and -60° layers is accounted for by making the 0° layer twice as thick as the remaining two layers. Li, et al have examined using more subcells with this type of analysis approach in order to more accurately simulate the fiber undulations which are present in the actual composite (Li, et al., 2010). However, increasing the number of subcells used to model the unit cell increases the computational cost of applying the methodology, especially when applied to realistic structural configurations. Furthermore, each subcell could be broken up into several elements, and preliminary investigation into mesh refinement has yielded equivalent results. Also, results to date have indicated that further discretization is not required in order to obtain acceptable macro scale results. Any pure matrix pockets present in the composite are not modeled explicitly, but are instead incorporated into the effective properties of each layer within the subcell, affecting the effective fiber volume ratio of each subcell. Xiao, et al developed a similar model in which the pure resin pockets are explicitly modeled as layers of pure matrix (Xiao, et al., 2011). In the future, such an approach may be attempted for this method to see if the simulation results can be improved.
Each subcell is modeled as an individual shell element in a finite element model. The unit cell thus consists of four shell elements. The assumption of using shell elements is justified since the length and width of typical structures composed of these materials
are much greater than the thickness. Impact tests conducted by Littell noted that the out-of-plane deformation in flat panel impact tests was found to be relatively small in relation to the panel dimensions (Littell, 2008). In addition, the availability of appropriate constitutive models for composites using solid elements in transient finite element codes such as LS-DYNA® are limited (Hallquist, 2007). Future efforts may involve developing an appropriate constitutive model suitable for use with solid elements. A full structure can be modeled by replicating this four element unit cell throughout the finite element mesh. This approach is advantageous in modeling large or macro-scale composite structures because the micromechanical mechanisms are accounted for even when using larger mesh sizes.

4.1.3 Material Model

The required inputs for the *Mat_58 material model are based on material axis system properties. The properties required by this model are properties of the homogenized laminate (the equivalent properties of each element). In order to characterize the composite material the Independently Homogenized Subcells method was developed. This method back calculates lamina properties from the experimental results and computes a set of homogenized properties for each of the subcells listed in Figure 4.1.2.2: subcells ‘A’, ‘B’, ‘C’, and ‘D’. The equivalent properties required by the *Mat_58 material model can be seen in Table 4.1.3.1. It should be noted that the “In Plane Poisson Ratio” required for the material model and listed in the table is $\nu_{21}$ not $\nu_{12}$. In the *Mat_58 material model there are stress limiting parameters, which allow for the material to accumulate strain even after the maximum stress is reached in addition to the inputs, shown in Table 4.1.3.1. These particular parameters will be discussed in greater
detail later. There are several advantages for using this approach. The constitutive model that was utilized is a continuum damage mechanics model, an important quality for impact analysis. Employing continuum damage mechanics enables the modeling of irreversible damage due to a monotonic evolutorial equation that involves a complex formula for pre-failure damage and ultimate failure (Kachanov, 1986). The experimental data required for characterizing strength comes solely from the triaxially braided composite samples; no additional inputs are required. In addition, the model is computationally efficient, and can be employed to simulate high speed impact and the dynamic response in large components.

<table>
<thead>
<tr>
<th>Material properties required for *Mat_58</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Modulus ($E_{11}$)</td>
</tr>
<tr>
<td>Transverse Modulus ($E_{22}$)</td>
</tr>
<tr>
<td>In Plane Shear Modulus ($G_{12}$)</td>
</tr>
<tr>
<td>In Plane Poisson Ratio ($v_{21}$)</td>
</tr>
<tr>
<td>Axial Tensile Failure Strain ($\varepsilon_{11T}$)</td>
</tr>
<tr>
<td>Axial Compressive Failure Strain ($\varepsilon_{11C}$)</td>
</tr>
<tr>
<td>Transverse Tensile Failure Strain ($\varepsilon_{22T}$)</td>
</tr>
<tr>
<td>Transverse Compressive Failure Strain ($\varepsilon_{22C}$)</td>
</tr>
<tr>
<td>In Plane Shear Failure Strain ($\varepsilon_{12}$)</td>
</tr>
<tr>
<td>Axial Tensile Stress at Failure ($\sigma_{11T}$)</td>
</tr>
<tr>
<td>Axial Compressive Stress at Failure ($\sigma_{11C}$)</td>
</tr>
<tr>
<td>Transverse Tensile Stress at Failure ($\sigma_{22T}$)</td>
</tr>
<tr>
<td>Transverse Compressive Stress at Failure ($\sigma_{22C}$)</td>
</tr>
<tr>
<td>In Plane Shear Stress at Failure ($\sigma_{12}$)</td>
</tr>
</tbody>
</table>

4.1.4 Material Properties Program

In order to accurately assign properties to the individual elements (subcells) in the finite element simulation, a quasi-empirical analytical algorithm called Independently Homogenized Subcells was developed. This algorithm was developed to calculate
material axis system inputs for *Mat_58 from the fiber and matrix stiffness properties, and the global quasi-isotropic coupon test strengths. Two sets of properties are defined. Subcells ‘A’ and ‘C’ have similar configurations only vertically inverted (Figure 4.1.2.2). Subcells ‘B’ and ‘D’ also have similar configurations only, again, vertically inverted. Therefore, only two sets of properties must be calculated. One set of mechanical properties are calculated for subcells ‘A’ and ‘C’, while another set are calculated for subcells ‘B’ and ‘D’. These mechanical properties, calculated though rigorous micromechanical and composite laminate theories were adapted to characterize the braid architecture approach developed by Cheng and Littell (Cheng, 2006), (Littell, 2008). This approach facilitates the simulation of architecturally dependent damage by defining the various subcells of the model as having different mechanical properties. The use of this type of algorithm to compute inputs for the finite element simulation allows for the systematic determination of properties.
The next six sections contain a detailed account of the inner workings of the Independently Homogenized Subcells program. In Figure 4.1.4.1, the pseudocode is presented, and each subroutine is labeled. The following paragraphs explain the algorithms of the individual subroutines employed.

4.1.4.1 Inputs

The inputs needed for the Independently Homogenized Subcells program are indicated in Figure 4.1.3.1 by green ovals. These inputs are the fiber volume ratios of the subcells, fiber and resin stiffness properties, lamina orientations, and global strengths of the composite. These are directly input into the GUI seen in appendix A.2.1.
Table 4.1.4.1.1: Constituent Properties

<table>
<thead>
<tr>
<th></th>
<th>$E_{11}$</th>
<th>$E_{22}$</th>
<th>$\nu_{12}$</th>
<th>$G_{12}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>T700 (fiber)</td>
<td>230.00</td>
<td>15.00</td>
<td>0.20</td>
<td>27.00</td>
</tr>
<tr>
<td>PR520 (resin)</td>
<td>4.00</td>
<td>4.00</td>
<td>0.36</td>
<td>1.47</td>
</tr>
<tr>
<td>E-862 (resin)</td>
<td>2.70</td>
<td>2.70</td>
<td>0.36</td>
<td>1.47</td>
</tr>
<tr>
<td>5208 (resin)</td>
<td>3.80</td>
<td>3.80</td>
<td>0.36</td>
<td>1.47</td>
</tr>
<tr>
<td>3502 (resin)</td>
<td>3.60</td>
<td>3.60</td>
<td>0.36</td>
<td>1.47</td>
</tr>
</tbody>
</table>

For this study, the average fiber volume ratio for each subcell and the fiber volume ratio for individual plies within each subcell, were determined based on high fidelity finite element models of triaxially braided composites developed by Li, et al (Li, et al., 2010). In the model developed by Li, et al, various dimensions for the unit cell and the fiber tows were taken from optical micrographs, and a finite element model with average dimensions was constructed. In order to compute the required fiber volume ratios for the Independently Homogenized Subcell program, the finite element mesh of Li, et al was virtually divided into subcells and layers within the subcell (including the fiber tow and resin rich regions) (Li, et al., 2010). The percentage of fiber and matrix within each subcell and subcell layer was then computed. Average fiber volume ratios for each subcell and the individual layers within each subcell were computed, given an assumed fiber volume fraction within the fiber tow, and knowing an overall fiber volume ratio for the composite. This was calculated assuming that the subcell layer consists of the fiber tow and pure resin regions. Other approaches have been developed by Liu and Xiao, et al. in which the unit cell geometry and dimensions, as measured by micrographs, are explicitly used to determine the required subcell and layer fiber volume ratios within the unit cell (Liu, et al., 2011) (Xiao, et al., 2011). These approaches will be applied in the future to the current method to see if improved geometry and fiber volume ratio values can be obtained.
Through these Li’s calculations based on Li’s approach, volume fractions of the axial plies in subcells ‘A’ and ‘C’ were identified as 0.80, and the volume fractions of the bias plies in all subcells were identified as 0.50 (Li, et al., 2010). The fiber and matrix constituent stiffness properties were given by the composite manufacturer (Table 4.1.4.1.1) (Roberts, et al., 2002). The lamina orientations are listed in Figure 4.1.2.1.

The global strengths are calculated in Chapter V and the data used was obtained from the quasi-static coupon test properties collected by Littell and Kohlman (Table 4.1.4.1.2) (Littell, 2008) (Kohlman, 2012).

<table>
<thead>
<tr>
<th></th>
<th>T700/PR520</th>
<th>T700/E862</th>
<th>T700/5208</th>
<th>T700/3502</th>
</tr>
</thead>
<tbody>
<tr>
<td>Global Stress at Failure (Axial Tension)</td>
<td>1048</td>
<td>800</td>
<td>696</td>
<td>634</td>
</tr>
<tr>
<td>Global Stress at Failure (Transverse Tension)</td>
<td>1048</td>
<td>800</td>
<td>696</td>
<td>634</td>
</tr>
<tr>
<td>Global Stress at Failure (Axial Compression)</td>
<td>378</td>
<td>337</td>
<td>249</td>
<td>363</td>
</tr>
<tr>
<td>Global Stress at Failure (Transverse Compression)</td>
<td>346</td>
<td>305</td>
<td>215</td>
<td>217</td>
</tr>
<tr>
<td>Stress at pt. of Nonlinearity (Axial Tension)</td>
<td>1048</td>
<td>495</td>
<td>650</td>
<td>524</td>
</tr>
<tr>
<td>Global Strain-at-Failure (Axial Compression)</td>
<td>0.019</td>
<td>0.012</td>
<td>0.005</td>
<td>0.010</td>
</tr>
<tr>
<td>Global Strain-at-Failure (Transverse Compression)</td>
<td>0.012</td>
<td>0.008</td>
<td>0.006</td>
<td>0.007</td>
</tr>
<tr>
<td>Global Strain-at-Failure (Shear)</td>
<td>0.024</td>
<td>0.020</td>
<td>0.020</td>
<td>0.014</td>
</tr>
</tbody>
</table>

4.1.4.2 Stiffness Properties

The first process was to utilize Goldberg’s Slice Model (Goldberg, et al., 2005) to calculate effective unidirectional lamina stiffness matrices for the composite laminates employed to represent each subcell. This model is a micromechanics based model developed for use in calculating effective composite properties (Goldberg, et al., 2005). This method was employed due to its simplicity and accuracy. The Slice Model calculates effective unidirectional ply properties from the fiber volume fraction, and the fiber and matrix stiffness properties. This model was employed to determine the stiffness
of a single ply within a subcell. The calculation for the stiffness of a single ply was performed for the ‘A’ and ‘C’ subcells, and the ‘B’ and ‘D’ subcells independently. The subcell volume fractions were used in addition to the fiber and matrix stiffness properties when calculating the stiffness and compliance matrices for the unidirectional laminae. The code for this subroutine can be seen in appendices A.2.4, A.2.5, and A.2.6. As can be seen in appendices A.2.4 and A.2.5 thirteen slices were used. Thirteen slices was determined optimum in an optimization study.

Classical lamination theory (CLT) (Jones, 1999) was then employed to compute the overall effective laminate stiffness matrices for each subcell. The effective stiffness matrices of the unidirectional lamina calculated in the previous process were utilized for the calculations. Only in-plane loads were assumed to be applied. Therefore, the moment curvature relations of classical laminate theory were neglected. In-plane normal-shear couplings were neglected as well, because the subcells were approximated as balanced composites. In addition, tension-bending coupling was neglected even though the composite layups were anti-symmetric, because no out-of-plane bending was observed in testing when in-plane strains were applied. In the global braided material, the composite was approximately symmetric, which resulted in the elimination of the tension-bending coupling. The composites were assumed to have unit thickness in these calculations, so the loads per unit length were approximated as stresses. From the calculated laminate stiffness matrices, the overall stiffness matrix, moduli, and Poisson’s ratio for each subcell were then computed using orthotropic elasticity theory (Jones, 1999). In the triaxially braided composite, the zero degree fiber bundles contained twice as many fibers as the bias fiber bundles. To account for this, in subcells ‘A’ and ‘C’ the
zero degree laminae were given twice the thickness of the bias laminae in the composite stiffness matrix calculations. These effective stiffness properties were then applied as the stiffness inputs for the finite element model. The calculations for this subroutine can be seen in appendix A.2.7.

4.1.4.3 Strength and Stiffness Properties - Axial

The next subroutine in the Independently Homogenized Subcells program identified the global strengths needed to back calculate the subcell strengths and then computed those properties. These properties can be seen in 4.1.4.5.1. The global strength properties were utilized to run the “Subcell Calculations”. The global stresses, the effective laminate stiffness properties from the previous step, and various uniform stress-strain assumptions between subcells were employed to calculate homogenized strengths for each subcell. Because these subcells have equivalent properties the same material was characterized for both ‘A’ and ‘C’. This was also the case for subcells ‘B’ and ‘D’. Therefore, there were only two independent materials to characterize. Separate subroutines for the “Subcell Calculations” were run for axial, transverse, and shear loading cases. The subroutine for the axial loading case can be seen in appendix A.2.8.
In order to calculate the axial tension strength in the subcells, both the stress at failure and the stress at the point where the stress-strain curve becomes nonlinear were used. For three out of the four materials, at a particular stress level, significant stiffness degradation occurred, reflected in the stress-strain curves becoming nonlinear. For example, as shown in Figure 4.1.4.3.1, for the T700/3502 material, this stiffness degradation (point of nonlinearity) occurs at a stress level of 524 MPa. The point where the stress-strain curve becomes non-linear was identified by Littell as being correlated to the stress level where there is a significant amount of bias fiber bundle splitting (local matrix microcracking) and out of plane deformation (Littell, 2008). This phenomenon was simulated in the analysis model by assuming that failure in subcells ‘B’ and ‘D’ occurred at the global stress level where the stress-strain curve became nonlinear due to
the fact that these subcells contain bias fibers only. Therefore, the stress at the point of nonlinearity was used to compute the strength for subcells ‘B’ and ‘D’. For subcells ‘A’ and ‘C’, the axial strength in the subcells was calculated employing the ultimate axial tension failure stress. This subroutine can be seen in appendices A.2.2 and A.2.3, where GSATF is the global stress at axial tension failure, and GSATS is the global stress at axial tension splitting (point of nonlinearity).

To account for the fact that the overall composite does not fail even after subcells ‘B’ and ‘D’ reach their maximum stress, the “stress limiting parameter” available in the *Mat_58 material model was used. By setting this parameter to one for the case of axial tension in subcells ‘B’ and ‘D’, the subcells could continue to accumulate strain even after the maximum stress in the subcells was reached. Catastrophic failure occurs experimentally in the axial tension test specimen only when the axial fibers break, which is represented in the current model as resulting in failure in subcells ‘A’ and ‘C’ (Littell, 2008). A visualization of how the SLIMT1 parameter changes the simulation results can be seen in Figure 4.1.4.3.2. It can be seen in the figure that the subcells ‘B’ and ‘D’ continue to elongate until the subcells ‘A’ and ‘C’ fail, resulting in all of the subcells failing at the same axial tensile strain.
The global strength from the axial tension coupon test where significant stiffness degradation occurs (point-of-nonlinearity), was used to calculate the axial tension strength in subcells ‘B’ and ‘D’. Using mechanics of materials theory and appropriate uniform stress and uniform strain assumptions, the following equations were derived to compute the partitioned subcell stress. The axial tension stress in subcells ‘A’ and ‘C’ at the point-of-nonlinearity (PON) was calculated using Equation 4.1.4.3.1. The axial tension strength in subcells ‘B’ and ‘D’ was then calculated using Equation 4.1.4.3.2.

\[
\sigma_{T11A(pon)} = \frac{\sigma_{T11(pon)}}{0.5 (1+S_{11A}/S_{11B})} \quad (4.1.4.3.1)
\]

\[
\sigma_{T11B} = \frac{S_{11A}}{S_{11B}} \sigma_{T11A(pon)} \quad (4.1.4.3.2)
\]

The axial tension failure strength of subcells ‘A’ and ‘C’ were computed using the axial tension failure stress for the overall composite and the failure strength of subcells ‘B’ and ‘D’ by using Equation 4.1.4.3.3. This equation is use because of the strength

---

Figure 4.1.4.3.2: Illustration of the SLIMT1 (Stress Limiting Parameter in Axial Tension)
correction that is needed due to enforcing the SLIM. As can be seen in Figure 4.1.4.3.3
a) the average strength output will be less than average strength input if mechanics of
materials is used to compute the strength for subcells ‘A’ and ‘C’. However, this is easily
corrected by using Equation 4.1.4.3.3 to increase the strength for subcells ‘A’ and ‘C’ so
that the average strength output is equal to the average strength input (Figure 4.1.4.3.3
b)).

\[ \sigma_{T1\alpha} = 2(\sigma_{T1\alpha} - 0.5 \sigma_{T1\beta}) \]  
\[ (4.1.4.3.3) \]

Figure 4.1.4.3.3: Illustration of the Strength Correction

The axial tension failure strain for both subcells ‘A’ and ‘C’ and subcells ‘B’ and
‘D’ were assumed to be equal to the global axial tension failure strain obtained from the
experiments, as shown in Equations 4.1.4.3.4 and 4.1.4.3.5. This can be seen the
subroutine in appendices A.2.2 and A.2.3, where GRATF is the global strain at axial tension failure.

\[ \varepsilon_{T11A} = \varepsilon_{T11} \]  \hspace{1cm} (4.1.4.3.4)

\[ \varepsilon_{T11B} = \varepsilon_{T11A} \]  \hspace{1cm} (4.1.4.3.5)

In order to calculate the axial compression strength in the subcells, Equation 4.1.4.3.6 and Equation 4.1.4.3.7 were used. The global composite axial compressive failure stress was used in the computations due to the assumption that all of the subcells failed at the global composite axial compressive failure strain. The axial compressive failure strains were set equal to the global axial compression failure strains obtained from experiments Equations 4.1.4.3.8 and 4.1.4.3.9. This subroutine can be seen in appendices A.2.2 and A.2.3, where GSACF is the global stress at axial compression failure, and GRACF is the global strain at axial compression failure.

\[ \sigma_{C11A} = \sigma_{C11} \left/ \left(0.5 \left(1 + S_{11A}/S_{11B}\right)\right) \right. \]  \hspace{1cm} (4.1.4.3.6)

\[ \sigma_{C11B} = S_{11A}/S_{11B} \sigma_{C11A} \]  \hspace{1cm} (4.1.4.3.7)

\[ \varepsilon_{C11A} = \varepsilon_{C11} \]  \hspace{1cm} (4.1.4.3.8)

\[ \varepsilon_{C11B} = \varepsilon_{C11A} \]  \hspace{1cm} (4.1.4.3.9)
4.1.4.4 Strength and Stiffness Properties – Transverse

As previously, triaxially braided composite transverse tensile tests conducted on straight sided coupons may not yield results that are truly representative of the actual material response. For this reason, the experimental testing in transverse tension is not used in this method. The calculations were instead based on the ultimate stress obtained from the axial tension coupon test. As described in Chapter III, this is a reasonable assumption for transverse strength. Using mechanics of materials theory the following equations were derived to calculate the partitioned subcell stresses. The transverse tension strengths for the individual subcells were then set equal to the global assumed transverse tension strength, as shown in Equations 4.1.4.4.1 and 4.1.4.4.2. This subroutine can be seen in appendices A.2.2 and A.2.3, where GSTTF is the global stress at transverse tension failure.

\[
\sigma_{T22A} = \sigma_{T22} \tag{4.1.4.4.1}
\]

\[
\sigma_{T22B} = \sigma_{T22} \tag{4.1.4.4.2}
\]

The transverse tension failure strain for subcells ‘B’ and ‘D’ was calculated using Equation 4.1.4.4.3. Because of the orientation of the zero degree ply fiber bundles, these bundles experience large transverse elongation during transverse tension testing due to axial fiber splitting and local matrix micro cracking (Figure 4.1.4.4.1) (Littell, 2008).
Due to these mechanisms, it is postulated that the zero degree ply bundles carry almost no load at the point where the overall composite fails in transverse tension. To incorporate this effect into the finite element simulation, the compliance matrix of the zero degree ply bundles was adjusted by setting the transverse elastic modulus of the zero degree layers in subcells ‘A’ and ‘C’ to fifty percent of the nominal value, Equation 4.1.4.4.4. Fifty percent of nominal was chosen because it was the smallest percentage that would yield realistic transverse strains. Using this value, a modified compliance matrix was assembled as shown in Equation 4.1.4.4.5. The transverse tension failure strain for subcells ‘A’ and ‘C’ was then calculated using this modified compliance matrix. This failure strain for subcells ‘A’ and ‘C’ was calculated using Equation 4.1.4.4.6. This subroutine can be seen in appendices A.2.2 and A.2.3.

\[
\varepsilon_{T22B} = S_{12B} \frac{(S_{12A} - S_{12B})}{(S_{11A} + S_{11B})} \sigma_{T22} + S_{22B} \sigma_{T22} \tag{4.1.4.4.3}
\]

\[
S_{22A} = 2 \times \left( \frac{1}{E_{22A}} \right) \tag{4.1.4.4.4}
\]

\[
[S]_{mod} = \begin{bmatrix}
\frac{1}{E_{11}} & -\nu_{12}/E_{11} & 0 \\
-\nu_{12}/E_{11} & \frac{2}{E_{22}} & 0 \\
0 & 0 & \frac{1}{G_{12}}
\end{bmatrix} \tag{4.1.4.4.5}
\]

\[
\varepsilon_{T22A} = S_{12A} \frac{(S_{12B} - S_{12A})}{(S_{11A} + S_{11B})} \sigma_{T22} + S_{22A} \sigma_{T22} \tag{4.1.4.4.6}
\]
Transverse compression for the subcells were assigned to the subcells in the same way as was done for the transverse tension strength, with the subcell strengths being set equal to the global composite failure stress. This is shown in Equations 4.1.4.4.7 and 4.1.4.4.8. This subroutine can be seen in appendices A.2.2 and A.2.3, where GSTCF is the global stress at transverse compression failure.

\[
\sigma_{C22A} = \sigma_{C22} \quad (4.1.4.7)
\]

\[
\sigma_{C22B} = \sigma_{C22} \quad (4.1.4.8)
\]

The subcell failure strains were calculated as follows. For the case of transverse compression, examinations of the optical strain data obtained during tests conducted by Littell indicated that the strain field was uniform throughout the gage section (Littell, 2008). Because the strain was uniform throughout, all four of the subcells were assumed to fail simultaneously. Because of the highly nonlinear characteristics of the experimental transverse compression curves, the subcell failure strains of these curves were computed based on the experimental global composite failure strain data, specifically the values indicated in Table 4.1.4.1.1. The transverse compression failure strains for each subcell were computed by multiplying the global composite failure strains by a scale factor determined by applying uniform strain and stress assumptions and mechanics of materials theory. In these calculations the actual ratios of the failure strains in the subcells are assumed to be equivalent to the ratios of the failure strains in
each subcell computed using the failure stresses and elasticity theory. This can be seen in Equations 4.1.4.4.9-4.1.4.4.12.

\[
\varepsilon_{22b} = S_{12B} \frac{(S_{12A} - S_{12B})}{(S_{11A} + S_{11B})} + S_{22B} \quad (4.1.4.4.9)
\]

\[
\varepsilon_{22a} = S_{12A} \frac{(S_{12B} - S_{12A})}{(S_{11A} + S_{11B})} + S_{22A} \quad (4.1.4.4.10)
\]

\[
\varepsilon_{C22A} = 2 \frac{\varepsilon_{22b}}{\varepsilon_{22a} + \varepsilon_{22b}} \varepsilon_{C22} \quad (4.1.4.4.11)
\]

\[
\varepsilon_{C22B} = 2 \frac{\varepsilon_{22a}}{\varepsilon_{22a} + \varepsilon_{22b}} \varepsilon_{C22} \quad (4.1.4.4.12)
\]

4.1.4.5 Strength and Stiffness Properties – Shear

Shear strength for the subcells were assigned to the subcells in the same way as was done for the transverse tension strength, with the subcell strengths being set equal to the global composite failure stress. This is shown in Equations 4.1.4.5.1 and 4.1.4.5.2. This subroutine can be seen in appendices A.2.2 and A.2.3, where GSSF is the global stress at shear failure. The global shear stress values used were collected by Littell (Littell, 2008).

\[
\sigma_{12A} = \sigma_{12} \quad (4.1.4.5.1)
\]

\[
\sigma_{12B} = \sigma_{12} \quad (4.1.4.5.2)
\]
The subcell failure strains were calculated similarly to the transverse compression strains. In these calculations the actual ratios of the failure strains in the subcells are assumed to be equivalent to the ratios of the failure strains in each subcell computed using the failure stresses and elasticity theory. This can be seen in Equations 4.1.4.5.3 and 4.1.4.5.4.

\[
\varepsilon_{12A} = 2 \frac{S_{66B}}{S_{66A} + S_{66B}} \varepsilon_{12} \quad (4.1.4.5.3)
\]

\[
\varepsilon_{12B} = 2 \frac{S_{66A}}{S_{66A} + S_{66B}} \varepsilon_{12} \quad (4.1.4.5.4)
\]

Table 4.1.4.5.1: Material property values used for the quasi-static coupon simulations

<table>
<thead>
<tr>
<th>Material Parameter Name</th>
<th>T700 – PR520</th>
<th>T700 – E862</th>
<th>T700 – 5208</th>
<th>T700 – 3502</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Modulus ((E_{11}))</td>
<td>79600</td>
<td>8600</td>
<td>78300</td>
<td>6400</td>
</tr>
<tr>
<td>Transverse Modulus ((E_{22}))</td>
<td>38400</td>
<td>38000</td>
<td>36800</td>
<td>31800</td>
</tr>
<tr>
<td>In Plane Shear Modulus ((G_{12}))</td>
<td>14500</td>
<td>23600</td>
<td>13500</td>
<td>23100</td>
</tr>
<tr>
<td>In Plane Poisson Ratio ((\nu_{21}))</td>
<td>0.15</td>
<td>1.43</td>
<td>0.15</td>
<td>1.63</td>
</tr>
<tr>
<td>Axial Tensile Failure Strain ((\varepsilon_{11T}))</td>
<td>0.021</td>
<td>0.021</td>
<td>0.018</td>
<td>0.018</td>
</tr>
<tr>
<td>Axial Compressive Failure Strain ((\varepsilon_{11C}))</td>
<td>0.016</td>
<td>0.016</td>
<td>0.012</td>
<td>0.012</td>
</tr>
<tr>
<td>Transverse Tensile Failure Strain ((\varepsilon_{22T}))</td>
<td>0.053</td>
<td>0.016</td>
<td>0.044</td>
<td>0.014</td>
</tr>
<tr>
<td>Transverse Compressive Failure Strain ((\varepsilon_{22C}))</td>
<td>0.014</td>
<td>0.008</td>
<td>0.011</td>
<td>0.007</td>
</tr>
<tr>
<td>In Plane Shear Failure Strain ((\varepsilon_{12}))</td>
<td>0.030</td>
<td>0.018</td>
<td>0.025</td>
<td>0.015</td>
</tr>
<tr>
<td>Axial Tensile Stress at Failure ((\sigma_{11T}))</td>
<td>1793</td>
<td>193</td>
<td>1480</td>
<td>120</td>
</tr>
<tr>
<td>Axial Compressive Stress at Failure ((\sigma_{11C}))</td>
<td>713</td>
<td>77</td>
<td>604</td>
<td>49</td>
</tr>
<tr>
<td>Transverse Tensile Stress at Failure ((\sigma_{22T}))</td>
<td>993</td>
<td>993</td>
<td>800</td>
<td>800</td>
</tr>
<tr>
<td>Transverse Compressive Stress at Failure ((\sigma_{22C}))</td>
<td>323</td>
<td>323</td>
<td>303</td>
<td>303</td>
</tr>
<tr>
<td>In Plane Shear Stress at Failure ((\sigma_{12}))</td>
<td>308</td>
<td>308</td>
<td>257</td>
<td>257</td>
</tr>
</tbody>
</table>
The output of the Independently Homogenized Subcells program can be seen in Table 4.1.3.1. This table lists all the input parameters required by the *Mat_58 material model for all four materials under investigation. The Independently Homogenized Subcells program is a rigorous and repeatable approach. This method was developed based on accepted theoretical algorithms and methods, and can easily be adapted to calculate material inputs for any woven or braided composite.

4.1.5 Conclusion

An in-depth analysis was required to characterize the impact response of carbon fiber triaxially braided composite turbo jet engine cases. The analytical tool Independently Homogenized Subcells was designed to meet this need. This more accurate and versatile numerical simulation tool helps to better analyze the effect of dominant composite properties on impact deformation and failure in composite structures. Composites are required to resist ballistic impact loading to be used as protective structures. It was necessary to develop a simulation tool that systematically calculates input parameters to accurately simulate both ballistic limit and damage patterns. In order to capture the complex composite material systems, a top-down approach was merged with a bottom-up approach for simulating the ballistic limit and damage patterns in textile composites. The top-down portion used global strengths obtained from macro-scale experimental quasi-static coupon tests to characterize the material strengths. The bottom-up portion used micro-scale fiber and matrix stiffness properties to characterize the material stiffness. Using the combination of the two portions, the dominate micromechanical mechanisms were captured without having to individually identify and characterize them. The four composites analyzed were
correlated and compared with coupon level properties. The model was then expanded to account for composite material strain rate sensitivity, which was necessary due to the high rate of loading involved in ballistic impact. This is an advantageous approach for modeling large or macro-scale composite structures because the micromechanical mechanisms are accounted for even when using larger mesh sizes. The systematic calculation methodology rendered in this paper is applicable to any braded or woven composite.
CHAPTER V
COUPON CORRELATION

5.1 Statistics

Statistics were conducted on the experimental ASTM standard coupon results in order to obtain representative curves for simulation correlation. The experimental data utilized for statistical analysis was obtained by Littell and Kohlman (Littell, 2008), (Kohlman, 2012). Because the approach presented in Chapter IV uses experimental coupon test data to correlate appropriate material characteristics for simulation, the material model characteristics used must be as representative of the experimental material response as possible. By conducting statistics on the experimental results, not only are more representative material model characteristics defined, but also the tolerances for error in both the coupon simulations and the impact simulations can be established. The experimental coupon test dimensions can be seen in Figure 5.1.1. The blue portions of the figure represent the gripped regions for the various tests, and the orange portions represent the gauge regions. The four different types of tests, axial tension, transverse tension, axial compression, and transverse compression, are used to correlate material behaviors for simulation (Littell, 2008) (Kohlman, 2012). In the axial tests, the axial fiber tows run vertically throughout the specimens, and the bias fiber tows undulate through the axial fibers at angles of +60 degrees and -60 degrees. In the transverse tests the axial fiber tows run horizontally, and the bias fiber tows are rotated 90 degrees from vertical.
Statistics were conducted on the results for axial tension, transverse tension, axial compression, and transverse compression tests by calculating the standard deviations of the ultimate stresses and using the resulting percent of error to linearly determine the height of the error bars (starting at zero for no stress and incrementing linearly to the standard deviation of the ultimate stress at the ultimate stress). Linearizing the standard deviation, as described above, is an approximation. The standard deviation would realistically be different for each discrete point. However, because the experimental data has a varying distance between discrete points and the points themselves are different for each test, a linearly increasing error was assumed in order to conduct the analysis.
efficiently. The standard deviation equation used can be seen below in Equation 5.1.1 where \(n\) refers to the number of values analyzed and \(x\) refers to the individual values.

The results of a representative example standard deviation calculation can be seen in Table 5.1.1. In order to average the curves, discrete points were chosen at every 0.0005 increments of strain. The arithmetic mean of the discrete points is used to compute the average curve. Stress values for each experimental test were then determined via interpolation from the points bounding each identified strain increment. This interpolation is demonstrated in Table 5.1.2. A representative example of raw test data can be seen in Figure 5.1.2. The discretization can be seen in the following charts included in this chapter.

\[
Stdev = \frac{\sqrt{n \cdot \sum_{x=1}^{n} x - (\sum_{x=1}^{n} x)^2}}{n(n-1)} \quad (5.1.1)
\]

\[
Percentage = \frac{Stdev}{Mean} \quad (5.1.2)
\]

The percentage represented as error bars in the following charts can be seen in the example in Figure 5.1.3. This percentage is calculated by dividing the standard deviation by the mean for the ultimate stress, Equation 5.1.2. Using this method the standard deviation can be visualized as a percentage of potential error in the experimental results. Typically three tests were conducted for each type of test on each material. If only two test are conducted, as is seen in Figure 5.1.2, the theoretical error is higher and conversely if four tests are conducted the error is lower.
Table 5.1.1: Mean, Standard Deviation, and Percentage Results

<table>
<thead>
<tr>
<th>Specimen Name</th>
<th>Braid Orientation</th>
<th>Min Stress (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>TC01</td>
<td>Transverse</td>
<td>-166.5</td>
</tr>
<tr>
<td>TC02</td>
<td>Transverse</td>
<td>-215.1</td>
</tr>
<tr>
<td>Mean</td>
<td></td>
<td>-190.8</td>
</tr>
<tr>
<td>Stdev</td>
<td></td>
<td>34.4</td>
</tr>
<tr>
<td>Percentage</td>
<td></td>
<td>18%</td>
</tr>
</tbody>
</table>

Figure 5.1.2: Raw Experimental Data from 2 Tests
Table 5.1.2: Interpolated Values at Strain Increments

<table>
<thead>
<tr>
<th>ΔY/Y (in/in)</th>
<th>T01</th>
<th>T02</th>
<th>Avg. Stress (Mpa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.00050</td>
<td>23.90</td>
<td>9.59</td>
<td>16.74</td>
</tr>
<tr>
<td>0.00100</td>
<td>48.77</td>
<td>23.88</td>
<td>36.33</td>
</tr>
<tr>
<td>0.00150</td>
<td>70.71</td>
<td>49.72</td>
<td>60.21</td>
</tr>
<tr>
<td>0.00200</td>
<td>82.27</td>
<td>71.64</td>
<td>76.95</td>
</tr>
<tr>
<td>0.00250</td>
<td>113.80</td>
<td>82.21</td>
<td>98.01</td>
</tr>
<tr>
<td>0.00300</td>
<td>123.40</td>
<td>102.17</td>
<td>112.79</td>
</tr>
<tr>
<td>0.00350</td>
<td>150.24</td>
<td>120.38</td>
<td>135.31</td>
</tr>
<tr>
<td>0.00400</td>
<td>166.49</td>
<td>139.56</td>
<td>153.03</td>
</tr>
<tr>
<td>0.00450</td>
<td>166.49</td>
<td>158.74</td>
<td>162.62</td>
</tr>
<tr>
<td>0.00500</td>
<td>166.49</td>
<td>176.75</td>
<td>171.62</td>
</tr>
</tbody>
</table>

The materials will be discussed in order from the toughened resin to brittle resin. The first material is T700/PR520. The average of the axial tension coupon tests is shown in Figure 5.1.4. This curve has a maximum strength of 993 MPa and a maximum strain of 0.0205. The standard deviation of the four tests conducted is 60.8 MPa at ultimate...
stress or 6%. The average of the transverse tension coupon tests is shown in Figure 5.1.5. This curve has a maximum strength of 599 MPa and a maximum strain of 0.0164. The standard deviation of the three tests conducted is 24.8 MPa at ultimate stress or 4%. The average stress-strain curve from the axial compression tests can be seen in Figure 5.1.6. This curve has a maximum strength of 395 MPa and a maximum strain of 0.016. The standard deviation of the three tests conducted is 9.09 MPa at ultimate stress or 2%. The average curve from the transverse compression tests can be seen in Figure 5.1.7. This curve has a maximum strength of 323 MPa and a maximum strain of 0.011. The standard deviation of the three tests conducted is 12.1 MPa at ultimate stress or 3%. The summery of these values is tabulated in Table 5.1.3.

<table>
<thead>
<tr>
<th>T700/PR520</th>
<th>AT</th>
<th>TT</th>
<th>AC</th>
<th>TC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stress (Mpa)</td>
<td>993</td>
<td>599</td>
<td>395</td>
<td>323</td>
</tr>
<tr>
<td>Strain</td>
<td>0.0205</td>
<td>0.0164</td>
<td>0.016</td>
<td>0.011</td>
</tr>
<tr>
<td>StaDev (Mpa)</td>
<td>60.8</td>
<td>24.8</td>
<td>9.09</td>
<td>12.1</td>
</tr>
<tr>
<td># of Tests</td>
<td>4</td>
<td>3</td>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td>Error</td>
<td>6%</td>
<td>4%</td>
<td>2%</td>
<td>3%</td>
</tr>
</tbody>
</table>
Figure 5.1.4: Average Curve for T700/PR520 Axial Tension Test with Error Bars representing Standard Deviation
Figure 5.1.5: Average Curve for T700/PR520 Transverse Tension Test with Error
Bars representing Standard Deviation

[Graph showing the relationship between Stress (MPa) and Strain.]
Figure 5.1.6: Average Curve for T700/PR520 Axial Compression Test with Error Bars representing Standard Deviation
The second material which was analyzed is T700/E862. The average of the axial tension coupon tests is shown in Figure 5.1.8. This curve has a maximum strength of 800 MPa and a maximum strain of 0.018. The standard deviation of the four tests conducted is 0.70 MPa at ultimate stress and less than 1%. The average of the transverse tension coupon tests is shown in Figure 5.1.9. This curve has a maximum strength of 430 MPa and a maximum strain of 0.014. The standard deviation of the five tests conducted is 38.5 MPa at ultimate stress and 9%. The average stress-strain curve from the axial compression tests can be seen in Figure 5.1.10. This curve has a maximum strength of 327 MPa and a maximum strain of 0.012. The standard deviation of the three tests conducted is 13.0 MPa at ultimate stress or 4%. The average curve from the transverse
compression tests can be seen in Figure 5.1.11. This curve has a maximum strength of 303 MPa and a maximum strain of 0.009. The standard deviation of the three tests conducted is 4.0 MPa at ultimate stress or 1%. The summery of these values is tabulated in Table 5.1.4.

<table>
<thead>
<tr>
<th>T700/E862</th>
<th>AT</th>
<th>TT</th>
<th>AC</th>
<th>TC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stress (Mpa)</td>
<td>800</td>
<td>430</td>
<td>327</td>
<td>303</td>
</tr>
<tr>
<td>Strain</td>
<td>0.018</td>
<td>0.014</td>
<td>0.012</td>
<td>0.009</td>
</tr>
<tr>
<td>StaDev (Mpa)</td>
<td>0.7</td>
<td>38.5</td>
<td>13</td>
<td>4</td>
</tr>
<tr>
<td># of Tests</td>
<td>4</td>
<td>5</td>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td>Error</td>
<td>1%</td>
<td>9%</td>
<td>4%</td>
<td>1%</td>
</tr>
</tbody>
</table>

Figure 5.1.8: Average Curve for T700/E862 Axial Tension Test with Error Bars representing Standard Deviation
Figure 5.1.9: Average Curve for T700/E862 Transverse Tension Test with Error Bars representing Standard Deviation
Figure 5.1.10: Average Curve for T700/E862 Axial Compression Test with Error Bars representing Standard Deviation
The final material is T700/5208. The average of the axial tension coupon tests is shown in Figure 5.1.12. This curve has a maximum strength of 689 MPa and a maximum strain of 0.016. The standard deviation of the five tests conducted is 53.8 MPa at ultimate stress or 8%. The average of the transverse tension coupon tests is shown in Figure 5.1.13. This curve has a maximum strength of 326 MPa and a maximum strain of 0.0095. The standard deviation of the five tests conducted is 18.0 MPa at ultimate stress or 6%. The average stress-strain curve from the axial compression tests can be seen in Figure 5.1.14. This curve has a maximum strength of 251 MPa and a maximum strain of 0.0065. The standard deviation of the two tests conducted is 16.4 MPa at ultimate stress or 7%. The average curve from the transverse compression tests can be seen in Figure

Figure 5.1.11: Average Curve for T700/E862 Transverse Compression Test with Error Bars representing Standard Deviation
5.1.15. This curve has a maximum strength of 172 MPa and a maximum strain of 0.005.

The standard deviation of the two tests conducted is 34.4 MPa at ultimate stress or 18%.

The summary of these values is tabulated in Table 5.1.5.

<table>
<thead>
<tr>
<th>T700/5208</th>
<th>AT</th>
<th>TT</th>
<th>AC</th>
<th>TC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stress (Mpa)</td>
<td>689</td>
<td>326</td>
<td>251</td>
<td>172</td>
</tr>
<tr>
<td>Strain</td>
<td>0.016</td>
<td>0.0095</td>
<td>0.0065</td>
<td>0.005</td>
</tr>
<tr>
<td>StaDev (Mpa)</td>
<td>53.8</td>
<td>18</td>
<td>16.4</td>
<td>34.4</td>
</tr>
<tr>
<td># of Tests</td>
<td>5</td>
<td>5</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>Error</td>
<td>8%</td>
<td>6%</td>
<td>7%</td>
<td>18%</td>
</tr>
</tbody>
</table>

Figure 5.1.12: Average Curve for T700/5208 Axial Tension Test with Error Bars representing Standard Deviation
Figure 5.1.13: Average Curve for T700/5208 Transverse Tension Test with Error Bars representing Standard Deviation
Figure 5.1.14: Average Curve for T700/5208 Axial Compression Test with Error Bars representing Standard Deviation
Multiple T700/3502 material tests were not available for analysis. Therefore, statistics on this material were not performed. However, a representative test was used for the tension loading cases to correlate material properties.

The analyzed experimental data was then drawn from to develop inputs for homogenization algorithm discussed in Chapter IV. Each of the axial tension, axial compression, and transverse tension cases was drawn from to identify the median global stress and strain at failure, Table 5.1.6. In addition a global stress at the point of nonlinearity was identified for the axial tension test. This was done to somewhat supplement the lack of reliable transverse tension test data, and is discussed in detail in Chapter IV.
Table 5.1.6: Inputs for Homogenization Algorithm

<table>
<thead>
<tr>
<th></th>
<th>PRS20</th>
<th>E862</th>
<th>S208</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Data from Axial Tension Test</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Global Stress at Failure</td>
<td>990</td>
<td>800</td>
<td>691</td>
</tr>
<tr>
<td>Global Strain at Failure</td>
<td>0.0205</td>
<td>0.018</td>
<td>0.016</td>
</tr>
<tr>
<td>Global Stress at pt. of Nonlinearity</td>
<td>990</td>
<td>793</td>
<td>657</td>
</tr>
<tr>
<td><strong>Data from Axial Compression Test</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Global Stress at Failure</td>
<td>395</td>
<td>327</td>
<td>251</td>
</tr>
<tr>
<td>Global Strain at Failure</td>
<td>0.016</td>
<td>0.012</td>
<td>0.0065</td>
</tr>
<tr>
<td><strong>Data from Transverse Compression Test</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Global Stress at Failure</td>
<td>323</td>
<td>303</td>
<td>172</td>
</tr>
<tr>
<td>Global Strain at Failure</td>
<td>0.011</td>
<td>0.009</td>
<td>0.005</td>
</tr>
<tr>
<td><strong>Data from Shear Test</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Global Stress at Failure</td>
<td>308</td>
<td>257</td>
<td>308</td>
</tr>
<tr>
<td>Global Strain at Failure</td>
<td>0.024</td>
<td>0.020</td>
<td>0.020</td>
</tr>
</tbody>
</table>

The data for the transverse tension tests was not used to correlate due to the decrease in load capacity due to edge effects. These edge effects are fully described in Chapter III. Due to this edge effect issue a new type of specimen was developed by Kohlman to mitigate the amount of edge effects by extending the gauge region horizontally (Kohlman, 2012). A schematic of this specimen can be seen in Figure 5.1.16. In this way, Kohlman was able to constrain more of the bias fiber bundles in the transverse tension coupon tests. Therefore the ultimate stress for the notched transverse tension test should be closer to the in-situ transverse tension strength. However, the test is not uniaxial, so the stiffness and therefore the stress-strain curve cannot be used to calibrate the simulations. In the future this test may be used in conjunction with a traditional coupon test to obtain more realist in-situ understanding of the material. Statistics were computed on the notched test similar to those computed for the other test.
described above, Table 5.1.7. Only the T700/PR520 material has notched specimen data available at this time.

<table>
<thead>
<tr>
<th>Specimen Name</th>
<th>Max Stress (MPa)</th>
<th>Max Axial Strain (in/in)</th>
<th>E (GPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Test1</td>
<td>730.8</td>
<td>0.0149</td>
<td>60</td>
</tr>
<tr>
<td>Test2</td>
<td>744.4</td>
<td>0.0142</td>
<td>63.54</td>
</tr>
<tr>
<td>Test3</td>
<td>691.7</td>
<td>0.0163</td>
<td>61.45</td>
</tr>
<tr>
<td>Mean</td>
<td>722.3</td>
<td>0.0151</td>
<td>61.66</td>
</tr>
<tr>
<td>Standard Deviation</td>
<td>27.3</td>
<td>0.0011</td>
<td>1.78</td>
</tr>
<tr>
<td>Percentage</td>
<td>4%</td>
<td>7%</td>
<td>3%</td>
</tr>
</tbody>
</table>
5.2 Correlation

Static coupon simulations were conducted to check the correlation of the computed stiffness and strength values. The axial tension, transverse tension, axial compression, and transverse compression simulations were conducted in the same way as those described in Chapter III.

In addition to these four simulations, shear simulations were performed on the four composites under study in order to qualitatively compare the different materials.
The shear simulations did not use the coupon schematic from the experimental testing conducted by Littell (Littell, 2008). It was not feasible to model the bowtie specimen with four node shell elements, due to the diagonal cuts in the specimen. Therefore, a small rectangular section was used for the simulations (Figure 5.2.2). The bottom nodes of this section were fixed in all six directions, and the top nodes were fixed in two directions and then given a displacement in the direction of the arrow in Figure 5.2.2. Because the model is made up of shell elements, there is a lack of rigidity in the in-plane shear direction and these simulations will likely fail prematurely. However, these simulations will be used to compare the different shear properties of the materials qualitatively.

These coupon simulations were used to correlate the Independently Homogenized Subcells program, explained in detail in the previous chapter. The program used the experimental data to compute equivalent homogenized subcell properties which are used as inputs for the finite element material model. These inputs for each of the material systems examined in the study can be seen below in Table 5.2.1.
### Table 5.2.1: Inputs for the Material Model

<table>
<thead>
<tr>
<th>Inputs for Micromechanics Approach</th>
<th>PR520</th>
<th>E862</th>
<th>S208</th>
</tr>
</thead>
<tbody>
<tr>
<td>AC</td>
<td>BD</td>
<td>AC</td>
<td>BD</td>
</tr>
<tr>
<td>MID</td>
<td>1</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>RO</td>
<td>0.000167</td>
<td>0.000167</td>
<td>0.000167</td>
</tr>
<tr>
<td>EA</td>
<td>81235</td>
<td>8567</td>
<td>79711</td>
</tr>
<tr>
<td>EB</td>
<td>47861</td>
<td>38032</td>
<td>46197</td>
</tr>
<tr>
<td>GAB</td>
<td>14998</td>
<td>22993</td>
<td>14748</td>
</tr>
<tr>
<td>PRBA</td>
<td>0.178</td>
<td>1.427</td>
<td>0.181</td>
</tr>
<tr>
<td>E11C</td>
<td>0.016</td>
<td>0.016</td>
<td>0.012</td>
</tr>
<tr>
<td>XC</td>
<td>715</td>
<td>75</td>
<td>605</td>
</tr>
<tr>
<td>E11T</td>
<td>0.021</td>
<td>0.021</td>
<td>0.018</td>
</tr>
<tr>
<td>XT</td>
<td>1796</td>
<td>189</td>
<td>1482</td>
</tr>
<tr>
<td>E22C</td>
<td>0.013</td>
<td>0.009</td>
<td>0.010</td>
</tr>
<tr>
<td>YC</td>
<td>323</td>
<td>323</td>
<td>303</td>
</tr>
<tr>
<td>E22T</td>
<td>0.042</td>
<td>0.016</td>
<td>0.036</td>
</tr>
<tr>
<td>YT</td>
<td>993</td>
<td>993</td>
<td>800</td>
</tr>
<tr>
<td>GMS</td>
<td>0.029</td>
<td>0.019</td>
<td>0.029</td>
</tr>
<tr>
<td>SC</td>
<td>308</td>
<td>308</td>
<td>257</td>
</tr>
<tr>
<td>SLIMT1</td>
<td>0</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>SLIMT2</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>SLIMC1</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>SLIMC2</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>SLIMS</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

The Independently Homogenized Subcells program is calibrated such that the simulations conducted, using the material inputs calculated, will represent the triaxially braided composite at the macro level. In order to calibrate the Independently Homogenized Subcells program, simulations of coupon specimens were compared with experimental data from quasi-static coupon tests conducted by Littell (Littell, 2008). Figures 5.2.2–5.2.17 display comparisons of the axial tension, transverse tension, axial compression, transverse compression, and shear coupon simulation results (red) with the experimental straight sided coupon stress-strain curves (blue) for the T700/PR520,
T700/E862, and T700/5208 materials. Error bars to represent the percent of error in the experimental curves, and a shaded light blue area indicating the region in which failure occurred are included in the following figures. Figure 5.2.2 contains the axial tension curves for the composite material T700/PR520. The computed axial tension values for the T700/PR520 composite compare well with the experimental axial tension test curve. Overall the strength of the simulation is very close to the experimental strength and the initial stiffness of the simulation is within the error bars. The simulation is also linear until failure. The experimental curve has some artificial stiffening beginning at around 700 MPa due to the way the statistics are computed. For these particular experiments, the tests that had a higher strength also exhibited a higher stiffness, therefore the average stiffness increases at the point when the weaker tests start failing. This is the primary cause for the difference between the curves near failure. Figure 5.2.3 contains the transverse tension curves for the composite material T700/P5520. The computed transverse tension values for the T700/PR520 composite do not compare quite as well with the experimental transverse tension test curve. However, the stiffness of the simulation is much greater than the experimental straight sided stiffness because the transverse tension simulations are not used to compute the properties used in the simulation. As discussed previously the low strength observed is due to the edge effects noted in the straight sided coupon testing (Littell, 2008). These effects are not explicitly modeled in the simulation. In order to get a more accurate in-situ stiffness response, the T700/PR520 material has also been testing using a notched specimen test (Kohlman, 2012). For comparison purposes the average failure stress resulting from the notched test (green) can be seen in Figure 5.2.4. Theoretically the in-situ material strength should be
slightly greater than the resulting strength from the notched test. Therefore, the simulation stiffness in transverse tension is reasonable.

Figure 5.2.2: Axial Tension Experimental vs. Computed T700/PR520 Composite Coupon Test

Figure 5.2.3: Transverse Tension Experimental vs. Computed T700/PR520 Composite Coupon Test
Simulations of the compression coupon tests were also conducted for the T700/PR520 material. Figure 5.2.5 contains the axial compression curves for the composite material T700/PR520. The computed axial compression values for the T700/PR520 composite compare well with the experimental axial compression curve. Overall, the stiffness of the simulation compares very closely to the experimental stiffness and the simulated failure occurs inside the experimental failure region. Figure 5.2.6 contains the transverse compression curves for the composite material T700/PR520. The computed transverse compression values for the T700/PR520 composite do not compare quite as well with the experimental transverse compression simulation. The simulated failure does occur inside the failure region and the failure strain is within the error range. However, the stiffness of the simulation is slightly greater than the experimental straight sided stiffness before failure.
Figure 5.2.5: Axial Compression Experimental vs. Computed T700/PR520 Composite Coupon Test

Figure 5.2.6: Transverse Compression Experimental vs. Computed T700/PR520 Composite Coupon Test
Figure 5.2.7 contains the shear curves for the composite material T700/PR520. The computed shear values for the T700/PR520 are much stiffer and weaker than the properties input into the material model. However, as stated above, these shear simulations are not fully representative of a global shear response and are only a means of comparing the shear properties of the different composites qualitatively.

The material T700/E862 was also used for calibration. Figures 5.2.8-5.2.12 display comparisons of the coupon simulation results (red) with the experimental coupon stress-strain curves (blue). Error bars to represent the percent of error in the experimental curves, and a shaded light blue area indicating the region in which failure occurred are included in the following figures. Figure 5.2.8 contains the axial tension curves for the composite material T700/E862. The computed axial tension values for the T700/E862 composite compare somewhat well with the experimental axial tension test curve. The
strength for the simulation is close to the strength in the experimental curve. The simulation stiffness is slightly less than the experimental stiffness and the simulated failure occurs slightly past the failure region. An alternate method for calculated the statistics for the point of nonlinearity may be required to correlate better with the stiffness. However, the experimental curves for this case have a tight margin so the simulated results are reasonable. Figure 5.2.9 contains transverse tension curves for the composite material T700/E862. The computed transverse tension values for the T700/E862 composite do not compare as well with the experimental transverse tension test curve. The simulated failure occurs outside of the experimental failure region and the simulation stiffness is much greater than the experimental straight sided stiffness. Again this is most likely due to the edge effects noted in the straight sided coupon testing (Littell, 2008). Notched specimen testing was not available for this material.

![Stress vs. Strain Diagram](image)

Figure 5.2.8: Axial Tension Experimental vs. Computed T700/E862 Composite Coupon Test
Simulations of the compression coupon tests were also conducted for the T700/E862 material. Figure 5.2.10 contains the axial compression curves for the composite material T700/E862. The computed axial compression values for the T700/E862 composite compare well with the experimental axial tension test curve. Overall, the simulation stiffness compares very closely to the experimental stiffness and the simulated failure occurs inside the experimental failure region. Figure 5.2.11 contains the transverse compression curves for the composite material T700/E862. The computed transverse compression values for the T700/E862 composite do not compare quite as well with the experimental transverse compression simulation. The simulated failure occurs just outside the failure region and the simulation stiffness is slightly greater than the experimental straight sided stiffness. However, the failure region has a tight margin so the simulated results are reasonable.
Figure 5.2.10: Axial Compression Experimental vs. Computed T700/E862 Composite Coupon Test

Figure 5.2.11: Transverse Compression Experimental vs. Computed T700/E862 Composite Coupon Test
Figure 5.2.12 contains the shear curves for the composite material T700/E862. The computed shear values for the T700/E862 composite compare well qualitatively with the shear simulation for the T700/PR520 material. Both the max stress and strain are below but not much different from the toughened epoxy composite.

![Shear Computed T700/E862 Composite Coupon Test](image)

A third material, the composite T700/5208 was also used for calibration. Figures 5.2.13-5.2.17 display comparisons of the coupon simulation results (red) with the experimental coupon stress-strain curves (blue). Error bars to represent the percent of error in the experimental curves, and a shaded light blue area indicating the region in which failure occurred are included in the following figures. Figure 5.2.13 contains the axial tension curves for the composite material T700/5208. The computed axial tension values for the T700/5208 composite compare very well with the experimental axial tension test curve. The stiffness of the simulation is very close to the experimental values.
stiffness and the simulated failure occurs within the failure region. Figure 5.2.14 contains the transverse tension curves for the composite material T700/5208. The computed transverse tension values for the T700/5208 composite do not compare quite as well with the experimental transverse tension test curve. The simulated failure occurs outside of the experimental failure region and the stiffness of the simulation is much greater than the experimental straight sided stiffness. Again this is most likely due to the edge effects noted in the straight sided coupon testing (Littell, 2008). Notched specimen testing was also not available for this material.

Figure 5.2.13: Axial Tension Experimental vs. Computed T700/5208 Composite Coupon Test
Simulations of the compression coupon tests were also conducted for the T700/5208 material. Figure 5.2.15 contains the axial compression curves for the composite material T700/5208. The computed axial compression values for the T700/5208 composite compare somewhat with the experimental axial compression test curve. Overall, the stiffness of the simulation compares very closely to the experimental stiffness and the simulated failure occurs slightly outside the experimental failure region. Figure 5.2.16 contains the transverse compression curves for the composite material T700/5208. The computed transverse compression values for the T700/5208 composite compare fairly well with the experimental transverse compression simulation. The simulated failure does occur inside the failure region and the failure strain is within the error range.
Figure 5.2.15: Axial Compression Experimental vs. Computed T700/5208 Composite Coupon Test

Figure 5.2.16: Transverse Compression Experimental vs. Computed T700/5208 Composite Coupon Test
Figure 5.2.17 contains the shear curves for the composite material T700/5208.

The computed shear values for the T700/5208 composite compare well qualitatively with the shear simulations for the T700/PR520 and T700/E862 materials. Both the max stress and strain for this simulation are below those of these other composites.

![Shear Computed T700/5208 Composite Coupon Test](image)

Only one representative experimental test was available for the composite material T700/3502 for each unidirectional state: axial tension, transverse tension, axial compression, and transverse compression. These curves were gathered by Littell (Littell, 2008). Therefore, no statistics were conducted for this material and the representative curves were used for calibration. The computed axial tension T700/3502 composite coupon simulation also compares well with the experimental axial tension test curve, Figure 5.2.18. For the T700/3502 material, the simulation is slightly stiffer and weaker,
so the area under the curve is equivalent, and the nonlinearity is captured the simulation,

Figure 5.2.18.

Figure 5.2.18: Axial Tension Experimental vs. Computed T700/3502 Composite Coupon Test

Figure 5.2.19: Transverse Tension Experimental vs. Computed T700/3502 Composite Coupon Test
The computed transverse tension T700/3502 composite coupon simulation is the weakest of the four simulations, Figure 5.2.19. This simulation correlates well with the input properties, qualitatively. The computed compression simulations do not compare well with the experimental test, but compare well to the other three materials qualitatively. The computed axial compression T700/3502 composite coupon simulation can be seen in Figure 5.2.20, and the computed transverse compression T700/3502 composite coupon simulation can be seen in Figure 5.2.21.

Figure 5.2.20: Axial Compression Experimental vs. Computed T700/3502 Composite Coupon Test
Figure 5.2.21: Transverse Compression Experimental vs. Computed T700/3502 Composite Coupon Test

Figure 5.2.22 contains the shear curves for the composite material T700/3502. The computed shear values for the T700/3502 composite compare well qualitatively with the shear simulations for the other three materials. Both the max stress and strain for this simulation are below those of the other three composites. This is expected because T700/3502 is the weakest composite of the four. The inputs for these simulations can be seen in Table 4.1.4.1.2.
5.2.1 Integration Point Study

An integration point study was conducted to analyze the effect caused by the number of through-the-thickness integration points. First, the number of integration points was analyzed for the quasi-static coupon simulations. These simulations were conducted with the number of through thickness integration points ranging from 2 to 8. There was no difference between any of the quasi-static coupon simulations. This was determined to be due to the lack of out of plane deformation in the quasi-static coupon simulations (which reflects the test results). After this, the number of integration points was analyzed for the impact simulations described in the next section of this report. For the impact simulations the number of through thickness integration points was started at 2 and then increased until the solutions converged. There was no difference between the impact simulations using 12 and 15 integration points. Therefore, 12 integration points through-the-thickness were used for all the simulations used in this analysis. All quasi-

![Figure 5.2.22: Shear Computed T700/3502 Composite Coupon Test](image)
static and impact simulation results reported in this paper were run using 12 integration points through the thickness which amounts to 2 integration points per braid layer.

5.3 Conclusion

The analytical tool Independently Homogenized Subcells required calibration. This was done using quasi-static properties from experimental testing. However, the raw experimental data had to be transformed into usable data for the calibration. Statistics were performed on the data to create average curves that were then used to calibrate the Independently Homogenized Subcells program. This more accurate and versatile numerical simulation tool helps to better analyze the effect of dominant composites properties on impact deformation and failure in composite structures. It was necessary to calibrate this simulation tool so that the systematical calculation of input parameters to accurately simulate both ballistic limit and damage patterns could be accomplished.
CHAPTER VI
IMPACT, TUBE, & DEFLECTION

6.1 Introduction

Once the correlation with the quasi-static level testing was completed, the predictive capability of this study’s approach could be investigated. Because all of the material characterization was conducted based on results from the quasi-static coupon tests, the impact and tube simulations discussed in this section of this study are truly predictive. There is a large amount of variability in impact testing with triaxially braided composites due to the test setup. This is explained in detail in Chapter I.

Because of this variability, one type of highly controlled test setup was simulated to verify the Independently Homogenized Subcells approach. This setup will be described in detail in the following section. Additional tests were then simulated in order to examine the predicative capability of the approach when modeling other structures with different boundary conditions. These additional simulations used a larger flat panel with a square boundary and a tube that is internally pressurized.

6.2 Approach

After the material properties for each of the composites examined in this study were determined using the Independently Homogenized Subcells program (Chapter IV) and correlated using quasi-static coupon tests (Chapter V), a series of simulations were conducted.
6.2.1 Impact Simulation

In order to examine the predictive capability of the analysis approach, impact tests conducted by Pereira, et al. were simulated (Pereira, et al., 2010). The impact tests, which were conducted in the NASA Glenn Research Center Ballistic Impact Lab, utilized a single state compressed gas gun to propel an aluminum 2024 projectile into 0.305 m X 0.305 m X 0.0032 m composite panels composed of six layers of the T700/E862 composite. The composite panel was held in a circular fixture with an aperture of 0.254 m. The other materials have not been tested using this particular setup at the current time.

This set of high velocity impact tests were performed using a hollow hemispherical projectile (Pereira, et al., 2010). The projectile was a thin walled hollow cylinder with a nominal mass of 50 grams and a front face with a compound radius. The projectile’s dimensions can be seen in Figure 6.2.1.1. The overall length of the projectile was 0.0495 m, the wall thickness was 0.00076 m and the nominal diameter was 0.05067 m. The compound radius on the nose of the projectile was used to prevent stress concentrations that occur in impact test with projectiles that have sharp edges. The outer 1.5 inch radius was set to mimic the curvature of the composite under impact and therefore further minimize stress concentrations. An aluminum projectile was used because the properties of this material are well defined, and this limits the amount of variability due to the projectile in both the experiments and the simulation. Seven projectile shape iterations were successively analyzed with finite element analysis and the last three of these iterations were also tested in order to achieve the final shape and mass of the projectile seen in Figure 6.2.1.1. The projectile was Al-2024 aerospace grade
aluminum and was modeled with material properties derived from the extensive experimental testing performed on this material by Buyuk et al. (Buyuk, et al., 2009). The projectile was modeled as a linear elastic material to accurately account for energy absorption due to elastic deformation of the projectile. Plastic deformation was assumed to be negligible due to the lack of plastic deformation seen in the projectiles during testing. The density and stiffness of the projectile used in the simulation were measured from the aluminum used in the experimental testing.

![Figure 6.2.1.1: Simulated Al 2024 Projectile Dimensions (inches)](image)

The composite mesh simulated in the impact analysis had dimensions and constraints replicating those of the experiments. The panels were 30.5 x 30.5 cm (1 x 1
ft) squares with clamped boundaries in 25.4 cm (10 in) diameter circles centered in the panels. The nodes in the circular boundary had all six degrees of freedom constrained. The panels were made up of 810 unit cells; 15 unit cells across and 54 unit cells vertically. Because there were four elements in a unit cell, the panel had 2700 elements, and 2805 nodes (Figure 6.2.1.2). The contact card employed was the LS-DYNA® contact “*Contact_Automatic_Single_Surface” (Hallquist, 2007).

Figure 6.2.1.2: Finite Element Mesh for Impact Simulations

The initial position of the projectile in the analysis is perpendicular to the composite plate heading directly at the center of the circular constrained area. The distance between the nose of the projectile and the composite plate is initially two inches in order to conserve time but allow enough space for the velocity of the projectile to reach steady state before impacting the plate. An initial velocity is given to the projectile in the z direction (Figure 6.2.1.3).
Experimental tests were performed over a range of impact velocities to determine the velocity for the onset of damage, the growth of damage with increasing velocities, the penetration threshold, and the damage pattern induced by penetration. Twelve panel tests were conducted using impact velocities ranging from 157 m/s to 175 m/s. The threshold velocity for penetration was determined to be between 161 m/s and 168 m/s (Pereira, et al., 2010).

6.2.2 Deflection

In order to examine the deflection produced by the Independently Homogenized Subcells approach, impact tests conducted by Roberts, et al. were simulated (Roberts, et al., 2002). This set of impact tests, which were conducted at the NASA Glenn Research Center Ballistic Impact Lab, utilized a single state compressed gas gun to propel a gelatin
projectile into 0.610 m X 0.610 m X 0.0032 m composite panels composed of six layers of the T700/PR520 composite. The composite panel was bolted in a rectangular fixture.

The high velocity impact tests were performed using a ballistic gelatin projectile (Roberts, et al., 2002). The projectile was a solid gelatin cylinder with a density of 0.93 grams/cc. The projectile's dimensions can be seen in Figure 6.2.2.1. The overall length of the projectile was 10.67 cm and the nominal diameter was 6.35 cm. The projectile was modeled using the smooth particle hydrodynamics (SPH) gridless meshing technique available in LS-DYNA® (Hallquist, 2007). The material model *Mat_Null was used with an equation of state (EOS) for a gelatin material with a porosity of forty percent. The EOS used was developed by the National Institute for Aviation Research (NIAR) for ballistic gelatin used in impact analysis (Tufano, 2011). There is far more data available for this impact test method. However, it was not used to verify the ballistic limits and damage patterns of the composites materials because of the variability of the gelatin material model. Ballistic gelatin material testing and modeling is an active research area and a systematic approach to mixing and modeling gelatin is still in the process of being defined.
The composite mesh simulated in the deflection analysis had dimensions and constraints replicating those of the experiments. The panels were 60.96 x 60.96 cm (2 x 2 ft) squares with clamped boundaries at the edge of the panels. The nodes in the boundary had all six degrees of freedom constrained. The panels were made up of 2346 unit cells; 34 unit cells across and 69 unit cells vertically. Because there were four elements in a unit cell, the panel had 9384 elements, and 9589 nodes (Figure 6.2.2.2). The contact card employed was the LS-DYNA® contact "*Contact_Automatic_Nodes_to_Surface" (Hallquist, 2007).
The initial position of the projectile in the analysis is perpendicular to the composite plate heading directly at the center of the square constrained area. The distance between the near side of the projectile and the composite plate is initially two inches, similar to the baseline impact simulation. An initial velocity is given to the projectile in the $z$ direction.

6.2.3 Tube Simulation

The simulation of a tube pressure test is another way to exercise the model under different conditions. Tube testing was initiated in order to obtain more representative in-situ triaxially braided composite transverse tension properties. This is because tubes can be manufactured such that the bias fiber bundles have no transverse edge termination point. The fibers are braided continuously winding around the tube and only terminate at
the top and bottom of the tube. The tube in Figure 6.2.3.1 is 10.7 cm in diameter and has six layers of braid fitted concentrically together.

![Composite Tube Simulation and Tube Test Sample (Salem, et al., 2011)](image)

Figure 6.2.3.1: Composite Tube Simulation and Tube Test Sample (Salem, et al., 2011)

Tube tension, torsion, and pressurization simulations can be performed using the material inputs developed in Chapter IV. Currently the only test data available to compare with is for tube pressurization. The composite tube pressurization testing was conducted at NASA Glenn Research Center by Salem et al. (Salem, et al., 2011). The tubes are 15.2 cm tall, have an inner diameter of 10.7 cm, an outer diameter of 9.9 cm, and a wall thickness of 3.2 mm. The composite tube simulation was built to represent these dimensions. An elastomer with a Poisson’s ratio of approximately 0.5 was placed in the center of the composite tubes during the tests. Two pistons were then loaded in compression on the top and bottom surfaces of the elastomer in order to pressurize the center of the composite tubes. In order to simulate the elastomer in the model, the *Airbag_Linear_Fluid card was used in LS-DYNA® along with a linear elastic material.
(Hallquist, 2007). Because a Poisson’s ratio of exactly 0.5 can cause numerical instabilities, a Poisson’s ratio of 0.499999 was used. Two circular ridged plates were placed above and below the simulated elastomer, and the elastomer was then pressurized by inflation. The height of the elastomer was modeled at 4.8cm (1.875 in) to account for the loss of height due to compression during the test. The original height of the elastomer in the tests is 5.08cm (2 in). *Contact_Automatic_Single_Surface was used to define contact between the four separate parts. The overall simulation diagram can be seen in Figure 6.2.3.2.

![Figure 6.2.3.2: Tube Pressurization Simulation Diagram](image)
6.3 Results

Since all of the characterization and correlation was conducted based on results from the quasi-static coupon tests, the simulations discussed in this section of this report are truly predictive.

6.3.1 Impact Simulation

Only one of the material systems examined in this paper, T700/E862, has undergone impact testing with the circular simulation setup described above. The setup for the impact simulation can be seen in Figure 6.2.1.2. The composite mesh simulated in the impact analysis is described above.

Figure 6.3.1.1 displays a comparison between the impact simulation and the experimental impact test. The black and white image is test specimen after impact. The black and white textured surface is painted on prior to testing to create a grid that is used in the digital image mapping process. There are two predictions to point out in Figure 6.3.1.1. The first notable item is the prediction of penetration velocity. The simulation predicted the ballistic limit of the projectile at 160 m/sec (525 ft/s) which was slightly below the experimental threshold velocity range of 162 - 171 m/sec (530-560 ft/s). The discrepancy in velocities is within 1% and therefore fairly reasonable.
The second item of note is the simulation’s prediction of the damage pattern. There are a few limitations when comparing the simulations with experimental impact results. First, the experimental image displayed in Figure 6.3.1.1 is the image of the composite panel after the test has been completed. Because of this, it is difficult to assess the mode of initial failure and failure propagation in the experimental impact data. Second, there is a great deal of scatter in the data, as with most high velocity impact testing, and therefore the failure patterns differ greatly from test to test even though the ballistic limit has been identified within a reasonably tight range.

In the experimental impact test specimen, the composite failure area appears to be shaped like an oval with more damage along the axial fibers (Figure 6.3.1.1). The simulation provides more detail on the initiation and propagation of damage. The initial area where failure propagates through all six layers of composite occurs in the lower left
corner of the impact zone. From this point the failure begins to propagate, first along the bias fibers and then vertically along the axial fibers and finally horizontally. The projectile continues to damage the composite by folding back petals of the fractured region. The macro-scale simulation cannot capture fiber bundle splintering, but it can be interpreted as one of the physical mechanisms simulated by the petaling. The overall damage pattern in the experiment is slightly more elongated than the damage pattern seen in the simulation. However, it is safe to say that the simulation damage pattern is far closer to the experimental than a typical homogenized damage pattern where a perfect cross is formed. It is possible the difference could be due, again, to the use of quasi-static properties. There may be some anisotropic stiffening due to rate effects. Rate effects are investigated in detail in Chapter VII.

High velocity impact simulations were conducted for the other three materials by using the same simulation setup. Here they are only examined qualitatively as there is no experimental data currently available to compare with directly. The T700/PR520 material had a ballistic limit of 240 m/s (775 ft/s) as seen in Figure 6.3.1.2. This was higher than the ballistic limit noted in the T700/E862 material simulation. The damage in the T700/PR520 simulation formed in a completely different way than the damage in the previous simulation. The damage in this simulation initiated at the bottom center of the impact zone and propagated, first in both horizontal directions, and second upwards along the edges of the zone vertically cutting out a door like flap of the composite plate. Variations in material properties can lead to a variation in the impact simulation results because all the properties are not necessarily scaling linearly.
The initiation of vertical damage on the right side of the impact zone can be seen in the time step displayed in Figure 6.3.1.3. The T700/5208 material had a ballistic limit of 110 m/s (350 ft/s) as seen in Figure 6.3.1.3. This was the lowest ballistic limit of the examined materials and may be due to the low quasi-static compression properties used in the Independently Homogenized Subcells program. In this simulation the failure initiated in the center of the impact zone and first propagated along the +60 bias fibers. The material continued to fail vertically and horizontally. The three petals that fold back during failure can be seen in Figure 6.3.1.3. The simulated T700/3502 material had a ballistic limit of 150 m/s (500 ft/s) as seen in Figure 6.3.1.4. Damage propagation, again, occurred in a different way than the previous simulations. Damage in this simulation initiated in the center right of the impact zone and quickly propagated along the bias fibers both above and below the damaged area. While damage propagated in the horizontal direction, a new area of damage initiated in the center of the impact zone. This
resulted in the damage of nearly all material in the impact zone. Because of this, no petaling occurred. This result differed from all the simulations with the other materials. The varying predicted damage patterns of the different materials show that material and architecturally dependent damage mechanisms have been captured by the method. These simulations are also purely predictive because there has been no correlation at the impact level.

Figure 6.3.1.3: Simulated Damage Patterns for T700/5208 Composite in Impact
6.3.2 Deflection

The gelatin impact test case was compared for deflection due to the extensive experimental z-axis displacement available for the gelatin tests (Roberts, et al., 2002). The gelatin simulation, however, is not achieving the same gelatin flow that is seen in high speed impact testing. Therefore, this finite element simulation can only be used to approximately simulate deflection and out of plate displacement. In Figure 6.3.2.1, the displacement of the simulation is compared with the experimental results. This is the back side of the composite; the side opposite the projectile. A digital image correlation software was used on the high speed video footage of the composite test to digitally map the displacement (Roberts, et al., 2002). In the simulation the initial velocity of the gelatin projectile was 186 m/s (609 ft/s) in order to mimic the velocity in the experimental test. The final displacement of the simulation compares well with the experimental displacement. However, none of the elements completely fail in the
simulation, unlike the experiment, only a few integration points fail. This lack of complete failure in any elements may be due to a number of factors. There may be a slight difference in the gelatin used to compute the EOS properties and the gelatin used in this simulation, there may not be enough interaction in the SPH nodes, or there may simply be an incompatibility with the gelatin and the composite model when simulating impact. These are some of the reasons an aluminum projectile was used in the previous section. However, even with these potential issues, the energy exhibited in the simulation indicates that the results are still useful, and therefore can be used to compare deflection. Figure 6.3.2.2 shows the z-axis displacement graphically for both the simulation and the experiment. The curves that are shown representing the simulation, are the curves for the outer layer of integration points. These simulation curves also compare somewhat well with the experimental curves. However, the curves from the simulation are slightly more elongated due to the lack of transverse flow of the gelatin upon impact.

Figure 6.3.2.1: Out of plane deflection for T700/PR520 Composite (Roberts, et al., 2002)
6.3.3 Tube Simulation

Because only the pressurized tube testing was completed at the time these simulations were conducted, only preliminary simulations were conducted for tube tension and torsion. This section will therefore focus on the comparison of the simulation of the pressurized tube with the experimental test data. A transverse tension tube simulation was conducted to compare with the experiment conducted by Salem et al. (Salem, et al., 2011). The experimental tube pictured in Figure 6.3.3.1 was pressurized using the method described above until failure. A number of tubes were tested in the same manner with the hoop stress acting in the overall transverse material direction. In general, failure occurred due to a bias fiber bundle failing in the pressurized zone and then rapidly zipping in both vertical directions until ultimate failure occurred where the composite tube would then open up and in some tests fly completely off the compressed elastomer (Salem, et al., 2011). Ultimate failure in the simulated composite was equally as dramatic, however due to the intrinsic symmetry of simulating a flawless composite material, failure occurred in four locations around the composite simultaneously. Similar to what was noted in experimental testing, the four failures, equally spaced around the
circumference, occurred in simulation in subcells ‘A’, with initiation in the center of the pressurized zone and then propagating with vertical zipping, both up and down, until ultimate failure. The strain patterns before failure can also be observed in Figure 6.3.3.1 where the localized vertical strain concentrations occur in subcells ‘A’ and ‘C’ in the simulation and then directly above axial fiber tows in the experiment.

![Simulation vs Experiment](Image)

**Figure 6.3.3.1: Transverse Strain Comparison for Pressurized Tubes**

6.4 Conclusion

A macro level finite element based approach has been developed that allows for the simulation of the response of a triaxially braided composite in a manner that takes into account the architecture of the braided material within a large scale structure. Two representative flat panel impact tests were simulated using the analysis model. Since the input material parameters were correlated based on the coupon level tests, these
simulations were purely predictive. For the one material system where experimental data was available for the circular setup, the predicted ballistic limit was only slightly below the experimental value. The predicted impact damage patterns followed the details of the braided architecture, and were similar to the experimental damage patterns. The small discrepancies may be due to a strain rate effect which is investigated in the following chapter. The predicted impact damage patterns for the four material systems examined in this study were significantly different, indicating that the analysis method could capture the effects of differing material strengths on the impact damage in the material. A deflection validation simulation was then conducted and compared well with the experimental deflection. Last, a pressurized tube simulation was conducted and it compared well to the experimental tube test. In future work, these tube tests will be used to correlate the transverse tension properties for the Independently Homogenized Subcells Program. Overall, the analysis method appears promising in its ability to simulate the architecturally dependent impact damage in braided polymer matrix composites. The next chapters will involve investigation into incorporating the effects of strain rate into the model, as well as investigating in more detail the capability of the model to simulate impact events, particularly damage before final penetration occurs.
CHAPTER VII
STRAIN RATE

7.1 Strain Rate Dependence for Composites

Triaxially braided composites are being investigated for their optimum static and impact properties. Accurate impact damage prediction may require the incorporation of strain rate dependence. In general, carbon fiber has very low strain rate dependence. However, it has been shown that the resin materials being investigated have some significant strain rate dependence (Littell, 2008). The motivation for this portion of the investigation is to assess the overall composite strain rate dependence and develop the capabilities to incorporate the effects of strain rate in impact simulations.

In braided composites numerous micromechanical phenomena occur contributing to both the ballistic limit and the damage patterns, such as preferred crack growth along tow interfaces and viscoplasticity within resin rich regions. An analysis method was developed to capture high velocity (on the order of 350 m/s) impact failure patterns, which may be critically dependent on strain rate effects. A key requirement of the method is having a systematic approach for accurate characterization of the constitutive model material parameters necessary for the finite element material analysis. The method, developed previously in Chapter IV, has been expanded to incorporate strain rate dependence into the simulations. However, obtaining coupon level test data on braided composites at high strain rates is very difficult. Increasing the strain rate in a test typically corresponds with decreasing the size of the specimen. For composites the
smallest specimen can only be as small as the unit cell, and because the unit cell for the materials currently under study is large, high strain rates are extremely difficult to obtain. Therefore, virtual experiments were conducted to fill the gaps in the experimental data and calibrate the macroscale model. The virtual experiments were conducted by Liu (Liu, 2011). The composite properties for a variety of strain rates are taken from the results of the virtual experiments and then used to calculate the material parameters required for incorporating strain rate dependence into the macro scale finite element analysis. The approach is validated by conducting a simulation of a high strain rate compression test and comparing the results to the limited experimental data.

7.2 Method for Strain Rate Incorporation

Liu used the Multiscale Generalized Method of Cells (MSGMC) to provide virtual experiment data for the FE simulation as well as provide insight into damage mechanisms and failure modes. MSGMC bridges scale between the microscale, which represents the individual fibers and matrix, and the weave scale, which represents the braid architecture at the ply level. A macro scale finite element simulation was also developed employing two advanced continuum damage mechanics material models in the commercial transient dynamic finite element code LS-DYNA® (Hallquist, 2007). For these aerospace applications, the model needed to accurately simulate the textile composite under both static loading and impact loading situations. The novel methodology in which the fiber architecture was captured into the finite element analysis has been discussed previously. The composite material analyzed in this portion of the study was T700/PR520.
7.3 High Rate Testing

Extremely strain high rate testing cannot typically be accomplished with general electromechanical load frames. A Split Hopkinson Bar test may be used in these situations to test the high strain rate material properties. This test is used to impose a dynamic load on the material specimen. There are a number of test setup variations for the Split Hopkinson Bar test, so the sketch seen in Figure 7.3.1 is a general test setup. The test specimen is placed between the incident bar and the transmitter bar. A striker bar is then used to create a stress wave (incident wave) which propagates toward the specimen. This may propagate through the specimen and become the transmitted wave, or be reflected back into the incident bar and become the reflected wave. Stress and strain can then be calculated from the amplitudes of the incident, transmitted and reflected waves. Strain gages are used on the incident and transmitter bar to collect the necessary wave data. This type of unique testing is still a very active research area for composite materials, and data is typically difficult to obtain. The Split Hopkinson Bar test data used for this study was collected by Pereira et al. (Pereira, et al., 2010). A single compressive test was performed by Pereira et al. on a unit cell of the T700/PR520 composite material at 420/s (Pereira, et al., 2010).
7.4 Virtual Experiments

The Multiscale Generalized Method of Cells (MSGMC) was employed by Liu to simulate the virtual experiments (Liu, 2011). There are a number of architectural parameters required to fully define the discretized subcell geometries. The viscoplasticity model developed by Goldberg et al. was used by Liu with a small modification to capture the rate dependent response of the resins in the composite materials (Goldberg, et al., 2005).

The virtual experiment utilized for the macroscale analysis was axial compression. This case was chosen in order to allow for comparison with the test data, which was only available in compression. This can be done because the rate dependent material model only has one rate dependent definition for all tension, compression, and shear loading cases. For both the axial compression loading case, analyses at four independent strain rates were considered: 0.5/s, 420/s, 1500/s, and 2500/s. The rate 0.5/s was chosen because that is the rate at which the quasi-static coupon tests were performed. Next, 420/s was chosen because that was the rate at which the Split Hopkinson Bar test was performed. Then, 2500/s was chosen because this rate would fully bound the peak...
strain rate observed in impact simulations, which was just below 2400/s. Finally, 1500/s was chosen as an approximate midpoint. The addition of this arbitrary point was important to complete the set of four total strain rate curves needed to appropriately define the material rate sensitivity. The ultimate fiber failure is determined at the microscale through a modified Hashin-Rotem failure criterion. The results from the virtual experiments are shown in Table 7.4.1 (Liu, 2011). The trend clearly visible in the table below is that as strain rate increases, the difference between the tangent moduli decreases. This trend was expected and is why three rate dependent curves must be used to define the non-linear progression of strain rate in the material model.

<table>
<thead>
<tr>
<th>Strain Rate</th>
<th>Tangent Modulus (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5/s</td>
<td>48309</td>
</tr>
<tr>
<td>420/s</td>
<td>48544</td>
</tr>
<tr>
<td>1500/s</td>
<td>48780</td>
</tr>
<tr>
<td>2500/s</td>
<td>49020</td>
</tr>
</tbody>
</table>

7.5 Macroscale Simulation Methodology

The algorithm described in Chapter IV was modified to include strain rate sensitivity. Littell proposed that the actual material response may be a function of strain rate (Littell, 2008). The incorporation of strain rate dependence was a difficult task because the composite panels undergo a wide array of strain rates during an impact event. Each concentric ring of elements emanating from the central point of impact has a unique strain rate; the nearer the elements are to the impact site, the higher the strain rate they experience. A number of high velocity impact simulations were evaluated to establish a
range of elemental strain rates. The strain rates calculated ranged from static to approximately 2400/s.

The LS-DYNA® material model *Mat_158 (*Mat_ Rate_ Sensitive_ Composite_ Fabric) was used to capture strain rate sensitivity in the finite element material model (Hallquist, 2007). *Mat_158 is an expansion of the material model *Mat_58. The expansion includes a method to model material strain rate dependence by applying a viscoelasticity approach including the use of a Prony series. In applying the material model, the strain rate data for strain rates observed by each individual element in the simulation is utilized.

The material model *Mat_158, utilized for the strain rate dependent analyses, is an extension of *Mat_58, uses all the same material constants as *Mat_58, plus requires additional parameters to account for the strain rate effects (Hallquist, 2007). Therefore, the quasi-static properties, determined using the procedures described in Chapter IV, were utilized as the baseline set of parameters. To incorporate the effects of strain rate, additional parameters including a shear modulus (G) and a decay constant (β) were required to define the strain rate sensitivity of the material response. The shear modulus is used by the material model to determine an initial stiffness for the material. The decay constant is then used help determine the continuum damage (Equation 7.6.1). A shear modulus and a decay constant are defined for each strain rate curve included in the material model. *Mat_158 allows the individual response of the elements to differ depending on their exact strain rate at a given time step. The material model creates curves from the shear modulus and decay constant parameters and interpolates between these curves for each elemental strain rate. Therefore, two or more non-static curves
should be included to capture any non-linear rate dependence. Typically, three additional curves are used. The two parameter scale factors are assumed to be constant in all material directions, which is a limitation of this method. Three strain rates above the quasi-static rate were chosen to characterize the model and define the full range of strain rates experienced during the simulations. A schematic of the effects of strain rate on the material response can be seen in Figure 7.5.1.

![Figure 7.5.1: Idealized image of strain rate curves needed to characterize the material model](image)

In the finite element model, only the material for subcells ‘B’ and ‘D’ needed to be converted to *Mat_158 in order to capture strain rate effects because of these subcell’s relatively low fiber volume ratio. Carbon fiber has an insignificant strain rate sensitivity, and therefore the fiber rich subcells ‘A’ and ‘C’ can continue to be modeled using *Mat_58. To characterize the strain rate dependent material model, high strain rate data, and the variation of the ultimate strength as a function of strain rate are required. Using this data, a set of global composite stress-strain curves at various strain rates were generated and then used to characterize the strain rate sensitive model.
7.6 Macroscale Simulation Results

For this study, a strain rate of 420/s was used to calculate the first set of parameters, a strain rate of 1500 1/s was used to calculate the second set of parameters, and a strain rate of 2500 1/s was used to calculate the third set of parameters. In total, there are three sets of terms used in this study to define the strain rate sensitivity portion of the *Mat_158 material model (Hallquist, 2007). The tangent moduli for these sets are obtained from the virtual experiments. The decay constant for each strain rate is set equal to the longitudinal modulus of elasticity of the material at the strain rate divided by the strain rate and multiplied by the ultimate axial tensile strength of the material at the given strain rate, seen in Equation 7.6.1 (Hallquist, 2007). For this particular study, the material constants that were utilized for the T700/PR520 material which was examined included the decay constant ($\beta$) and the shear modulus listed in Table 7.6.1. All other material constants can be found in Chapter IV.

$$\beta = \frac{E}{\dot{\varepsilon}} \times \sigma_{ult} \quad (7.6.1)$$

<table>
<thead>
<tr>
<th>Beta</th>
<th>Shear Modulus (MPa)</th>
<th>Strain Rate (1/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.26E+04</td>
<td>7.71E+03</td>
<td>420</td>
</tr>
<tr>
<td>8.10E+04</td>
<td>7.75E+03</td>
<td>1500</td>
</tr>
<tr>
<td>1.36E+05</td>
<td>7.79E+03</td>
<td>2500</td>
</tr>
</tbody>
</table>

Table 7.6.1: Beta and Strain Rate Values for T700/PR520
To validate the material model and the analysis method, an axial compression test conducted by Pereira, et al. of the T700/PR520 material conducted at a strain rate of 420/s was simulated (Pereira, et al., 2010). The test was conducted by using the Split Hopkinson Bar approach. This limited set of high strain rate data was the only high strain rate experimental data available at this time.

The axial compression test was simulated at a high strain rate of 420/s to compare with the experimental test. It was simulated using the geometry shown in Figure 7.6.1 with the hatch marks representing the fixed boundary and the arrows representing the displacement controlled boundary. The full Split Hopkinson Bar setup was not simulated because this axial compression simulation was to be used for model correlation, and the addition of more materials due to the bars would add more assumptions than necessary.

![Figure 7.6.1: Axial Compression Simulation](image)

The results can be seen in Figure 7.6.2. The curves displayed in the figure are the high strain rate simulations along with simulation results conducted at a quasi-static strain rate, and quasi-static experimental results for comparison (Littell, 2008), (Roberts, et al., 2002). The simulated quasi-static curve correlates reasonably well with the experimental
results. The curve shown in Figure 7.6.2 represents the failure stress and strain obtained from the Split Hopkinson Bar test data (Pereira, et al., 2010). The fact that the failure stress increases as the strain rate increases was captured by the simulations, and the computed failure stress at the high strain rate correlated reasonably well to the limited experimental data. The high failure strain seen in the high rate simulation may be due to the coarseness of the resulting stress-stress curve (Figure 7.6.2).

![Figure 7.6.2: Axial Compression (T700/PR520) - Experimental Split Hopkinson Bar Test (black), Experimental quasi-static test (green), High rate simulation at 420 1/s (blue), and Static simulation (red) (Pereira, et al., 2010)](image)

Efforts were also made to include the effects of strain rate in impact simulations. Initially, utilizing the material model *Mat_58 without strain rate dependence for some elements and the material model *Mat_158 with strain rate dependence for other elements resulted in numerical instabilities. To help solve this issue, the elements consisting of subcells ‘A’ and ‘C’ were converted to be modeled with *Mat_158.
However, with the parameters set such that the strain rate dependence was neglected. Modeling all the elements with the same type of material model helped eliminate the instabilities. The coupon simulations did not exhibit the same numerical instabilities, which most likely involved the simulation of contact. Therefore, the coupon simulations were used to compare the two methods. The coupon simulations with the *Mat_58 material model used for subcells ‘A’ and ‘C’ had exactly the same results as the coupon simulations with *Mat_158 used for these subcells. In addition, the run time of the impact simulation, which included strain rate dependence, was about three times that of a baseline impact simulation in which only strain rate independent (quasi-static) properties were used.

The impact simulation was designed to model the impact experiment detailed in Pereira et al. (Pereira, et al., 2010). Full details of the panel impact setup with the circular boundary can be found in Chapter VI.

Using only static properties, a ballistic limit of 221 m/s was predicted. Incorporating strain rate dependence into the analysis increased the simulated ballistic limit by 9%, which resulted in a ballistic limit of 236 m/s. Theoretically, the ballistic limit obtained from the strain rate dependent simulation should be more realistic. However, at this time there is no experimental impact data to compare with directly for this particular material system. Detailed comparisons with experimental impact test will be performed in future work. The predicted impact damage patterns also change as a result of incorporating strain rate dependence into the analysis. As can be seen in Figure 7.6.3, for the simulation with strain rate independent quasi-static material properties the damage initiates on the right top corner of the impact zone and propagates first downward.
along the axial fiber bundles, then upward at an angle along the bias fiber bundles, and then downward from the bottom corner at an angle along the bias fiber bundles forming a flap with opens allowing the projectile to propagate through. However, for the simulations which incorporated strain rate dependence of the material, the damage initiated at the bottom left corner of the projectile, then propagated horizontally along the bottom of the projectile, then propagated vertically upward along both sides of the projectile, creating a flap which opened allowing the projectile to travel through the material. The incorporation of strain rate dependence into the analysis resulted in an increase in the predicted ballistic limit and a variation in the predicted impact damage patterns compared to the results obtained using quasi-static material properties.

Figure 7.6.3: Impact Simulation Comparison
7.7 Conclusion

A modeling approach was developed to analyze the impact response of triaxially braided polymer matrix composites including the architecturally dependent damage patterns. The methodology has been expanded to incorporate the strain rate dependence of the material response. A methodology has been developed to determine the initial quasi-static input material properties for the analysis based on the results of coupon level experiments on the braided composite. Due to the lack of experimental data at high strain rates, a multiscale micromechanics based analysis method was utilized to conduct a series of virtual experiments in order to characterize the parameters which control the strain rate dependence of the material’s constitutive model. In these virtual experiments, mechanical tests at high strain rates were simulated, which provided data to characterize the macro scale material model. Utilizing the developed method, a series of flat plate impact simulations were carried out using a representative braided composite. The simulations demonstrated that incorporating the effects of strain rate increased the predicted ballistic limit to a small extent. Furthermore, the predicted impact damage patterns changed as a result of accounting for the strain rate dependence of the material response. However, the run time of the simulations significantly increased when the effects of strain rate were accounted for. Also this material system may not have the same strain rate response in both in-plane directions, so the constant material scaling in all directions may not be appropriate. Therefore, the advantages of accounting for the effects of strain rate in the impact simulations need to be investigated further. Furthermore, other material systems where the effects of strain rate may be more pronounced need to be further investigated. Overall, the developed methodology shows
promise in providing a systematic and accurate method of simulating the impact response of textile composites.
8.1 Background

For these types of applications, it is necessary to have a model that accurately simulates not only the ballistic limit and damage patterns in an impact region, but also the effect of barely visible impact damage (BVID). BVID plays an important role in the design process of many composite structures. In an impact event, a projectile may not penetrate through the composite material, but there may still be a significant decrease in the load carrying capacity of the structure. It is important to be able to simulate the breakpoints in energy when BVID starts to occur, when BVID becomes significant, and when BVID should be considered catastrophic for a given structure. These points will depend on the material and the purpose of the structure. The BVID mechanisms are highly dependent on the material constituents, the structure’s layup, the textile architecture, and the shape of the projectile. Specifically, in a braided composite, numerous micromechanical phenomena occur contributing to the ballistic limit, the damage patterns, and BVID.

Currently the only failure mechanism under study is delamination between the layers of braid. There have also been a number of approaches to model delamination and failure. Elmarakbi et al. developed a way to model delamination in solid element structures with cohesive elements (Elmarakbi, et al., 2009). Loikkanen investigated an approach where a similar delamination could be simulated with shell elements using a
mechanism known as tie-break (Loikkanen, 2011). Cohesive elements and tie-breaks are the two most common ways of modeling delamination.

The approach, initially developed in Chapter IV to fill this need for quasi-isotropic triaxially braided carbon-epoxy composites, has been expanded to also simulate BVID. This method is theoretically applicable to any textile composite architecture.

8.2 Approach

The previously investigated analytical approach to capture architecturally dependent damage discussed in Chapter IV in triaxially braided composites under impact conditions is utilized for this method. The nature of the subcell discretization allows the braid architecture to be modeled as a series of laminated composites. In order to model delamination between each lamina a tie constraint is placed, this can be seen in Figure 8.2.1. The overall composite laminate is comprised of six plies of braided composite and each ply is modeled explicitly as a shell element, as seen in Figure 8.2.1. Six shell elements through the thickness are used in the current analysis. Each of these shell elements has the same subcell properties as those documented in Chapter IV.
The previous method has been expanded to simulate delamination in order to capture BVID. Due to the construction of the composite material, the resin regions between layers are relatively weak areas in the material structure and can initiate damage by delaminating. Therefore, for the six layer braided composite, the cohesive zone is assumed to be between each layer of braid and have the characteristic properties of the matrix material. Because shell elements are used in this simulation, the cohesive zone of the material will be molded using tie-breaks. These ties connect adjacent nodes with a kind of simulated cohesive zone. In LS-DYNA® the tie-break contact option utilized here (option 11) defines the length of the cohesive zone using Equation 1 where $M$ is the convergence scale factor, $E$ is the elastic modulus, $T$ is the peak traction (stress), and $G_c$ is the strain normal energy release rate (Hallquist, 2007).

$$L_{cz} = M \cdot E \cdot G_c / (T^2) \quad (8.2.1)$$
The strain energy release rate can be found using Equation 8.2.2 where $K_{1c}$ is the resin fracture toughness, and $E$ is the elastic modulus (Irwin, 1957). The resin fracture toughness can typically be found experimentally via a resin fracture toughness test, Figure 8.2.2.

$$G_c = \frac{K_{1c}^2}{E} \quad (8.2.2)$$

Figure 8.2.2: Resin Fracture Toughness Test

In order to appropriately simulate the cohesive zone using tie-breaks, three elements must fit within the cohesive zone, Figure 8.2.3. Epoxy cohesive zones are typically between 1 and 2 mm in length. This poses a problem when conducting macro scale analysis. For this analysis, the elements are 4.45 mm in length, meaning there would be less than half an element in the cohesive zone if modeled traditionally. However, LS-DYNA® allows for the use of a convergence scale factor. This term is used to artificially lengthen the cohesive zone to achieve three elements along the length while maintaining fracture energy. Turon et al. conducted a study on the convergence scale factor to determine the values for different sizes of elements, Figure 8.2.4 (Turon, et al., 2007). As can be seen in the figure, the element length was capped at 4 mm. Therefore,
for this exercise the convergence scale factor for the 4.45 mm long element has been extrapolated to be 0.2.

Figure 8.2.3: Length of the Cohesive Zone

Figure 8.2.4: Convergence Scale Factor vs. Element Length
The normal strain energy release rate for the resin E862 was determined to be 162 J/m\(^2\) (0.0057 lbf/in) by Nicolais (Nicolais, 2011). However, the shear energy release rate required to define the mixed-mode traction-separation law needed for tie break was not available (Hallquist, 2007). In order to calculate a reasonable approximation of the shear energy release rate, a ratio of the resin strengths was used. Tensile and shear tests conducted by Littell were used (Figure 8.2.5 and Figure 8.2.6) (Littell, 2008). In this way, the maximum shear strength (66 MPa) can be divided by the maximum normal strength (82 MPa) and then multiplied by the normal strain energy release rate to yield an approximate strain energy release rate of 130 J/m\(^2\) (0.0046 lbf/in). The resin from the T700/E862 material is used exclusively in these calculations because the three point bending test data and impact test data was available for this material, but similar calculations could be used for the other materials discussed in the previous chapters.

Once the strain energy release rates have been determined, the peak traction stresses can be calculated. These are the tie-break parameters where the convergence scale factor is included. The calculation for the normal peak traction stress can be seen in Equation 8.2.3. Similarly, the calculation for the shear peak traction stress can be seen in Equation 8.2.4.

\[
T = \sqrt{\frac{M+E+G_1c}{LCZ}} \quad (8.2.3)
\]

\[
S = \sqrt{\frac{M+G+G_{sc}}{LCZ}} \quad (8.2.4)
\]

From these equations the normal peak traction stress was calculated to be 2.07x10\(^5\) J/m\(^2\) (9.21 psi), and the shear peak traction stress was calculated to be 9.04x10\(^4\) J/m\(^2\) (6.10 psi). These may seem low, but the due to the convergence scale factor, a larger element
length translates to a smaller peak traction stress. The E862 resin elastic and shear moduli used can be found in Chapter IV, Table 4.1.4.1.1.

Figure 8.2.5: Tensile stress-strain response for E862 at Room Temperature (Littell, 2008)
The final parameters needed to define the tie-break constraints are the exponent in the damage model, the ratio of normal stiffness to tangential stiffness, and normal stiffness. The exponent in the damage model is the exponent of the mixed mode criteria used to account for total relative displacement in the transverse direction (Matzenmiller, et al., 2006). The exponent in the damage model can be obtained by curve-fitting the fracture toughness curves of the mixed-mode tests. This curve-fitting was not performed for this study, so a representative epoxy value of 1.5 is used. Next, the ratio of normal stiffness to tangential stiffness is 1 for E862 because the material is isotropic. Last, the elastic modulus is used for this parameter. For the E862 resin, the elastic modulus is 2.70 GPa (3.92x10^5 psi). The full set of parameters required to define tie-break can be found in Table 8.2.1.
### Table 8.2.1: Tie-Break Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>E862</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normal Peak Traction Stress (NFLS)</td>
<td>2.07e5 J/m$^2$</td>
</tr>
<tr>
<td>Shear Peak Traction Stress (SFLS)</td>
<td>9.04e4 J/m$^2$</td>
</tr>
<tr>
<td>Exponent in Damage Model (PARAM)</td>
<td>1.5</td>
</tr>
<tr>
<td>Normal Energy Release Rate (ERATEN)</td>
<td>162   J/m$^2$</td>
</tr>
<tr>
<td>Shear Energy Release Rate (ERATES)</td>
<td>130   J/m$^2$</td>
</tr>
<tr>
<td>Ratio of Normal Stiffness to Tangential Stiffness (CT2CN)</td>
<td>1</td>
</tr>
<tr>
<td>Normal Stiffness (CN)</td>
<td>2.70  GPa</td>
</tr>
</tbody>
</table>

8.3 Results

A unit cell in a three point bending configuration was used to calibrate the simulation, Figure 8.3.1. A visualization of this simulation can be seen in this figure. Because of the thickness of each layer, the nodes of the bottom of the top layer and the top of the second layer are actually coincident, and so on. In this simulation the bottom corners are fixed in the vertical direction and a load is applied to the top middle nodes (Figure 8.3.1). The three point bending simulation is correlated with experimental three point bending tests. The primary validation of this test is that a few of the individual tie-breaks fail at sub ultimate stress while the overall elements remain intact, Figure 8.3.2. This is desirable because delamination should occur before ultimate failure of the material. The circles indicate tie-break failure. In this way the simulation will be able to predict BVID.
These three point bending simulations can be used to calibrate the overall tie-break constraints in order to simulate accurate BVID in impact. For each material they can be compared with the three point bending test data. The simulation may need to be correlated somewhat with the experimental data to achieve realistic results.

In Figure 8.3.3 the tie-break simulation has been correlated with the experimental results for the material T700/E862. Four sets of experimental tests, which were collected by Kohlman, are shown (1, 2, 3, and 4) (Kohlman, 2012). The bending simulation for the material T700/E862 is also shown. This simulation used the properties tabulated in Table 8.2.1 for the tie break and used the material properties from Chapter IV for the
T700/E862 material. Overall the peak deflection before failure is captured by the simulation. However, slightly more deflection is seen earlier in the simulation and the load curve increases at a slightly more gradual rate than the experimental tests. This may be due to the sensitivity of the simulation. Another item to note is the slight nonlinearity seen at 250 lbf. This is due to the initial tie break failures. The small areas of nonlinearity prior to this are due to dynamic vibration of the model because the upper layers are only constrained by ties. An image of the deflection in the simulation at ultimate failure can be seen in Figure 8.3.4. The load-deflection curve for a bending simulation of the baseline unit cell with only one layer of shell elements can also be seen for comparison (Figure 8.3.3).

Figure 8.3.3: T700/E862 3 Point Bending Tests (1 to 4), Baseline Simulation (black) and Tie-Break Simulation (red) Comparison
Preliminary investigation of impact modeling has also been conducted. The same kind of setup is used for the impact simulation. A 30.48 cm square plate is modeled using six shell elements through the thickness, where each of the coincident nodes are then connected with a tie constraint using the same tie-break definition as the three point bending simulation. Because of the setup of the impact analysis, six separate contacts are defined. Five of those contacts are used for the tie-break definitions between each layer, and one contact is used between the shell elements and the projectile. It is important to be sure none of the contacts interfere with one another. To do this the automatic surface to surface option was used for contact between the projectile and the plate.

The simulation in Figure 8.3.5 has a 30.48 cm square panel constrained with all six layers of elements constrained in all six degrees of freedom for the edges, with a projectile traveling at 213 m/s. This impact simulation also uses the material properties described for the three point bending simulation. Figure 8.3.5 illustrates the delamination of the composite plate in an impact event. The projectile hits in the center of the plate and almost immediately there is an outward progression of damage. The outward progression of delamination is particularly manifest in the horizontal direction.
8.4 Conclusion

For these types of applications presented in this study, it is necessary to have a model that accurately simulates not only the ballistic limit and damage patterns in an impact region, but also the effect of barely visible impact damage (BVID). For polymer matrix composites BVID is typically caused by delamination within the layers of the material. In an impact event, a projectile may not penetrate through the material, but there may still be a significant decrease in the load carrying capacity of the structure due to delamination. It has been shown that delamination can be captured in a macro-scale finite element simulation. For design, it is important to be able to simulate the breakpoints in energy when BVID starts to occur, when BVID becomes significant, and when BVID should be considered catastrophic for a given structure. These break points
will depend on the material and the purpose of the structure, but can be captured by simulating delamination. Future work will involve correlating with the three point bending test for each of the composite materials currently under study, and conducting impact simulations for all of these materials. For each of these material the breaks will be determined where BVID starts to occur, where it becomes significant, and where is should be considered catastrophic.
CHAPTER IX
SUMMARY & FUTURE WORK

9.1 Summary

An in-depth analysis was required to characterize the impact response of carbon fiber triaxially braided composite turbo jet engine cases. An effort was made to characterize the material model being used to simulate triaxially braided composite materials. The material properties utilized for the analysis were determined based on experimental tests conducted on the braided composite. During this effort to capture the unique failure mechanisms of laminated textile composites, several steps were taken to ensure that the simulation output is a direct result of scientifically utilizing experimental data. The various material strength parameters that are input into the model have a significant effect on the overall results of the simulations, sometimes in non-intuitive ways. Through these parametric studies, a deeper understanding of the sensitivity of the macromechanical model has been obtained (Chapter III). This illuminated the need for a quasi-empirical program to capture material failure. This quasi-empirical program needed to systematically calculate input parameters exclusively from experimental and manufacturing data, and then improve the existing approaches to capture both the ballistic limit and architecturally dependent damage. The analytical tool Independently Homogenized Subcells was designed to meet this need (Chapter IV). This more accurate and versatile numerical simulation tool aids to better analyze the effect of dominant composites properties on impact deformation and failure in composite structures. In
order to capture the complex composite material systems, a top-down approach was merged with a bottom-up approach for simulating the ballistic limit and damage patterns in textile composites. The top-down portion used global strengths obtained from macroscale experimental quasi-static coupon tests to characterize the material strengths. The bottom-up portion used micro-scale fiber and matrix stiffness properties to characterize the material stiffness. Using the combination of the two portions, the dominate micromechanical mechanisms were captured without having to individually identify and characterize them. The analytical tool Independently Homogenized Subcells required calibration. This was done using quasi-static properties from experimental testing. However, the raw experimental data had to be transformed into usable data for the calibration. Statistics were performed on the data to create average curves that were then used to calibrate the Independently Homogenized Subcells program (Chapter V). It was necessary to calibrate this simulation tool so that the systematical calculation of input parameters to accurately simulate both ballistic limit and damage patterns could be accomplished. Then, representative flat panel impact tests were simulated using the analysis model (Chapter VI). Since the input material parameters were correlated based on the coupon level tests, these simulations were purely predictive. For the one material system where experimental data was available, the predicted ballistic limit was only slightly below the experimental value. The predicted impact damage patterns followed the details of the braided architecture, and were similar to the experimental damage patterns. The predicted impact damage patterns for the four material systems examined in this study were significantly different, indicating that the analysis method could capture the effects of differing material strengths on the impact damage in the material.
The methodology was then expanded to incorporate the strain rate dependence of the material response (Chapter VII). Utilizing the developed method, a series of flat plate impact simulations were carried out using a representative braided composite. The simulations demonstrated that incorporating the effects of strain rate increased the predicted ballistic limit to a small extent. Furthermore, the predicted impact damage patterns changed as a result of accounting for the strain rate dependence of the material response. The approach was then further expanded to account for delamination between the layers of braid. For ballistic impact loading types of applications, it may be important to have a model that accurately simulates not only the ballistic limit and damage patterns in an impact region, but also the effect of barely visible impact damage (BVID). It was shown that delamination between braid layers can be captured in a macro-scale finite element simulation (Chapter VIII). Overall, the approach summarized here is an advantageous approach for modeling large or macro-scale composite structures because the micromechanical mechanisms are accounted for even when using larger mesh sizes. The systematic calculation methodology rendered in this paper is also applicable to any braided or woven composite.
9.2 Future Work

There are a number of avenues open to expanding the work presented in this study. The Independently Homogenized Subcells program could be expanded to incorporate a variety of additional material phenomena, the tested properties could be refined, and a number of extensions could be investigated. The method presented could be somewhat easily modified to allow for more general textile composite geometries. A more advance extension of the program would be to create a user integrated program that would allow the user to select from different tow sizes and angles. This type of extension would enable an analyst or a designer to quickly define reasonable macro-mechanical properties for any composite material. The properties used in the method could also be refined. Because there are a number of new tests methods being developed, new methods such as tube tests to correlate the transverse tension properties could be used. Last, delamination and damage before failure is becoming more important in the characterization of composites, and will become more important in future composite simulation. Further investigation into the complex mechanisms for these types of phenomena may prove useful. Particularly involving the concept of real-time re-meshing. Overall, there are many ways to expand on the research and method presented here, and several of these avenues will be important in future composite research and simulations.
BIBLIOGRAPHY


APPENDIX

INDEPENDENTLY HOMOGENIZED SUBCELLS (I.H.S.) MATLAB CODE

A.1 Subroutines

calc_Input_SAE.m

calc_Input_SI.m

icanstr2_SAE.m

icanstr2_SI.m

subican.m

compliance.m

axial_t_LL.m

trans_t_LL.m

trans_t_LLzero.m

shear_LL.m

writeInputCard.m

writeInputExcel.m

input_calc.m
A.2.1 GUI Screen

Figure A.2.1: I.H.S. Graphical User Interface Screen
A.2.2  calc_Input_SAE.m

%calc_Input_SAE.m
%Runs Input Calculations for braided composite material
%in SAE units
%
%Created: 1-19-10
%By: Brina Blinzler (University of Akron)
%Last Edited: 5-28-11

warning off;

%GUI Inputs
GSAF = str2double(get(handles.GSAFedit,'String'));
GRAF = str2double(get(handles.GRAFedit,'String'));
GSATS = str2double(get(handles.GSATSedit,'String'));
GSTIF = str2double(get(handles.GSTIFedit,'String'));
GSACF = str2double(get(handles.GSACFedit,'String'));
GSTCF = str2double(get(handles.GSTCFedit,'String'));
GSF = str2double(get(handles.GSFedit,'String'));
GRACF = str2double(get(handles.GRACFedit,'String'));
GRTCF = str2double(get(handles.GRTCFedit,'String'));
GRS = str2double(get(handles.GRSedit,'String'));
RMod = str2double(get(handles.RModedit,'String'));
vFA = str2double(get(handles.vFedit,'String'));
vFB = str2double(get(handles.vBedit,'String'));
ro = str2double(get(handles.ROedit,'String'));

%Calculations
%Run Sun Chen Model for both fiber volume ratios
VF = vFA;
icanstr2_SAE
QAg = Cequiv;
VF = vFB;
icanstr2_SAE
QBg = Cequiv;

%Run Compliance Calcs
compliance
ea(1)=E11A;%Modulus of Elasticity in 11 for Subcell A
eb(1)=E22A;%Modulus of Elasticity in 22 for Subcell A
gab(1)=G12A;%Shear Modulus for Subcell A
nu21A=E22A/E11A*nu12A;
prba(1) = nu21A;%Poisson's Ratio for Subcell A
ea(2)=E11B;%Modulus of Elasticity in 11 for Subcell B
eb(2) = E22B;%Modulus of Elasticity in 22 for Subcell B
gab(2) = G12B;%Shear Modulus for Subcell B
nu21B=E22B/E11B*nu12B;
prba(2) = nu21B;%Poisson's Ratio for Subcell B

Figure A.2.2: I.H.S. calc_Input_SAE.m Source Code
% Run Layer Level Calculations

% For XT A

gStress = GSATF;
axial_t_LL
xt(1) = sig1la;
% For XC A
gStress = GSACF;
axial_t_LL
xc(1) = sig1la;
% For YT A
yt(1) = GSTTF;
% For YC A
gStress = GSTCF;
trans_t_LL
yc(1) = sig22a;
% For SC A
sc(1) = GSSF;
% For E22T A
gStress = GSTTF;
trans_t_LLzero
e22t(1) = e22a;
e22t(2) = e22b;

% For XT B
gStress = GSATS;
axial_t_LL
xt(2) = sig1lb;
% For XC B
gStress = GSACF;
axial_t_LL
xc(2) = sig1lb;
% For YT B
yt(2) = GSTTF;
% For YC B
gStress = GSTCF;
trans_t_LL
yc(2) = sig22b;
% For SC B
sc(2) = GSSF;

% Correction for Slim

xt(1) = 2*GSATF - xt(2);

% Calculate Strains
% For E11T A & B
e11t(1) = GRAIT;
e11t(2) = e11t(1);

Figure A.2.2: I.H.S. calc_Input_SAE.m Source Code (Continued)
%For E22C A & B
98  gStress = 1;
99  trans_c_LL
100  e22C(1) = 2*e22a/(e22a+e22b)*GRTCF;
101  e22C(2) = 2*e22b/(e22a+e22b)*GRTCF;
102
103 %For GMS A & B
104  gStress = 1;
105  shear_LL
106  gms(1) = 2*e66a/(e66a+e66b)*GRSF;
107  gms(2) = 2*e66b/(e66a+e66b)*GRSF;
108
109   for i=1:2
110
111     %For E11C A & B
112     e11C(i) = GRACF;
113  end

Figure A.2.2: I.H.S. calc_Input_SAE.m Source Code (Continued)
%calc_Input_SI.m

% Runs Input Calculations for braided composite material
% in SAE units
%
% Created: 1-19-10
% By: Brina Blinzler (University of Akron)
% Last Edited: 5-28-11

warning off;

% GUI Inputs
GSAF = str2double(get(handles.GSAFedit,'String'));
GSAI = str2double(get(handles.GSAIedit,'String'));
GSTIF = str2double(get(handles.GSTIFedit,'String'));
GSAIC = str2double(get(handles.GSAICedit,'String'));
GSTIC = str2double(get(handles.GSTICedit,'String'));
GSSY = str2double(get(handles.GSSYedit,'String'));
GRACF = str2double(get(handles.GRACFedit,'String'));
GRICF = str2double(get(handles.GRICFedit,'String'));
GRSF = str2double(get(handles.GRSFedit,'String'));
RMOD = str2double(get(handles.RMODedit,'String'));
vfA = str2double(get(handles.vfAedit,'String'));
vfB = str2double(get(handles.vfBedit,'String'));
ro = str2double(get(handles.ROedit,'String'));

% Calculations
% Run Sun Chen Model for both fiber volume ratios
VF = vfA;
icestr2_SI
QAg = Cequiv;
VF = vfB;
icestr2_SI
QAg = Cequiv;

% Run Compliance Calcs
compliance
ea(1)=E11A;% Modulus of Elasticity in 11 for Subcell A
eb(1)=E22A;% Modulus of Elasticity in 22 for Subcell A
gab(1)=G12A;% Shear Modulus for Subcell A
nu21A=E22A/E11A*nu12A;
prba(1) = nu21A;% Poisson's Ratio for Subcell A
ea(2)=E11B;% Modulus of Elasticity in 11 for Subcell B
eb(2)=E22B;% Modulus of Elasticity in 22 for Subcell B
gab(2) = G12B;% Shear Modulus for Subcell B
nu21B=E22B/E11B*nu12B;
prba(2) = nu21B;% Poisson's Ratio for Subcell B

Figure A.2.3: I.H.S. calc_Input_SI.m Source Code
%Run Layer Level Calculations

%For XT A

\text{gStress} = \text{GSAIT};
\text{axial}_t\_LL
\text{xt}(1) = \text{sig}11a;

%For XC A

\text{gStress} = \text{GSACF};
\text{axial}_t\_LL
\text{xc}(1) = \text{sig}11a;

%For YT A

\text{yt}(1) = \text{GSTTF};

%For YC A

\text{gStress} = \text{GSTCF};
\text{trans}_t\_LL
\text{yc}(1) = \text{sig}22a;

%For SC A

\text{sc}(1) = \text{GSSF};

%For E22T A

\text{gStress} = \text{GSTTF};
\text{trans}_t\_LLzero
\text{e22t}(1) = \text{e}22a;
\text{e22t}(2) = \text{e}22b;

%For XT B

\text{gStress} = \text{GSATS};
\text{axial}_t\_LL
\text{xt}(2) = \text{sig}11b;

%For XC B

\text{gStress} = \text{GSACF};
\text{axial}_t\_LL
\text{xc}(2) = \text{sig}11b;

%For YT B

\text{yt}(2) = \text{GSTTF};

%For YC B

\text{gStress} = \text{GSTCF};
\text{trans}_t\_LL
\text{yc}(2) = \text{sig}22b;

%For SC B

\text{sc}(2) = \text{GSSF};

%Correction for Slim

\text{xt}(1) = 2*\text{GSAIT} - \text{xt}(2);

%Calculate Strains

%For E11T A & B

\text{e11t}(1) = \text{xt}(1)/\text{ea}(1);
\text{e11t}(2) = \text{e11t}(1);
```matlab
% For E22C A & B
trans_t_LL

% For GMS A & B
shcar_LL

for i=1:2
    % For E11C A & B
    e11C(i) = GRACF;
end
```

Figure A.2.3: I.H.S. calc_Input_SI.m Source Code (Continued)
A.2.4  icanstr2_SAE.m

% icanstr2_SAE.m
% ICANSTR Micro Mechanics Model (Author: Rob Goldberg)
% Triaxially Braided Composite: T700/PFR20
% 
% Created By: Brina Blinzler
% Date Created: 11/5/10
% Modified By: Brina Blinzler
% Date Modified: 12/20/10

% Fiber Properties
% SAE
E11f = 3.33E7;
E22f = 2.10E6;
u12f = 0.2;
G12f = 3.48E6;

% Matrix Properties
% SAE
Em = RMod;
um = 0.363;
Gm = Em/(2*(1+num));

% Vf should already be declared

% Compute Compliance Matrix for Fiber and Matrix
nu23f = nu12f;
G23f = G12f;

S11f = 1/E11f;
S22f = 1/E22f;
S12f = -nu12f/E11f;
S23f = -nu23f/E22f;
S44f = 1/G23f;
S66f = 1/G12f;

S11m = 1/Em;
S12m = -num/Em;
S66m = 1/Gm;

% Compute Volume and Height Fraction for Each Slice
nfbdiv = 13;% Number of slices
df = sqrt(4*Vf/p1);
rf = df/2;
nfb = floor(nfbdiv/2+1);
nfb1 = nfb+1;
tfdiv = df/nfbdiv;
sumh = 0;

Figure A.2.4: I.H.S. icanstr2_SAE.m Source Code
%Compute Y Coordinates for Slice
for i=1:nfb,
    if i==nfb
        yr(i) = rf;
    else
        yr(i) = ((i-1)+ 0.5)*tfdiv;
    end
end

%Compute Areas and Heights for Slice
for i=1:nfb,
    if i==nfb
        xnt(i) = VF/4;
    else
        xnt(i) = 0.5*(yr(i)*sqrt(VF/pi-yr(i)^2)+VF/pi*asin(yr(i)*sqrt(pi/VF))
    end
end
if i==1
    ar(i) = xnt(i);
    ht(i) = yr(i);
else
    ar(i) = xnt(i)-xnt(i-1);
    ht(i) = yr(i)-yr(i-1);
end

%Compute Volume and Height Fraction for Slice
for i=1:nfb,
    ak(i) = ar(i)/(ht(i)/2);
    hf(i) = ht(i)*2;
    sumh = sumh+hf(i);
end

%Compute Volume and Height Fraction for Matrix Only Slice
ak(nfb1) = 0;
hf(nfb1) = 1-sumh;

%Determine Terms in Coefficient Matrix
for i=1:nfb,
    frac = ak(i);
    subican
    Qmat(:,;:i) = Qmat;
end

Figure A.2.4: I.H.S. icanstr2_SAE.m Source Code (Continued)
%Compute Effective Elastic Constants for Matrix Layer
S11P(nfb1) = 1/Em;
S22P(nfb1) = 1/Em;
S12P(nfb1) = -num/Em;
S21P(nfb1) = -num/Em;
S66P(nfb1) = 1/Gm;
Smatm = [S11P(nfb1) S12P(nfb1) 0; S21P(nfb1) S22P(nfb1) 0; 0 0 S66P(nfb1)];
Qmat(:, :, nfb1) = inv(Smatm);

%Compute Total Stiffness Matrix
Q11 = 0;
Q12 = 0;
Q21 = 0;
Q22 = 0;
Q66 = 0;

for i=1:nfb1
    Q11 = Q11 + Qmat(1,1,i)*hf(i);
    Q12 = Q12 + Qmat(1,2,i)*hf(i);
    Q21 = Q21 + Qmat(2,1,i)*hf(i);
    Q22 = Q22 + Qmat(2,2,i)*hf(i);
    Q66 = Q66 + Qmat(3,3,i)*hf(i);
end

Qtot = [Q11 Q12 0; Q21 Q22 0; 0 0 Q66];

%C0mpliance Matrix
Cequiv=Qtot;

Figure A.2.4: I.H.S. icanstr2_SAE.m Source Code (Continued)
\begin{verbatim}
%icanstr2_SI.m
%ICANSTR Micro Mechanics Model (Author: Rob Goldberg)
%Triaxially Braided Composite: T700/PR320
%
%Created By: Brina Blinzler
%Date Created: 11/5/10
%Modified By: Brina Blinzler
%Date Modified: 1/4/11

%Fiber Properties
%SI
E11f = 230;
E22f = 15;
u12f = 0.2;
G12f = 27;

%Matrix Properties
%SI
Em = RMod;
um = 0.363;
Gm = Em/(1/(1+num));

%Vf should already be declared

%Compute Compliance Matrix for Fiber and Matrix
nu23f = nu12f;
G23f = G12f;

S11f = 1/E11f;
S22f = 1/E22f;
S12f = -nu12f/E11f;
S23f = -nu23f/E22f;
S44f = 1/G23f;
S66f = 1/G12f;

S11m = 1/Em;
S12m = -num/Em;
S66m = 1/Gm;

%Compute Volume and Height Fraction for Each Slice
nfbdiv = 13;%Number of slices
df = sqrt(4*Vf/pi);
rf = df/2;
nfb = floor(nfbdiv/2+1);
nEb1 = nfb+1;
tfdiv = df/nfbdiv;
sumh = 0;
\end{verbatim}

Figure A.2.5: I.H.S. icanstr2_SI.m Source Code
Figure A.2.5: I.H.S. icanstr2_SI.m Source Code (Continued)
%Compute Effective Elastic Constants for Matrix Layer

S11P(nfb1) = 1/Em;
S22P(nfb1) = 1/Em;
S12P(nfb1) = -num/Em;
S21P(nfb1) = -num/Em;
S66P(nfb1) = 1/Gm;

Smatm = [S11P(nfb1) S12P(nfb1) 0;S21P(nfb1) S22P(nfb1) 0;0 0 S66P(nfb1)];
Qmat(:, :, nfb1) = inv(Smatm);

%Compute Total Stiffness Matrix
Q11 = 0;
Q12 = 0;
Q21 = 0;
Q22 = 0;
Q66 = 0;

for i = 1:nfb1
    Q11 = Q11 + Qmat(:, i, i)*hf(i);
    Q12 = Q12 + Qmat(:, i, 1)*hf(i);
    Q21 = Q21 + Qmat(:, i, 1)*hf(i);
    Q22 = Q22 + Qmat(:, i, 1)*hf(i);
    Q66 = Q66 + Qmat(:, i, 1)*hf(i);
end

Qtot = [Q11 Q12 0;Q21 Q22 0;0 0 Q66];

%Compliance Matrix
Cequiv=Qtot;

Figure A.2.5: I.H.S. icanstr2_SI.m Source Code (Continued)
%subican.m

%Square Cell Micro Mechanics Model
%Triaxially Braided Composite
%
%Created By: Brina Blinzler
%Date Created: 11/4/10
%Modified By: Brina Blinzler
%Date Modified: 11/10/10

%Vf should already be declared
vf = frac;

%Compute Effective Elastic Properties
%Determine Terms in Coefficient Matrix
Vm = 1-vf;
dtrm = S12f+S11m*(vf/Vm);
A1 = S11f-(S12f^2)/dtrm;
A2 = S12f*S12m/dtrm;
A3 = S12f+S12f*(S12m-S23f)/dtrm;
A4 = (vf/Vm)*S12f*S12m/dtrm;
A5 = S11m-(vf/Vm)*(S12m^2)/dtrm;
A6 = S12m-(vf/Vm)*S12m*(S12m-S23f)/dtrm;
A7 = vf*(S12f-S12f*(S23f-S12m)/dtrm);
A8 = Vm*S12m+vf*S12m*(S23f-S12m)/dtrm;
A9 = vf*S22f+Vm*S11m+vf*(S23f-S12m)*(S12m-S23f)/dtrm;
Amat = [A1 A2 A3;A4 A5 A6;A7 A8 A9];
Ainv = inv(Amat);

%Determine Inverse Compliance Terms
Q11 = vf*(Ainv(1,1)+Ainv(1,2))+Vm*(Ainv(2,1)+Ainv(2,2));
Q12 = vf*Ainv(1,3)+Vm*Ainv(2,3);
Q21 = Ainv(3,1)+Ainv(3,2);
Q22 = Ainv(3,3);
Q66 = 1/(vf/G12f+Vm/Gm);
Qmats = [Q11 Q12 0;Q21 Q22 0;0 0 Q66];
Smat = inv(Qmats);

Figure A.2.6: I.H.S. subican.m Source Code
%compliance.m
%Classical Lamination Theory
%applied to a Triaxially Braided Composite
%
%Created: 5-19-10
%By: Brina Blinzler (University of Akron)
%Last Edited: 2-17-11
%
% subcell layout

| A | B | C | D |

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>+60</td>
<td>+60</td>
<td>+60</td>
<td>+60</td>
</tr>
<tr>
<td>0</td>
<td>-60</td>
<td>0</td>
<td>-60</td>
</tr>
<tr>
<td>-60</td>
<td>-60</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
%

warning off;

%Input Stiffness Matrix For Layers
Q11A=QAg(1,1);
Q12A=QAg(1,2);
Q22A=QAg(2,2);
Q66A=QAg(3,3);
Q11B=QBg(1,1);
Q12B=QBg(1,2);
Q22B=QBg(2,2);
Q66B=QBg(3,3);
QA=[Q11A,Q12A,0;Q12A,Q22A,0;0,0,Q66A];%0.8 Vf axial ply
QB=[Q11B,Q12B,0;Q12B,Q22B,0;0,0,Q66B];%0.5 Vf bias plys

m = cos(pi*60/180);
np = sin(pi*60/180);
nn = sin(pi*(-60)/180);

Tp = [m^2,np^2,2*m*np;np^2,-m^2,-2*m*np;0,0,m^2-4*np^2];
Tinv = inv(Tp);
Tptran = transpose(Tp);
Tn = [m^2,nn^2,2*m*nn;nn^2,-m^2,-2*m*nn;0,0,m^2-4*nn^2];
Tinv = inv(Tn);
Tntran = transpose(Tn);
%
%Stiffness Matrix for Lamina
%axial ply
Qbar0A = QA;
Qbar60A = Tpinv*QA*Tp;
Qbar60A = Tninv*QA*Tn;

Figure A.2.7: I.H.S. compliance.m Source Code
\texttt{%bias plos}
\texttt{Qbarp60B=Tpinv*QB*Tptran;}
\texttt{Qbarn60B=Tninv*QB*Tntran;}
\texttt{\%
\texttt{%Stiffness and Compliance Matrix for Composite - double weight}
\texttt{A11a=Qbarp60B(1,1)*0.25+Qbar0A(1,1)*0.5+Qbarn60B(1,1)*0.25;}
\texttt{A12a=Qbarp60B(1,2)*0.25+Qbar0A(1,2)*0.5+Qbarn60B(1,2)*0.25;}
\texttt{A22a=Qbarp60B(2,2)*0.25+Qbar0A(2,2)*0.5+Qbarn60B(2,2)*0.25;}
\texttt{A66a=Qbarp60B(3,3)*0.25+Qbar0A(3,3)*0.5+Qbarn60B(3,3)*0.25;}
\texttt{A11B=Qbarp60B(1,1)*0.5+Qbarn60B(1,1)*0.5;}
\texttt{A12B=Qbarp60B(1,2)*0.5+Qbarn60B(1,2)*0.5;}
\texttt{A22B=Qbarp60B(2,2)*0.5+Qbarn60B(2,2)*0.5;}
\texttt{A66B=Qbarp60B(3,3)*0.5+Qbarn60B(3,3)*0.5;}
\texttt{\%
\texttt{%Stiffness Properties}
\texttt{A=[A11a,A12a,0;A12a,A22a,0;0,0,A66a];}
\texttt{SA=inv(AA);}
\texttt{E11A=1/SA(1,1);}
\texttt{nu12A=-SA(1,2)*E11A;}
\texttt{E22A=1/SA(2,2);}
\texttt{G12A=1/SA(3,3);}
\texttt{AB=[A11B,A12B,0;A12B,A22B,0;0,0,A66B];}
\texttt{SB=inv(AB);}
\texttt{E11B=1/SB(1,1);}
\texttt{nu12B=-SB(1,2)*E11B;}
\texttt{E22B=1/SB(2,2);}
\texttt{G12B=1/SB(3,3);}

Figure A.2.7: I.H.S. compliance.m Source Code (Continued)
A.2.8  axial_t_LL.m

```matlab
%axial_t_LL.m
%Micromechanical Calculations for stress and strain
%Triaxially Braided Composite
%Axial Tension Test

% Created: 6-4-09
% By: Erina Blinzler (University of Akron)
% Last Edited: 12-20-10

% subcell layout
% | ==|==|==|== |
% | A | B | C | D |
% |==|==|==|== |
% +60 +60 +60 +60
% 0 -60 0 -60
% -60 -60

warning off;

vfa = 0.5;
vfb = 0.5;
%variable declarations
sig11 = 0; % axial stress
sig22 = 0; % transverse stress
sig66 = 0; % shear stress
e11 = 0; % axial strain
e22 = 0; % transverse strain
sig11c = 0; % axial stress in subcell a
sig11b = 0; % axial stress in subcell b
vfc = vfa; % volume fraction of subcell c
sig11c = 0; % axial stress in subcell c
vfd = vfb; % volume fraction of subcell d
sig11d = 0; % axial stress in subcell d
e22a = 0; % transverse strain in subcell a
e22b = 0; % transverse strain in subcell b
e22c = 0; % transverse strain in subcell c
e22d = 0; % transverse strain in subcell d

% Compliance Matrix for Lamina
S11a=SA(1,1);
S12a=SA(1,2);
S22a=SA(2,2);
S66a=SA(3,3);


S11B=SB(1,1);
S12B=SB(1,2);
S22B=SB(2,2);
S66B=SB(3,3);
```

Figure A.2.8: I.H.S. axial_t_LL.m Source Code
%Initialization
sig11 = gStress;
sig22 = 0;
sig66 = 0;
A = [-pi/3, 0, pi/3];
B = [-pi/3, pi/3];
C = [pi/3, 0, -pi/3];
D = [pi/3, -pi/3];
subcell = [A, B, C, D];
% Calculations
% Combine subcell Calcs
sig22a = sig22;
sig22b = sig22;
sig22c = sig22;
sig22d = sig22;
sig11a = sig11/(0.5*(1+S11a/S11B));
sig11b = S11a/S11B*sig11a;
sig11c = sig11a;
sig11d = sig11b;
e11a = S11a*sig11a;
e11 = e11a;
e11b = S11B*sig11b;
e11c = e11a;
e11d = e11b;
e22a = S12a*sig11a;
e22b = S12B*sig11b;
e22c = e22a;
e22d = e22b;
SigTotA = [sig11a; sig22a; sig66]; %global stresses
SigTotB = [sig11b; sig22b; sig66]; %global stresses
SigTotC = [sig11c; sig22c; sig66]; %global stresses
SigTotD = [sig11d; sig22d; sig66]; %global stresses
EpsTotA = [e11a; e22a; 0]; %global strains
EpsTotB = [e11b; e22b; 0]; %global strains
EpsTotC = [e11c; e22c; 0]; %global strains
EpsTotD = [e11d; e22d; 0]; %global strains

Figure A.2.8: I.H.S. axial_t_LL.m Source Code (Continued)
A.2.9 trans_t_LL.m

```matlab
%trans_t_LL.m
%Micromechanical Calculations for stress and strain
%Triaxially Braided Composite
%Transverse Tension Test
%
%Created: 6-4-09
%By: Brina Blinzler (University of Akron)
%Last Edited: 12-20-10
%
% subcell layout
%|---|---|---|---|
%| A | B | C | D |
%|---|---|---|---|
% +60 +60 +60 +60
% 0 -60 0 -60
% -60 -60
%
warning off;

vfa = 0.5;
vfb = 0.5;

%variable declarations
sig11 = 0; %axial stress
sig22 = 0; %transverse stress
sig66 = 0; %shear stress
c11 = 0; %axial strain
e22 = 0; %transverse strain
sig1a = 0; %axial stress in subcell a
sig1b = 0; %axial stress in subcell b
vfc = vfa; %volume fraction of subcell c
sig1c = 0; %axial stress in subcell c
vfd = vfb; %volume fraction of subcell d
e22a = 0; %transverse strain in subcell a
e22b = 0; %transverse strain in subcell b
e22c = 0; %transverse strain in subcell c
e22d = 0; %transverse strain in subcell d
%
%Compliance Matrix for Lamina
S11a=SA(1,1);
S12a=SA(1,2);
S22a=SA(2,2);
S66a=SA(3,3);
%
S11b=SB(1,1);
S12b=SB(1,2);
S22b=SB(2,2);
S66b=SB(3,3);
```

Figure A.2.9: I.H.S. trans_t_LL.m Source Code
%Initialization

sig22 = gStress;
sig66=0;
A = [-pi/3, 0, pi/3];
B = [-pi/3, pi/3];
C = [pi/3, 0, -pi/3];
D = [pi/3, -pi/3];
subcell = [A, B, C, D];

% Calculations

% Combine subcell Calcs

sig22a = sig22;
sig22b = sig22;
sig22c = sig22;
sig22d = sig22;
sig11a = (S12B - S12a)/(S11a+S11B)*sig22;
sig11b = -1*sig11a;
sig11c = sig11a;
sig11d = sig11b;
e11a = S11a*sig11a+S12a*sig22;
e11b = S11B*sig11b+S12B*sig22;
e11c = e11a;
e11d = e11b;
e22a = S12a*sig11a+S22a*sig22;
e22b = S12B*sig11b+S22B*sig22;
e22c = e22a;
e22d = e22b;

SigTotA = [sig11a; sig22a; sig66]; %global stresses
SigTotB = [sig11b; sig22b; sig66]; %global stresses
SigTotC = [sig11c; sig22c; sig66]; %global stresses
SigTotD = [sig11d; sig22d; sig66]; %global stresses
EpsTotA = [e11a; e22a; 0]; %global strains
EpsTotB = [e11b; e22b; 0]; %global strains
EpsTotC = [e11c; e22c; 0]; %global strains
EpsTotD = [e11d; e22d; 0]; %global strains

Figure A.2.9: I.H.S. trans_t_LL.m Source Code (Continued)
A.2.10 trans_t_LLzero.m

```matlab
%trans_t_LLzero.m
%Micromechanical Calculations for stress and strain
%Triaxially Braided Composite
%Transverse Tension Test
%
%Created: 6-4-09
%BY: Brina Blinzler (University of Akron)
%Last Edited: 12-20-10
%
% subcell layout
%|---|---|---|---|
| A | B | C | D |
%|---|---|---|---|
% 60 60 60 60 60
% 0 -60 0 -60
% -60 -60
%
warning off;

vfa = 0.5;
vfb = 0.5;

% variable declarations
sig11 = 0; % axial stress
sig22 = 0; % transverse stress
sig66 = 0; % shear stress
e11 = 0; % axial strain
e22 = 0; % transverse strain
sig11a = 0; % axial stress in subcell a
sig11b = 0; % axial stress in subcell b
vfc = vfa; % volume fraction of subcell c
sig11c = 0; % axial stress in subcell c
vfd = vfb; % volume fraction of subcell d
sig11d = 0; % axial stress in subcell d
e22a = 0; % transverse strain in subcell a
e22b = 0; % transverse strain in subcell b
e22c = 0; % transverse strain in subcell c
e22d = 0; % transverse strain in subcell d
%
% Compliance Matrix for lamina
S11a=5A(1,1);
S12a=SA(1,2);
S22a=SA(2,2)*2;
S66a=SA(3,3);
%
S11B=SB(1,1);
S12B=SB(1,2);
S22B=SB(2,2);
S66B=SB(3,3);
```

Figure A.2.10: I.H.S. trans_t_LLzero.m Source Code
Figure A.2.10: I.H.S. trans_t_LLzero.m Source Code (Continued)
%shear_LL.m

%Investigation of material parameters as part of the diagnostic of the
%LS-DYNA material model MAT158 used to simulate a triaxially braided carbon
%-epoxy composite

%I700/FR520
%Axial Tention Test

% Created: 6-4-09
% By: Brina Blinzler (University of Akron)
% Last Edited: 5-11-10

% subcell layout
% |-----|-----|-----|-----|
% | A   | B   | C   | D   |
% |-----|-----|-----|-----|
% +60  +60  +60  +60
% 0   -60  0   -60
% -60  -60

warning off;

vfa = 0.5;
vhb = 0.5;

%variable declarations
sig11 = 0; %axial stress
sig22 = 0; %transverse stress
sig66 = 0; %shear stress
e11 = 0; %axial strain
e22 = 0; %transverse strain
sig1a = 0; %axial stress in subcell a
sig1b = 0; %axial stress in subcell b
vfc = vfa; %volume fraction of subcell c
sig1c = 0; %axial stress in subcell c
vfd = vhb; %volume fraction of subcell d
sig1d = 0; %axial stress in subcell d
e22a = 0; %transverse strain in subcell a
e22b = 0; %transverse strain in subcell b
e22c = 0; %transverse strain in subcell c
e22d = 0; %transverse strain in subcell d

%Compliance Matrix for Lamina
S11A=5A(1,1);
S12A=5A(1,2);
S22A=5A(2,2);
S66A=5A(3,3);

Figure A.2.11: I.H.S. shear_LL.m Source Code
```matlab
% Initialization
sig66 = qStress;
sig11=0;
sig22=0;
A = [-pi/3,0,pi/3];
B = [-pi/3,pi/3];
C = [pi/3,0,-pi/3];
D = [pi/3,-pi/3];
subcell = [A, B, C, D];
% Calculations
% Combine subcell Calcs
sig22a = sig22;
sig22b = sig22;
sig22c = sig22;
sig22d = sig22;
sig11a = sig11;
sig11b = sig11;
sig11c = sig11a;
sig11d = sig11b;
sig66a = sig66;
sig66b = sig66;
sig66c = sig66;
sig66d = sig66;
e11 = 0;
e11a = e11;
e11b = e11;
e11c = e11;
e11d = e11;
e22=0;
e22a = e22;
e22b = e22;
e22c = e22a;
e22d = e22b;
e66a = S66a*sig66a;
e66b = S66b*sig66b;
e66c = S66a*sig66c;
e66d = S66b*sig66d;
SigTotA = [sig11a; sig22a; sig66]; %global stresses
SigTotB = [sig11b; sig22b; sig66]; %global stresses
SigTotC = [sig11c; sig22c; sig66]; %global stresses
SigTotD = [sig11d; sig22d; sig66]; %global stresses
EpsTotA = [e11a; e22a; e66a]; %global strains
EpsTotB = [e11b; e22b; e66b]; %global strains
EpsTotC = [e11c; e22c; e66c]; %global strains
EpsTotD = [e11d; e22d; e66d]; %global strains
```

Figure A.2.11: I.H.S. shear_LL.m Source Code (Continued)
/*writeInputCard.m

Writes input card formatted for LS-DYNA *Mat_50 or *Mat_150
%
%Created: 1-19-10
%By: Brina Blinzler (University of Akron)
%Last Edited: 1-24-10
%
%Variables
mid = [1,2];
slim = zeros(2,5);%in order {slimcl1,slimcl2,slimnt2,slimmc2,slims}
%slim(1,3)= 1;
slim(2,1) = 1;
k = 6.762e6;
mat = zeros(16,8);

for i=1:2
    %row 1
    mat(i*8-7,1) = mid(i);
    mat(i*8-7,2) = ro;
    mat(i*8-7,3) = ea(i);
    mat(i*8-7,4) = eb(i);
    mat(i*8-7,6) = prte(i);
    %row 2
    mat(i*8-6,1) = gab(i);
    mat(i*8-6,4) = slim(1,1);
    mat(i*8-6,5) = slim(1,2);
    mat(i*8-6,6) = slim(1,3);
    mat(i*8-6,7) = slim(1,4);
    mat(i*8-6,8) = slim(1,5);
    %row 3
    mat(i*8-5,1) = 2;
    mat(i*8-5,3) = 1;
    mat(i*8-5,5) = -1;
    %row 4
    mat(i*8-4,5) = 1;
    %row 5
    mat(i*8-3,4) = 1;
    %row 6
    mat(i*8-2,1) = el1c(i);
    mat(i*8-2,2) = el1t(i);
    mat(i*8-2,3) = el2c(i);
    mat(i*8-2,4) = el2t(i);
    mat(i*8-2,5) = gms(i);
    %row 7
    mat(i*8-1,1) = xc(i);
    mat(i*8-1,2) = xt(i);
    mat(i*8-1,3) = yc(i);
    mat(i*8-1,4) = yt(i);
    mat(i*8-1,5) = sc(i);
    %row 8
    mat(i*8,1) = k;
end

Figure A.2.12: I.H.S. writeInputCard.m Source Code
Figure A.2.12: I.H.S. writeInputCard.m Source Code (Continued)
A.2.13 writeInputExcel.m

```matlab
1 %writeInputExcel.m
2 %Write Inputs to Excel Spreadsheet
3 %
4 %Created: 6-4-09
5 %By: Brine Blinzler (University of Akron)
6 %Last Edited: 6-4-09
7 excelName = ['materialInputs.xls'];
8 if exist (excelName, 'file')
9    delete (excelName);
10 end
11
12 title = {'Inputs for Micromechanics Approach'};
13 header = {'AC','BD'};
14 for i=1:2
15    colMatrix(:,i)=[mid(i);ro/ea(i);db(i);gab(i);prba(i);ell1C(i);ell1(L);xt(i);e22C(i);
16    yc(i);e22T(i);yt(i);gms(i);mc(i);slim1(1,2);slim1(1,3);slim1(1,4);slim1(1,5)];
17 end
18 titleMatrix = {'' ;'HID' ;'RO' ;'EA' ;'EB' ;'GAB' ;'PRBA' ;'ELL1C' ;'ELL1(L)' ;'XT' ;'E22C' ;'YC' ;'E22T' ;
19    'YT' ;'GMS' ;'MC' ;'SLIM1' ;'SLIM1(L)' ;'SLIM1(B)' ;'SLIM1(C)' ;'
20 end
21 xlswrite('materialInputs', title, 'Results', 'B2');
22 xlswrite('materialInputs', titleMatrix, 'Results', 'B4');
23 xlswrite('materialInputs', header(1), 'Results', 'C6');
24 xlswrite('materialInputs', colMatrix(:,1), 'Results', 'C9');
25 xlswrite('materialInputs', header(2), 'Results', 'E4');
26 xlswrite('materialInputs', colMatrix(:,2), 'Results', 'E9');
```

Figure A.2.13: I.H.S. writeInputExcel.m Source Code
function varargout = input_calc(varargin)

% INPUT_CALC M-file for input_calc.fig
% INPUT_CALC, by itself, creates a new INPUT_CALC or raises the existing
% singleton*.
% 
% H = INPUT_CALC returns the handle to a new INPUT_CALC or the handle to
% the existing singleton*.
% 
% INPUT_CALC('CALLBACK', hObject, eventdata, handles,...) calls the local
% function named CALLBACK in INPUT_CALC.M with the given input arguments.
% 
% INPUT_CALC('Property', 'Value', ...) creates a new INPUT_CALC or raises the
% existing singleton*. Starting from the left, property value pairs are
% applied to the GUI before input_calc_OpeningFcn gets called. An
% unrecognized property name or invalid value makes property application
% stop. All inputs are passed to input_calc_OpeningFcn via varargin.
% 
% *See GUI Options on GUIDE's Tools menu. Choose "GUI allows only one
% instance to run (singleton)".
%
% See also: GUIDE, GUIDATA, GUIDATA

% Edit the above text to modify the response to help input_calc

% Last Modified by GUIDE v2.5 28-May-2011 15:19:29

% Begin initialization code - DO NOT EDIT

% gui_Singleton - 1;
varargout = struct('gui_Name', mfilename, ...'
    'gui_Singleton', gui_Singleton, ...
    'gui_OpeningFcn', @input_calc_OpeningFcn, ...
    'gui_OutputFcn', @input_calc_OutputFcn, ...
    'gui_LayoutFcn', [], ..., ...
    'gui_Callback', []);

if nargin && ischar(varargin{1})
    gui_State.gui_Callback = str2func(varargin{1});
end

if nargin
    varargout{1:nargout} = gui_mainfcn(gui_State, varargin{});
else
    gui_mainfcn(gui_State, varargin{1});
end
% End initialization code - DO NOT EDIT

% --- Executes just before input_calc is made visible.
function input_calc_OpeningFcn(hObject, eventdata, handles, varargin)
% This function has no output args, see OutputFcn.
% hObject    handle to figure

% eventdata  reserved - to be defined in a future version of MATLAB
% handles  structure with handles and user data (see GUIDATA)
% varargin  command line arguments to input_calc (see VARARGIN)

% Choose default command line output for input_calc
handles.output = hObject;

% Update handles structure
guider(hObject, handles);

% UIWAIT makes input_calc wait for user response (see UIRESUME)
uiwait(handles.figure1);

% --- Outputs from this function are returned to the command line.
function varargout = input_calc_OutputFcn(hObject, eventdata, handles)
% varargout  cell array for returning output args (see VARARGOUT);
% hObject    handle to figure
% eventdata  reserved - to be defined in a future version of MATLAB
% handles    structure with handles and user data (see GUIDATA)

% Get default command line output from handles structure
varargout{1} = handles.output;

% --- Executes on button press in create.
function create_Callback(hObject, eventdata, handles)
% hObject    handle to create (see GCBO)
% eventdata  reserved - to be defined in a future version of MATLAB
% handles    structure with handles and user data (see GUIDATA)

if(get(handles.uni_option,'Value') == 1)
calc_Input_SAE;
else
  %calc_Input_SI;
end

writeInputCard;
writeInputExcel;

function GSATFedit_Callback(hObject, eventdata, handles)
% hObject    handle to GSATFedit (see GCBO)
% eventdata  reserved - to be defined in a future version of MATLAB
% handles    structure with handles and user data (see GUIDATA)

% Hints: get(hObject,'String') returns contents of GSATFedit as text;
%        str2double(get(hObject,'String')) returns contents of GSATFedit as a
%        % --- Executes during object creation, after setting all properties.

Figure A.2.14: I.H.S. input_calc.m Source Code (Continued)
function GSATEdit_CreateFcn(hObject, eventdata, handles)
    % hObject    handle to GSATEdit (see GCBO)
    % eventdata  reserved - to be defined in a future version of MATLAB
    % handles    empty - handles not created until after all CreateFcns called

    % Hint: edit controls usually have a white background on Windows.
    % See ISPC and COMPUTER.
    if ispc && isequal(get(hObject,'BackgroundColor'), get(0,'defaultUicontrolBackgroundColor'))
        set(hObject,'BackgroundColor','white');
    end

function GSATEdit_Callback(hObject, eventdata, handles)
    % hObject    handle to GSATEdit (see GCBO)
    % eventdata  reserved - to be defined in a future version of MATLAB
    % handles    structure with handles and user data (see GUIDATA)

    % Hints: get(hObject,'String') returns contents of GSATEdit as text
    str2double(get(hObject,'String')) returns contents of GSATEdit as a double

    % --- Executes during object creation, after setting all properties.
    function GSATEdit_CreateFcn(hObject, eventdata, handles)
    % hObject    handle to GSATEdit (see GCBO)
    % eventdata  reserved - to be defined in a future version of MATLAB
    % handles    empty - handles not created until after all CreateFcns called

    % Hint: edit controls usually have a white background on Windows.
    % See ISPC and COMPUTER.
    if ispc && isequal(get(hObject,'BackgroundColor'), get(0,'defaultUicontrolBackgroundColor'))
        set(hObject,'BackgroundColor','white');
    end

function GSACFEdit_Callback(hObject, eventdata, handles)
    % hObject    handle to GSACFEdit (see GCBO)
    % eventdata  reserved - to be defined in a future version of MATLAB
    % handles    structure with handles and user data (see GUIDATA)

    % Hints: get(hObject,'String') returns contents of GSACFEdit as text
    str2double(get(hObject,'String')) returns contents of GSACFEdit as a double

    % --- Executes during object creation, after setting all properties.
    function GSACFEdit_CreateFcn(hObject, eventdata, handles)
    % hObject    handle to GSACFEdit (see GCBO)
    % eventdata  reserved - to be defined in a future version of MATLAB
    % handles    empty - handles not created until after all CreateFcns called

Figure A.2.14: I.H.S. input_calc.m Source Code (Continued)
Figure A.2.14: I.H.S. input_calc.m Source Code (Continued)
```matlab
set(hObject,'BackgroundColor','white');

function GSTIFedit_Callback(hObject, eventdata, handles)
    % hObject    handle to GSTIFedit (see GCBO)
    % eventdata reserved - to be defined in a future version of MATLAB
    % handles    structure with handles and user data (see GUIDATA)

    % Hint: get(hObject,'String') returns contents of GSTIFedit as text
    % str2double(get(hObject,'String')) returns contents of GSTIFedit as a double

% --- Executes during object creation, after setting all properties.
function GSTIFedit_CreateFcn(hObject, eventdata, handles)
    % hObject    handle to GSTIFedit (see GCBO)
    % eventdata reserved - to be defined in a future version of MATLAB
    % handles    structure with handles and user data (see GUIDATA)

    % Hint: edit controls usually have a white background on Windows.
    % See ISPC and COMPUTER.
    if ispc && isequal(get(hObject,'BackgroundColor'),get(0,'defaultUicontrolBackgroundColor'))
        set(hObject,'BackgroundColor','white');
    end

function GSTISedit_Callback(hObject, eventdata, handles)
    % hObject    handle to GSTISedit (see GCBO)
    % eventdata reserved - to be defined in a future version of MATLAB
    % handles    structure with handles and user data (see GUIDATA)

    % Hint: get(hObject,'String') returns contents of GSTISedit as text
    % str2double(get(hObject,'String')) returns contents of GSTISedit as a double

% --- Executes during object creation, after setting all properties.
function GSTISedit_CreateFcn(hObject, eventdata, handles)
    % hObject    handle to GSTISedit (see GCBO)
    % eventdata reserved - to be defined in a future version of MATLAB
    % handles    structure with handles and user data (see GUIDATA)

    % Hint: edit controls usually have a white background on Windows.
    % See ISPC and COMPUTER.
    if ispc && isequal(get(hObject,'BackgroundColor'),get(0,'defaultUicontrolBackgroundColor'))
        set(hObject,'BackgroundColor','white');
    end
```

Figure A.2.14: I.H.S. input_calc.m Source Code (Continued)
function GSTCFedit_Callback(hObject, eventdata, handles)
% hObject handle to GSTCFedit (see GCBO)
% eventdata reserved - to be defined in a future version of MATLAB
% handles structure with handles and user data (see GUIDATA)

% Hints: get(hObject, 'String') returns contents of GSTCFedit as text
% str2double(get(hObject, 'String')) returns contents of GSTCFedit as a double

% --- Executes during object creation, after setting all properties.
function GSTCFedit_CreateFcn(hObject, eventdata, handles)
% hObject handle to GSTCFedit (see GCBO)
% eventdata reserved - to be defined in a future version of MATLAB
% handles empty - handles not created until after all CreateFcns called

% Hint: edit controls usually have a white background on Windows.
% See ISPC and COMPUTER.
if ispc && isequal(get(hObject,'BackgroundColor'), get(0,'defaultUicontrolBackgroundColor'))
    set(hObject,'BackgroundColor','white');
end

function vfEdit_Callback(hObject, eventdata, handles)
% hObject handle to vfEdit (see GCBO)
% eventdata reserved - to be defined in a future version of MATLAB
% handles structure with handles and user data (see GUIDATA)

% Hints: get(hObject, 'String') returns contents of vfEdit as text
% str2double(get(hObject, 'String')) returns contents of vfEdit as a double

% --- Executes during object creation, after setting all properties.
function vfEdit_CreateFcn(hObject, eventdata, handles)
% hObject handle to vfEdit (see GCBO)
% eventdata reserved - to be defined in a future version of MATLAB
% handles empty - handles not created until after all CreateFcns called

% Hint: edit controls usually have a white background on Windows.
% See ISPC and COMPUTER.
if ispc && isequal(get(hObject,'BackgroundColor'), get(0,'defaultUicontrolBackgroundColor'))
    set(hObject,'BackgroundColor','white');
end

function R0Edit_Callback(hObject, eventdata, handles)
% hObject handle to R0edit (see GCBO)
% eventdata reserved - to be defined in a future version of MATLAB

Figure A.2.14: I.H.S. input_calc.m Source Code (Continued)
% handles  structure with handles and user data (see GUIDATA)

% Hints: get(hObject,'String') returns contents of ROText as text
% str2double(get(hObject,'String')) returns contents of ROText as a double

% --- Executes during object creation, after setting all properties.
function ROText_CreateFcn(hObject, eventdata, handles)
% hObject handle to ROText (see GCBO)
% eventdata reserved - to be defined in a future version of MATLAB
% handles empty - handles not created until after all CreateFuns called

% Hint: edit controls usually have a white background on Windows.
% See ISPC and COMPUTER.
if ispc && isequal(get(hObject,'BackgroundColor'), get(0,'defaultUicontrolBackgroundColor'))
    set(hObject, 'BackgroundColor', 'white');
end

% --- Executes on selection change in unit_option.
function unit_option_Callback(hObject, eventdata, handles)
% hObject handle to unit_option (see GCBO)
% eventdata reserved - to be defined in a future version of MATLAB
% handles structure with handles and user data (see GUIDATA)

% Hints: contents = get(hObject,'String') returns unit_option contents as cell array
% contents(get(hObject,'Value')) returns selected item from unit_option

% --- Executes during object creation, after setting all properties.
function unit_option_CreateFcn(hObject, eventdata, handles)
% hObject handle to unit_option (see GCBO)
% eventdata reserved - to be defined in a future version of MATLAB
% handles empty - handles not created until after all CreateFuns called

% Hint: listbox controls usually have a white background on Windows.
% See ISPC and COMPUTER.
if ispc && isequal(get(hObject,'BackgroundColor'), get(0,'defaultUicontrolBackgroundColor'))
    set(hObject, 'BackgroundColor', 'white');
end

function GRACEdit_Callback(hObject, eventdata, handles)
% hObject handle to GRACEdit (see GCBO)
% eventdata reserved - to be defined in a future version of MATLAB
% handles structure with handles and user data (see GUIDATA)

Figure A.2.14: I.H.S. input_calc.m Source Code (Continued)
Figure A.2.14: I.H.S. input_calc.m Source Code (Continued)
% --- Executes during object creation, after setting all properties.
function EDIT_CreateFcn(hObject, eventdata, handles)
% hObject handle to EDIT (see GCBO)
% eventdata reserved - to be defined in a future version of MATLAB
% handles empty - handles not created until after all CreateFcn's called

% Hint: edit controls usually have a white background on Windows.

% See ISPC and COMPUTER.

if ispc && isequal(get(hObject,'BackgroundColor'), get(0,'defaultUicontrolBackgroundColor'))
    set(hObject,'BackgroundColor','white');
end

function RMedit_Callback(hObject, eventdata, handles)
% hObject handle to RMedit (see GCBO)
% eventdata reserved - to be defined in a future version of MATLAB
% handles structure with handles and user data (see GUIDATA)

% Hint: get(hObject,'String') returns contents of RMedit as text
% str2double(get(hObject,'String')) returns contents of RMedit as a double

% --- Executes during object creation, after setting all properties.
function RMedit_CreateFcn(hObject, eventdata, handles)
% hObject handle to RMedit (see GCBO)
% eventdata reserved - to be defined in a future version of MATLAB
% handles empty - handles not created until after all CreateFcn's called

% Hint: edit controls usually have a white background on Windows.
% See ISPC and COMPUTER.

if ispc && isequal(get(hObject,'BackgroundColor'), get(0,'defaultUicontrolBackgroundColor'))
    set(hObject,'BackgroundColor','white');
end

function GRATedit_Callback(hObject, eventdata, handles)
% hObject handle to GRATedit (see GCBO)
% eventdata reserved - to be defined in a future version of MATLAB
% handles structure with handles and user data (see GUIDATA)

% Hint: get(hObject,'String') returns contents of GRATedit as text
% str2double(get(hObject,'String')) returns contents of GRATedit as a double

% --- Executes during object creation, after setting all properties.
function GRATedit_CreateFcn(hObject, eventdata, handles)
% hObject handle to GRATedit (see GCBO)
% eventdata reserved - to be defined in a future version of MATLAB
% handles empty - handles not created until after all CreateFcn's called

% Hint: edit controls usually have a white background on Windows.
% See ISPC and COMPUTER.

if ispc && isequal(get(hObject,'BackgroundColor'), get(0,'defaultUicontrolBackgroundColor'))
    set(hObject,'BackgroundColor','white');
end

Figure A.2.14: I.H.S. input_calc.m Source Code (Continued)