MULTISCALE HYBRID ELEMENT MODELING OF TRIAXIAL
BRAIDED COMPOSITE

A Dissertation

Presented to

The Graduate Faculty of The University of Akron

In Partial Fulfillment

of the Requirements for the Degree

Doctor of Philosophy

Mingkun Sun

July 2018

MULTISCALE HYBRID ELEMENT MODELING OF
TRIAXIAL BRAIDED COMPOSITE

Mingkun Sun

Dissertation

Approved:

Advisor
Dr. Wieslaw K. Binienda

Committee Member
Dr. Craig Menzemer

Committee Member
Dr. Qindan Huang

Committee Member
Dr. Xiaosheng Gao

Committee Member
Dr. Kevin Kreider

Committee Member
Dr. Robert K. Goldberg

Accepted:

Department Chair
Dr. Wieslaw K. Binienda

Dean of College
Dr. Donald P. Visco Jr.

Dean of the Graduate School
Dr. Chand Midha

Date

ABSTRACT

Composite materials have been widely used in the aerospace industry, automobile manufacturing, and other commercial applications, due to their high strength, light weight and low cost. The two-dimensional triaxial braided composite (2DTBC), one of the widely used types of composite materials, is a quasi-isotropic structure which exhibits equal material stiffness properties in all directions. 2DTBC has been used to manufacture jet engine fan cases and fan blades in the aerospace industry. However, conducting experimental tests of 2DTBC under dynamic impact is time consuming and expensive. Numerical simulation provides an efficient way to investigate composite material response under ballistic impact loading.

The goal of the work in this study is to develop a sophisticated multiscale model to investigate the penetration velocity and panel deflection behavior of a composite structure under various impact velocities as well as to study damage mechanisms under ballistic impact loading conditions. A meso-scale model discretized using three dimensional solid element was generated to account for realistic geometry of the braiding fiber tows and was applied at the critical impact region to account for the impact damage mechanism. Micromechanical finite element models were used to predict the effective material properties for the unidirectional composite. Under the multiscale scheme, a macro-scale model using a simplified geometry in which the braiding architecture of the fiber tows is only accounted for in an approximate manner was incorporated at the region away from the critical region to enhance the computational efficiency of the whole numerical model. The entire multiscale model was created within the context of a finite element program,

Ls-DYNA, which has been widely used by the aerospace industry to simulate bird strikes and jet engine blade-out events due to its reliable explicit time integration algorithm and robust contact algorithm.

In this work, thoroughly investigation involving static tensile test and ballistic impact test was carried out on two composite material systems, T700s/E862 and T700s/PR520. Six-layer straight sided coupon bar tensile tests for both material systems were validated and correlated with experimental results very well. Then the validated multiscale model was applied to investigate the flat panel ballistic impact test. The penetration velocity threshold, damage mechanism under various impact velocity, and the delamination were intensively studied. The numerical analysis results of the two different composite systems were discussed and compared to each other to find out the effect of polymer resin on the mechanical behavior of the composite under dynamic impact loading. At last, the sensitive studies regarding to several internal parameters were carried out in order to better understand and control of the multiscale model.

ACKNOWLEDGEMENTS

I would like to thank my advisor Dr. Wieslaw Binienda for his continuous support, valuable guidance, and encourage throughout my Ph.D study. I am sincerely grateful to him for giving me this opportunity to pursue the research. I am also grateful to Dr. Robert Goldberg for his insightful comments, resource and support to the research throughout these years.

I would also like to thank all my dissertation committee, Dr. Craig Menzemer and Dr. Qindan Huang from Department of Civil Engineering Dr. Xiaosheng Gao from Department of Mechanical Engineering, Dr. Kevin Kreider from Department of Applied Mathematics for their constructive comments and support which widen my research from various perspectives.

I would like to thank Dr. Gary Roberts of NASA Glenn Research Center for his ideas and test results to conduct this research. Acknowledge are also extend to Duane Revilock of NASA Glenn Ballistic Impact Lab for assistance with the ballistic impact test data. I am also grateful to Sheila Person for her opinion and comments on the dissertation.

My sincere thanks are also goes to Dr. Xuetao Li, Dr. Chao Zhang, and Dr. Zifeng Nie for their help and discussion on my research. Special thanks are given to my colleagues, Brett Bell, Menglong Ding, Yanqi Hu, and Ayse Ozen for their discussion during my graduate study.

Last but not the least, I would like to give my gratitude to my family, my parents, parents in law, especially my husband Xuetao Li and my sons Alexander and Benjamin for their consistent love, support, and encourage during this journey.

Table of Contents

	Page
LIST OF TABLES	x
LIST OF FIGURES	xi
CHAPTER	
I. INTRODUCTION	1
1.1 Background	1
1.2 Introduction	3
II. LITERATURE REVIEW	7
2.1 Composite damage and failure	8
2.1.1 Composite damage and failure mechanisms	8
2.1.2 Failure criteria	10
2.1.3 Progressive damage study	17
2.1.4 Delamination simulation	20
2.2 Experimental studies on triaxial braided composites	26
2.2.1 Experiments conducted under static loading	26
2.2.2 Experiments conducted under dynamic loading	29
2.3 Numerical modeling approaches for composite structures	33
2.3.1 Micro-scale modeling	34
2.3.2 Meso-scale modeling	36
2.3.3 Macro-scale modeling	41
2.3.4 Multi-scale modeling	44

2.4 Current research	50
III. MULTI-SCALE MODELING METHODOLOGY OF TRIAXIAL BRAIDED	
COMPOSITE MATERIALS	53
3.1 Composite material system under study	54
3.2 Proposed mesomechanical model	57
3.2.1 Mesh for mesomechanical model	57
3.2.2 Mechanical model of the fiber tow	60
3.2.3 Mechanical model for the matrix	68
3.2.4 Mechanical model for the fiber/matrix interface	69
3.3 Macromechanical Model	76
3.4 Connection between meso-mechanical and macro-mechanical models	78
3.4.1 Constraints at the interface of the meso and macro region	78
3.4.2 Validation of interface constraint.....	79
3.5 Conclusions.....	81
IV. DAMAGE AND FAILURE STUDY OF MATERIAL SYSTEM T700s/E862..	
4.1 Tensile test of a six-layer T700s/E862 straight-sided coupon bar	84
4.2 Free edge effect of a single-layer T700s/E862 straight-sided coupon bar	95
4.3 Ballistic impact of a flat panel composed of a T700s/E862 system	102
4.3.1 Experiment setup and numerical model.....	102
4.3.2 Impact analysis results	106
V. DAMAGE AND FAILURE STUDY OF MATERIAL SYSTEM T700s/PR520120	
5.1 Experimental measurements of different polymer and composite materials..	121
5.2 Characterization of meso-mechanical model for the T700s/PR520	124

5.3 Static coupon testing of the T700s/PR520 material system	125
5.3.1 Straight-sided coupon axial tensile test.....	126
5.3.2 Straight-sided coupon transverse tensile test	128
5.3.3 Summary of static coupon tests	134
5.4 Simulation of impact on a flat panel of T700s/PR520 material	135
5.4.1 Impact results for the T700s/PR520 circular panel.....	137
5.4.2 Prediction results for a T700s/PR520 circular panel	139
5.4.2 Summary	146
VI. MODEL PARAMETER STUDY AND NUMERICAL INVESTIGATION	148
6.1 Resin strain rate effects	148
6.2 Fiber tensile failure strain	152
6.3 Effect of bonding interface	156
6.4 Summary	160
VII. SUMMARY AND FUTURE WORK.....	161
7.1 Summary	161
7.2 Future work.....	165
REFERENCES	167

LIST OF TABLES

Table	Page
3.1 Material mechanical properties for constituents	56
3.2 Effective elastic constants of unidirectional fiber tows	64
3.3 Failure strength of unidirectional fiber tows.....	64
3.4 Fracture energy for composites in this study	76
5.1 Polymer resin mechanical properties (data obtained from Littell (2010)).....	122

LIST OF FIGURES

Figure	Page
2.1 Illustration of different failure modes for a composite at microscale	9
2.2 Demonstration of maximum stress failure criteria.....	13
2.3 Multiscale scheme and connection between different scales of modeling	45
3.1 Braiding structure of (+60/0/-60).....	55
3.2 Micro image showing a typical composite cross section with a matrix and fiber tows	58
3.3 Idealized architecture of a fiber tow	58
3.4 3D discretization of a unit cell of a triaxial braided composite	59
3.5 Cross section of a representative volume element.....	62
3.6 Diagram showing parameters for MAT_221	67
3.7 Stress-strain relation for pure E862 resin.....	68
3.8 Three types of fracture modes for a material	70
3.9 Traction-separation relation	70
3.10 Pure mode bilinear traction-separation curve	75
3.11 Braided through the thickness approach and subcells	77
3.12 Multi-scale hybrid element approach scheme.....	78
3.13 Connection between regions of the meso-scale model and the macro-scale model	79

3.14 Transverse tensile coupon bar.....	80
3.15 Displacement distribution field of the transverse tensile bar.....	80
3.16 Resultant forces at cross sections.....	81
4.1 Specimen configuration for axial and transverse tension test.....	84
4.2 Simulated axial tensile specimen with MHEA approach	85
4.3 Simulated transverse tensile specimen with MHEA approach	85
4.4 Correlation of stress and strain in the axial tensile test.....	87
4.5 Correlation of stress and strain in the transverse tensile test	87
4.6 Predicted failure pattern of coupon bar tensile test by numerical model:.....	88
4.7 Matrix mode damage in the axial tensile test predicted by numerical model	90
4.8 Delamination damage in the axial tensile test predicted by numerical model.....	90
4.9 Matrix mode damage in the transverse tensile test predicted by the numerical model	91
4.10 Delaminated nodes in transverse tensile test predicted by the numerical model.....	91
4.11 Fiber tow splitting in the transverse tensile test.....	94
4.12 DIC of out-of-plane displacement contour at global strain of 1%	96
4.13 Numerical results of out-of-plane displacement at a global strain of 1%	98
4.14 Numerical prediction of failure pattern in single layer transverse tensile test.....	99
4.15 Free edge effect in axial tensile test.....	101
4.16 Panel impact model using the MHEA approach.....	105

4.17 Velocity history for projectiles with three different initial velocities	106
4.18 Penetrated composite panel.....	107
4.19 Simulation predicted damage pattern in a composite panel after impact with initial velocity of 165m/s.....	107
4.20 Simulation predicted damage pattern in a composite panel after impact with initial velocity of 100m/s.....	108
4.21 Axial fiber tows damage evolution at impact velocity of 100m/s	111
4.22 -60° fiber tows damage evolution at impact velocity of 100m/s	112
4.23 +60° fiber tows damage evolution at impact velocity of 100m/s	113
4.24 Axial fiber tows damage evolution at impact velocity of 157m/s	114
4.25 -60° fiber tows damage evolution at impact velocity of 157m/s	115
4.26 +60° fiber tows damage evolution at impact velocity of 157m/s	116
4.27 Delamination propagation of a composite panel using a projectile with an impact velocity of 165m/s.....	118
5.1 Measured axial stiffness and strength of triaxial braided composites systems with four different polymer resins	123
5.2 Measured transverse stiffness and strength of triaxial braided composites systems with four different polymer resins	123
5.3 Stress–strain relation for the axial tensile test.....	126
5.4 Matrix mode failure at (a) 1.5% strain and (b) 2.0% strain	127

5.5 Stress–strain relation for the transverse tensile test	129
5.6 Matrix failure progression: (a) 0.5% strain (b) 1.0% strain and (c) 1.5% strain	131
5.7 Fiber tensile failures at 1.5% strain.....	132
5.8 Delamination progression (a) 0.5% strain (b) 1.0% strain and (c) 1.5% strain	133
5.9 A T700s/PR520 flat panel in a circular fixture with DIC paint is applied to the back side of the specimen.....	136
5.10 Multi-scale model of a flat panel of T700s/PR520 material.....	136
5.11 Damaged panel under an impact velocity of 165.3 m/s (542.1 ft/s), where the projectile is contained with extensive damage to the front side and back side.....	138
5.12 Damaged panel under an impact velocity of 166.6 m/s (546.5 ft/s), where the projectile penetrates the panel, creates a through-hole, and has a residual velocity of 30.4 m/s (99.8 ft/s).	138
5.13 Comparison of experimental results with the numerical prediction for the residual velocity of a projectile striking a T700s/PR520 circular panel.....	139
5.14 Predicted projectile velocity time history for impact velocities of 165 m/s (541.3 ft/s), 175 m/s (574.1 ft/s), and 195 m/s (639.6 ft/s) when striking a T700s/PR520 circular panel.....	140
5.15 Panel damage at impact velocity of 165 m/s (541.3 ft/s).....	142
5.16 Prediction of panel damage under an impact velocity of 175 m/s (574.1 ft/s)	143
5.17 Prediction of panel damage under an impact velocity of 195 m/s (639.6 ft/s)	144

5.18 Predicted damage progression in a panel under an impact velocity of 175 m/s (574.1 ft/s)	146
6.1 Stress–strain curves for the resin of E862 obtained from experiments conducted at various loading rates	149
6.2 Assumed material behavior in finite element analysis at different strain rates	150
6.3 Finite element results for projectile velocity of 165 m/s with composites having different pure resin strain rate properties	151
6.4 Progressive damage model of a fiber tow in a stress–strain relationship	152
6.5 Sensitivity of the failure strain of a fiber tow (2.00%, 2.11%, 2.15% and .50%) with respect to the projectile velocity based on a test using an impact velocity of 165 m/s ..	153
6.6 Cross-sectional cut of damaged panels with different failure strains	155
6.7 Sensitivity of interface strength (50%, 100% and 150% of baseline values) on penetration velocity based on tests using an impact velocity of 165 m/s	157
6.8 Delamination and damage progression in the meso-mechanical region for materials with different interface strengths	159

CHAPTER I

INTRODUCTION

1.1 Background

Composite materials have been widely used in the aerospace industry, automobile manufacturing, and other commercial applications, due to their high strength, light weight and low cost. Compared to other engineering materials, composite materials have unique mechanical behavior characteristics; among other things, they are heterogeneous (the material properties are position related in the component) and orthotropic/anisotropic (the material properties depend on the direction or orientation at a point). The inherently heterogeneous nature of composite materials allows engineers to design a specific composite material by adjusting the proportion and arrangement of the constituents to achieve a certain specified strength and stiffness. A composite material can also be tailored to have unequal strength and stiffness in different material directions according to design requirements, which is a significant advantage over an isotropic material, which has excess strength and stiffness in any direction other than that of the largest required strength or stiffness.

The two-dimensional triaxial braided composite (2DTBC), one of the widely used types of composite materials, is partly composed of bundles of continuous filaments called *fiber tows*. Bias fiber tows undulate over and under axial fiber tows at a certain angle, and

the particular braiding architecture gives a composite material the flexibility to conform to the shape of a mold. The quasi-isotropic structure, which exhibits equal material stiffness properties in all directions, can result in efficient energy distribution during an impact event. Experiments have demonstrated that 2DTBCs can resist crack initiation and propagation as well as the induction of large delaminations during dynamic impact loading.

2DTBC has been used to manufacture jet engine fan cases and fan blades in the aerospace industry. During an airplane's service period, the fan case has to be able to contain a blade and fragments during an engine fan blade out event. Therefore, as part of the design process, a jet engine fan blade and a fan case made from composite materials must meet impact design requirements. Multiscale ballistic impact experiments have been conducted to evaluate composite material properties using small-scale composite panel specimens with a soft projectile, and to evaluate damage behavior in engine fan cases using a full-scale, full-size fan case with fan blades as projectiles. However, conducting experimental tests of 2DTBC under dynamic impact is time consuming and expensive. Numerical simulation provides an efficient way to investigate composite material response under ballistic impact loading. In addition, numerical models can provide insight regarding the time-dependent internal damage propagation, which cannot be obtained from experiments.

LS-DYNA is a general-purpose finite element program originally created for the modeling of impact events. It can perform highly nonlinear, transient dynamic finite

element analyses using explicit time integration. It is known for its reliable explicit time integration algorithm and robust contact algorithm; thus, it has been widely used by the aerospace industry to simulate bird strikes and jet engine blade-out events.

1.2 Introduction

From a structural design point of view, the penetration velocity and the deflection of a composite material are critical factors in the design of jet engine fan cases. Studying the response of a composite material to impacts of various velocities and determining a penetration velocity given a known composite material system is useful when designing composite materials for fan cases to contain fan blades. In addition, there is a critical deflection requirement for components of such a structure to avoid extremely high deformations, which might affect the normal performance of other components; thus, it is necessary to gain a better understanding of the deflection response of a composite material due to impact events. From a material design point of view, knowledge of damage initiation and propagation mechanisms under dynamic impact loading is important for improving the performance of a composite material with desirable properties.

The goal of the work in this study is to develop a sophisticated multiscale model to investigate the penetration velocity and panel deflection behavior of a composite structure under various impact velocities as well as to study damage mechanisms under ballistic impact loading conditions. Two representative composite material systems are investigated in this work, T700s/E862 and T700s/PR520, which were selected based on the fact that

E862 is a resin that is neither too brittle nor too ductile, while PR520 resin is a toughened resin. Toray's T700s carbon fiber is a commonly used standard modulus fiber that is designed for aerospace applications.

Numerical models can be generated at three levels: micromechanics, mesomechanics and macromechanics. For a 2DTBC, a micro-scale model defines the proportion and arrangement of fibers inside the composite fiber tows. By applying a homogenization methodology, effective material properties of the fiber tow can be determined in terms of the constitutive properties. A meso-scale model defines the complicated fiber tow braiding architecture and provides maximum information on the braiding geometry, which allows the identification of the damage initiation and propagation for each constituent. At the macro-level, the composite material is assumed to be homogenous, and the effect of the constituent materials are obtained as averaged (smeared) macro-level material properties.

Meso-scale models can provide information on the effects of the complex composite architecture on the material response as well as details of damage evolution in each constituent with accurately defined internal architectures. Therefore, they are ideal candidates for investigating damage and failure mechanisms of 2DTBC during the process of ballistic impact. However, given the complicated geometry and mesh for a meso-scale model, if the entire structure is meshed using a meso-scale model, the computational time increases dramatically. In order to balance the simulation accuracy and computational

efficiency, a multiscale simulation technique is proposed that replaces the meso-scale model at regions far from the area of interest with a macro-scale model. Using a geometry and mesh that are less complicated, macro-scale models can help to improve the computational efficiency.

In this study, a meso-scale model is first developed, a process that includes geometry/mesh generation and the determination of material properties. The geometry consists of two parts: a realistic geometry of the braiding fiber tows and the geometry of the pure resin region. The material properties of the fiber tows can be predicted using a micromechanical method by modeling them as unidirectional composites. It is found experimentally that delamination is one of the primary failure modes for a composite during impact. Therefore, in addition to the geometry and material properties, the interaction between adjacent layers and between the fiber tow and the matrix must also be considered. For the region away from the area of interest, a macro-scale model using a simplified geometry can be applied in which the braiding architecture of the fiber tows is only accounted for in an approximate manner. Material properties for the macro-scale model can be computed through the use of classical laminate theory.

Prior to impact test modeling, meso-scale model and hybrid modeling schemes were examined. The model results were correlated by simulating static tensile tests on six-ply-thick, straight-sided coupons. By comparing the predicted global stress–strain relation with the experimental results, the material properties used in the numerical model were

correlated. Damage initiation and propagation were illustrated and the failure mechanisms of the composite coupons under tension loading were identified using the meso-scale model.

Simulations using the developed numerical model were carried out to study the damage and penetration process in a circular composite panel which was impacted by a hollow aluminum hard projectile under the same conditions as corresponding experiments which were conducted at NASA Glenn Research Center in Cleveland, Ohio. The penetration threshold was found for the specific composite material system after conducting many simulations with various impact velocities, and the results were compared to the data obtained in the corresponding experiments for both material systems. By employing a meso-scale numerical model, the time-dependent damage evolution process of each constituent was investigated, and the internal damage behavior along the thickness direction of the composite was illustrated. Moreover, damage propagation behavior and failure mechanisms for the two different material systems under the same impact conditions were analyzed, and the results were compared to each other.

Sensitivity studies of several internal parameters of the finite element model was carried out to have better understanding and control of the numerical model and its effect on the damage and failure mechanisms.

CHAPTER II

LITERATURE REVIEW

Two-dimensional triaxial braided composites (2DTBCs) have been widely used in aerospace applications such as jet engine fan cases, fan blades and other components. It is composed of braiding fiber tows infused with resin through resin transfer molding (RTM) process. Axial fiber tows lay straight with equal space while two bias fiber tows undulate over and under axial fiber tows at an angle of 60° . The 2DTBC used in this work is fabricated with 24K filament in axial fiber tows and 12K filament in bias fiber tows in order to balance stiffness in different directions. Each fiber tow can be seen as a unidirectional composite with 1-direction defined along fiber direction, such that material properties of the fiber tow can be obtained using the method applied for unidirectional composite material.

Braided composites provide high stiffness and strength to aerospace components, and they can provide improved through-thickness behavior and impact strength as compared to laminated composites. In addition, triaxial braided composite materials are lightweight, and the associated reduction in fuel consumption results in cost benefits. In addition, these materials allow high flow during the resin transfer molding (RTM) process, and they can conform to the RTM mold very well, a feature that is not typically possible with woven textiles (Kohlman, 2012). The particular braiding architecture for the 2DTBC

results in a quasi-isotropic laminate in terms of the overall material response properties, and the global in-plane stiffness is expected to be the same in all directions, which results in efficient energy distribution during an impact event. Furthermore, this type of braiding architecture also can simplify the design and manufacturing process (Roberts, 2009).

It is well known that compared to conventional metallic composite materials, the material and structure of a TBC results in superior capabilities for energy absorption, and the damage and failure mechanisms are different as well. For metallic materials, damage and failure starts with grain dislocation and, at the macro level, it is exhibited as plastic yielding, while a composite material absorbs energy through matrix cracking and fiber failure at various levels. This section presents a review of composite failure and damage mechanics, modeling approaches at different scales, and test observations regarding the behavior of triaxial braided composite materials.

2.1 Composite damage and failure

2.1.1 Composite damage and failure mechanisms

A composite is a heterogeneous laminated material that typically exhibits local damage at the micro-level prior to rupturing into pieces at the macro-level. The initial damage does not necessarily correspond to the final failure. The failure mechanisms of composite materials are composed of a series of cracks or fractures at different scales.

Figure 2.1 Illustration of different failure modes for a composite at microscale

A variety of mechanisms at the fiber/matrix micro-level can cause fibrous composite materials to fail. These include fiber fracture, fiber buckling (kinking), fiber splitting, fiber pullout, fiber/matrix debonding, matrix cracking, and radial cracks. Fiber fracture occurs under tensile loading when the maximum allowable axial tensile stress (or strain) of the fiber is exceeded. Fiber pullout, which occurs when the fiber fractures, is accompanied by debonding at the fiber/matrix interface. Matrix cracking develops when the strength of the matrix is exceeded. Fiber kinking arises when the axial compressive stress causes fibers to buckle. Fiber splitting and radial interface cracks occur when the transverse or hoop stresses in the fiber or interphase region between the fiber and matrix reach their ultimate value (Herakovich, 1998).

At the laminate level, lamina failures result from the accumulation of micro-level mechanisms. These failures may appear in the form of transverse cracks in planes parallel

to the fibers, fiber-dominated failures in planes perpendicular to the fibers, and/or delamination between layers of the laminate (Herakovich, 1998).

2.1.2 Failure criteria

The prediction of failure due to a specified state of stress is one of the most important issues to consider in the analysis of a composite material. Although the failure starts with a micro-level damage event, micromechanics is rarely used to resolve this problem because of the complexity of the micro-level failure process. In the micromechanics method of predicting composite failure, successive micro-level failures are predicted by means of a study of micro-stress and micro-level failure criteria. Next, micro-level failures derived from the actual physical failure process must be integrated to form a theory of macro-level failures. As the micro-scale failure mechanism is complex, this method and process is not feasible to apply from the standpoint of composite structural design and analysis.

Therefore, in real engineering problems, a macro-level failure criterion is of interest to account for composite failure at the meso- or macro-level of the lamina, the laminate, and the structural component. The structural designer can use such failure criteria to evaluate the safety of designed components under the most general structural configuration and loading conditions (Herakovich, 1998). Such failure criteria can also be adopted in finite element analysis code to account for the onset of the damage process due to their simplicity and ease of use. For example, frequently used failure criteria such as maximum

stress, maximum strain, and Hashin failure criteria can be found in finite element codes of Abaqus (Dassault Systèmes) and LS-DYNA (Livermore Software Technology Corporation, 2007).

Over 100 different macroscopic failure theories have been proposed for fibrous reinforced plastic composites over the last few decades. A recent survey of studies evaluating and characterizing the differences between the various theories can be found in Paris (2001) and Hinton (2004). The existing failure theories can be divided into two broad categories (Camanho, 2002). Studies in the first category of failure criteria provide mathematical expressions to describe the failure surface through fitting of the available experimental data and are not associated with physical meaning. Studies in the second category have a physical basis and are associated with certain composite material failure modes.

One of the first failure criteria introduced for composite materials is the Tsai–Hill failure criteria (Tsai, 1965) which adopted the same mathematical polynomial as the yield criterion of an orthotropic ideally plastic material proposed originally by Hill (1948). The Tsai–Hill failure criterion is expressed in quadratic form, with stresses having coefficients determined in terms of failure strength in the principle material coordinates of a lamina. This criterion belongs to a category based on mathematical curve fitting without any consideration of the physical meaning of the curve, and it turns out to be quite suitable for predicting failure of E-glass/epoxy resin composites.

In order to account for unequal failure stresses in tension and compression, Hoffman (1967) added linear terms to the Tsai–Hill criterion. As expected, for equal strength in tension and compression, the Hoffman criterion is reduced back to the Tsai–Hill criterion. The Hoffman criterion is found to be in very good agreement with failure data obtained for glass-epoxy, graphite-epoxy, and boron-epoxy via experiments.

As one example of curve fitting failure criteria, the Tsai–Wu tensor failure criterion has the most general form for nonphysical-based polynomial failure criteria for composite materials. This criterion (Tsai and Wu, 1971) is definitely an improvement over previous criteria due to its generality and versatility, as it has a larger number of terms in the equation and represents the various strengths in tensor form. The Tsai–Wu criterion is restricted to quadratic order approximation, because it is the simplest representation that can fit the data reasonably well and, based on the observed experimental data, it is not worthwhile to employ higher-order approximation.

In several other well-known quadratic failure criteria — including the Tsai–Hill failure criterion (Tsai, 1965), the Azzi–Tsai failure criterion (Azzi, 1965), the Hoffman failure criterion (Hoffman, 1967) and the Chamis failure criterion (Chamis, 1969) — weight parameters in the Tsai–Wu failure criteria are varied to ensure a better fit of the failure surface to the experimental data (Camanho, 2002). Although significant achievements have been made in developing curve-fitting-based failure criteria, we lost the ability to

determine the precise failure mode from which the material fails. The curves merely connect unrelated data points obtained from the experiments.

To consider a damage mechanism with a physical meaning, another category of failure criteria associated with failure modes has been proposed. The most widely used in design are maximum stress and maximum strain. The maximum stress failure criterion assumes that failure occurs whenever any component of the stress exceeds its allowable value, independent of the values for all other components of stress. The allowable values are determined from a series of tests on unidirectional composite specimens. The same approach is used to acquire the maximum strain failure criterion. Since obtaining the maximum stress and maximum strain failure criteria is a simple and straightforward way to predict failure of a composite, these failure criteria are adopted most often in industry. However, these criteria neglect stress interaction effects, which can lead to insufficient accuracy in the failure prediction.

Figure 2.2 Demonstration of maximum stress failure criteria

Generally, materials undergo multiaxial stress that varies at different locations of a composite component. Failure initiation results from the contributions of more than one stress component; thus, it is desirable to consider the interaction of certain components and their association with observed failure mechanisms. From observations in experiments where loading was applied at various angles with respect to fiber direction for a unidirectional reinforced composite, Hashin and Rotem (1973) noted that specimens fail in two different modes depending on the angle θ , which is the angle between the fiber direction and the loading direction. When θ is 0° or is very small, the specimen fails by cumulative fiber failure; for larger angles of θ , the specimen fails due to formation of a crack through the matrix in a direction parallel to the fiber. Considering the non-homogeneous character of a composite, Hashin and Rotem identified two independent failure modes, each with a specific failure criterion for composite materials: a fiber failure mode when θ is very small, and a matrix failure mode when θ is relatively large. A simple failure mode-based, piecewise smooth curve failure criterion containing two independent mathematical expressions was developed for a plane stress situation in terms of the strength of the pure fiber in two directions: pure transverse and pure shear. Considering the different strengths for the fiber tension/compression mode and the matrix tension/compression mode, the criterion was expanded to a total of four mathematical expressions that correspond to the four failure modes (fiber tension, fiber compression, matrix tension, and matrix compression).

Hashin (1980) later extended this concept for a fiber-matrix failure mode to a three-dimensional (3D) criterion based on the composite's original plane stress condition. Similar to the previous failure criterion, a total of four failure modes are developed: tensile fiber mode, compressive fiber mode, tensile matrix mode, and compressive matrix mode. Similarly, a quadratic form for stress was adopted based on the curve fitting requirement. The derivative of the failure criterion originates from the consideration of invariant characterization under any rotation of the x_2 and x_3 axes around x_1 because of the transverse isotropic nature of a unidirectional composite. By applying the known axial and shear strengths and by observing failure mechanisms, the unknowns in the polynomial can then be determined.

Other similar failure criteria are defined in the same fashion while emphasizing the differences in certain failure models, e.g. the Chang–Chang criterion (Chang and Chang, 1987), the Christensen failure criterion (Christensen, 1997), and the Puck criterion (Puck and Schurmann, 1998).

Identification of the failure mode in the failure criteria is an important improvement if we wish to apply a progressive damage algorithm using the finite element method (FEM). During a progressive damage process, the composite failure mode must first be identified and the corresponding material stiffness will degrade with respect to the failure mode. As an example, this feature can be well illustrated by MAT_161 and MAT_162, two newly developed material models in LS-DYNA. In these material models, the damage initiation

is captured using a modified 3D Hashin failure criterion to include the effect of through thickness direction on composite failure. The models can be used to effectively simulate fiber failure, matrix damage, and delamination behavior (Xiao et al., 2007). Three quadratic damage functions are used for fiber-dominated failure: one in tension/shear, one in compression, and one in crush under pressure. Similarly, three additional damage functions are provided for matrix-dominated failure: one in transverse compression, one in perpendicular tension, and another for delamination. Once the stress state at a given point meets the failure criterion, the progressive damage analysis process starts to realize degradation of the mechanical properties of the material. A set of damage variables are introduced to relate the onset and growth of damage to the loss of stiffness in the material. Each stiffness component in the compliance matrix is then reduced in the value in such a way that after the failure mode is identified, the corresponding stiffness component in the compliance matrix is multiplied by a variable for the growth rate of damage. The material model uses a coupling matrix to realize the degradation of stiffness according to corresponding failure mode.

The abovementioned failure criteria are focused at the level of the unidirectional lamina, but they fall short in their ability to predict the failure of textile composites, due to the high anisotropy of a single ply. Only textile composites with minimal fiber undulations and low density in 3D reinforcement can be described by failure criteria developed at the lamina level. In some studies, researchers have tried to appropriate failure criteria to

address a specific type of textile composite. Juhasz et al. (2001) developed a failure criterion for 3D orthogonal fiber-reinforced non-crimp fabrics. Xiao et al. (2007) generalized the 3D Hashin failure criteria of a unidirectional lamina to characterize the damage for a plain weave layer, where the fill and warp fiber interaction are considered. Similar to the lamina level, six failure modes were considered, including the delamination mode, and as many as 15 strength parameters were used. Xiao's constitutive model also considers the effects of the strain rate.

2.1.3 Progressive damage study

In the numerical modeling of anisotropic composite materials, failure theory has been used to determine failure initiation for each failure, which is defined as the onset of damage for a specific failure mode in the composite. For some material models, once failure occurs, post-failure behavior is assumed to act in an ideally brittle manner, where the stiffness and stress are reduced to zero or where stiffness is reduced by applying a scaling factor. The experimental evidence has shown that failure in composites is often progressive, in a process where the damage accumulates. Damage mechanics is widely utilized to deal with the progressive degradation in material stiffness as a function of the type of failure mode at the lamina level, as described in the previous section (Fang and Liang, 2011).

In a continuum damage approach, a damage variable in either scalar or tensor form is introduced to represent the level of damage in the composite material, and it can range

from no damage to complete damage. With a damage variable, the kinematic constitutive equations for the damaged composite can be recalculated, which leads to a strain softening behavior though the degradation of the original material modulus.

One of the first attempts to incorporate damage mechanics into composite modeling was made by Talreja (1985). In this work, to represent the major material orientations in two directions (parallel to the fiber and transverse to the fiber), two damage variables were proposed. The model was used to predict the stiffness reduction in angle ply laminates, and it shows good agreement with experimentally measured results. Another early application was reported by Frantziskonis (1988), who proposed a scalar damage model, where the damage threshold and growth are polynomials of the strain. Tests were used to characterize this model, and predictions based on the proposed model showed good agreement with the laminate tests. Engblom and Yang (1995) later developed a straightforward model based on damage mechanics to predict the effect of intralaminar damage.

To describe the elastic–brittle behavior of fiber reinforced composites, Matzenmiller et al. (1995) developed a rigorous anisotropic damage constitutive model. The developed model assumes two-dimensional (2D) plane stress conditions and the Hashin failure criterion. Four damage variables, which correspond to four failure modes (Fiber rupture, Fiber buckling and kinking, Matrix cracking under transverse tension and shearing, and Matrix crushing under transverse compression and shearing), were introduced in this work. Later efforts were also made to create a damage model for a

composite laminate, such as the work of Williams and Vaziri (2001) and Maimi et al. (2007). In order to account for scales of different lengths, Talreja (2006) developed formulas using damage mechanics method to account for stiffness at different scales.

Numerically, a damage variable evolves with corresponding increments in strain, making it well suited for implementation in standard finite element codes, such as MAT58, MAT59 and MAT161 in LS-DYNA (Livermore Software Technology Corporation, 2007), as well as the anisotropic damage model for a fiber-reinforced composite in Abaqus (Dassault Systèmes, 2005). In addition, MAT162 was developed in LS-DYNA using continuum damage mechanics of anisotropic materials with a simple damage evolution law. These types of material model may be used to model the progressive failure in composite materials that consist of unidirectional and woven fabric layers. The failure criteria have been established by adopting modified 3D Hashin failure criteria with a generalization that includes the effect of through thickness direction on the failure of the composite. It can be used to effectively simulate fiber failure, matrix damage, and delamination behavior (Xiao et al, 2007). Three quadratic damage functions are used for fiber-dominated failure: one in tension/shear, one in compression, and another in crush under pressure. Similarly, three additional damage functions are provided for matrix-dominated failure: one for transverse compression, one for perpendicular tension, and another for delamination. In the MAT162 code, the methodology is generalized beyond the scope of unidirectional laminae to capture the behavior of a woven fabric; furthermore, 2D

in-plane Hashin failure criterion is extended to incorporate delamination mechanism that occurs between layers. Delamination is one of the predominant forms of failure in laminated composites due to the lack of reinforcement in the through thickness direction, which is a very important damage mechanism induced by ballistic dynamic impact or manufacturing defects.

Compared to other widely used delamination simulation methods such as those based on fracture mechanics, MAT162's ability to model composite delamination without physical interfaces provides a computationally efficient approach for composite modeling and simulation. A similar methodology is adopted for damage modeling in Abaqus. The effective stress, which is used to evaluate the initiation failure criteria, is computed by multiplying the true stress by a damage operator. The damage operator is a function of damage variables that characterize fiber, matrix, and shear damage corresponding to the Hashin and Rotem (1973) failure modes. Prior to damage initiation, the damage operator M is equal to the identity matrix. In contrast to an explicit finite element solver, an implicit finite element solving algorithm requires special treatment for the Jacobian matrix as the stiffness degrades in order to achieve a better convergence.

2.1.4 Delamination simulation

Delamination is a very common failure mode for laminated composite structures. It exists between layers of plies and is thus called *interlaminar failure* in order to differentiate it from *intralaminar failure* (which is failure within a single layer).

Delamination usually occurs at a very early stage of the damage process and is followed by crack growth within the ply interface. In carbon-reinforced epoxy composites, the characteristics of delamination mainly depend on the properties of the resin.

In engineering analysis, delamination is typically considered separately from other in-plane failure modes. In finite element analysis, one method is to use contact at the interface with strength defined in the normal and shear directions. At the material interface, surfaces belonging to two separate composite layers are defined as a pair of *slave* and *master* contact faces. The members of the pair are initially tied or bonded together to simulate a condition without delamination. Delamination occurs when contact stress at the interface exceeds an allowable value.

As mentioned in the previous section, fracture mechanics is widely used in composite delamination analysis to predict the growth of cracks. The basic concept of using fracture mechanics to predict delamination was first suggested by Griffith (1921), who assumed linear elasticity around the crack tip, in a process known as *linear elastic fracture mechanics* (LEFM). Later, this concept was developed into elasto-plastic fracture mechanics by Irwin (1956), and the path-independent integration of J-integrals was later proposed by Rice (1968). Since then, notable research studies have concentrated on the application of fracture mechanics for modeling delamination.

Currently, three main techniques have been widely adopted to study the crack initiation and propagation process: the virtual crack closure technique (VCCT; Rybicki and

Kanninen 1977), the cohesive zone model (CZM; Needleman 1987), and the extended finite element method (XFEM; Moës et al. 1999). These methods require only one complete analysis of the structure to obtain the local deformations and the total energy release rate at the crack front.

VCCT is a finite element technique developed to calculate the rate of strain energy release during the crack propagation process. The fundamental basis of VCCT is the assumption that the strain energy released by crack growth is the same as the energy required to close the crack to its original position. VCCT is based on Irwin's virtual crack closure method (Irwin, 1957), using the stresses ahead of the crack tip and the displacements behind the crack tip to obtain the strain energy release rate at tip front. Rybicki and Kanninen (1977) later applied this idea in finite element analysis and obtained a simple formula for calculating the strain-energy release rate for a numerical model with four-noded quadrilateral non-singular elements. In this formula, the energy release rates for three modes — opening (Mode I), in-plane shear (Mode II), and out-of-plane shear (Mode III) — are considered by using the tractions at the nodes located at the crack tip and the separation at the nodes located behind the crack tip. Based on this idea, many researchers further derived formulas for higher-order and singularity elements (Raju, 1987). The VCCT technique has since been adopted in finite element analysis software, e.g. Abaqus, to calculate the strain energy release rate and to predict the delamination given a crack of a predefined size and location.

In order to apply VCCT in a numerical framework, it is necessary to first define two distinct contact surfaces that, in the initial state, are partially bonded and between which the crack will propagate. The initial crack and the location of crack tips need to be identified explicitly, and this can be accomplished by defining a node set that identifies the initially bonded part in order to model the separation process of the bonded surface. The crack tip nodes debond when the energy release rate (as calculated by the VCCT assigned to the surface pair) is equal to or exceeds the critical energy release rate (Dassault Systèmes, 2018). The drawback of using VCCT is the requirement of assuming an existing crack in the material, which means it cannot model crack initiation in a pristine material. Furthermore, it requires the use of a complex remeshing technique to advance the crack tip during the simulation of delamination.

Traction-separation-based CZM is another common fracture mechanics-based method that is used to study composite delamination and that can be found in the finite element analysis software packages Abaqus and LS-DYNA. This method is based on the Dugdale–Barenblatt cohesive zone approach (Dugdale, 1960; Barenblatt, 1962), which can be related to Griffith’s fracture mechanics theory, where the area under the traction–separation relation is equal to the corresponding fracture toughness regardless of its shape. In this cohesive technique — which has been widely developed in finite element analysis in many research studies, such as Cui and Wisnom (1993), de Moura et al. (1997), and Camanho and Davila (2002) — zero-thickness cohesive elements are utilized to capture

delamination onset and growth. The three basic fracture failure modes (opening Mode I, in-plane shear Mode II, and out-of-plane shear Mode III) are considered. However, in reality, delamination typically occurs in a damage mode known as *mixed mode delamination*, which is a combination of the three pure modes. In CZM, delamination under pure mode and mixed mode follow similar algorithms:

- In *pure mode delamination*, before the onset of damage, a linear elastic procedure with high stiffness that holds the top and bottom faces of the cohesive element is generally assumed. Several failure criteria can be used to predict the initiation of delamination, including the maximum nominal stress criterion, the maximum nominal strain criterion, the quadratic nominal stress criterion, and the quadratic nominal strain criterion. Once an initiation criterion is satisfied, delamination begins, and the stiffness is gradually reduced to zero in linear, exponential or tabular behavior. The total decohesive rupture is predicted by comparing the strain energy release rate to the fracture toughness under pure mode loading.
- In *mixed mode delamination*, a delamination initiation criterion is used to predict the onset of the softening process, which is typically assumed to be a quadratic function that includes all three traction components. An additional delamination propagation criterion is applied to predict the total decohesive of the cohesive element. Considering the effect of the mode ratio on the fracture

toughness (Reeder, 1992; Benzeggagh, and Kenane, 1996), a criterion that is able to capture the mixed mode fracture toughness under different mode ratios should be used to predict the propagation of delamination (Camnho, 2002). Unlike the VCCT method, the CZM method does not require the presence of a preexisting crack. In fact, the precise locations where cracks initiate, as well as the evolution characteristics of such cracks, are determined as part of the solution.

For both VCCT and CZM, a delamination propagation path must first be assumed, and this path might not be the true delamination path in the actual delamination process. In addition, to apply either of these two methods, a predefined crack path must coincide with the edges of the finite elements.

A more recent finite element technique known as XFEM, which is now widely adopted in finite element software to model the crack growth, extends the classical finite element method by enriching the solution space for solutions to differential equations with discontinuous functions (Pathak et al., 2013; Fang and Liang, 2011). XFEM was first proposed by Belytschko and Black (1999) to realize the independence of meshes on the physical surfaces of a structure, can be used to simulate the cracks and interfaces of different constituents without having to predefine the crack path and crack location. Later, Moës et al. (1999) extended it so that it would be applicable to long cracks and to three dimensions. In XFEM, local enrichment functions and additional degrees of freedom are

added to the standard displacement approximation in order to enrich the standard displacement approximation in the region of interest. In addition, a partition of unity technique is applied in order to incorporate the enrichment function into the finite element approximation. As a result, these discontinuities are not considered as part of the finite element mesh.

2.2 Experimental studies on triaxial braided composites

Numerous experiments have been carried out to investigate the damage initiation and propagation mechanisms of triaxial braided composites under both static loading and impact (dynamic) loading conditions.

2.2.1 Experiments conducted under static loading

A mechanical study of constituent materials of a composite system was conducted by Littell (2008), who investigated triaxial braided composite material systems composed of T700S fiber (Toray Composite Materials America, Frederickson, Washington) with four different resins (Cytec CYCOM® PR520 resin, Cytec 5208 resin, Epon 862 resin, and Hexcel 3502 resin) and obtained global stress–strain curves and a better understanding of progressive composite failure under quasi-static loading. Littell found that the value of axial failure strain in the T700S fiber/PR520 resin material system corresponds to the ultimate failure strain of the T700S carbon fibers reported by the manufacturer (Toray Composite Materials America, Frederickson, Washington), which indicates that the fibers dominated the composite axial failure strain with little or no influence from the resin in this

material system. However, for the other three material systems, the axial failure strain of the composite was less than the failure strain of the T700S fiber. In addition, the T700S fiber/PR520 resin material system showed linear elastic and brittle composite behavior until fiber failure, while the other material systems exhibited a non-linear response starting at a much smaller strain than the failure strain. The damage (matrix cracking, fiber breaking, and layer delamination) that accumulated in the three weaker material systems (T700S fiber/E862 resin, T700S fiber/5208 resin, and T700S fiber/H3502 resin) could account for the nonlinear behavior in the axial material response curves. From the results of this study, it was concluded that the T700S fiber/PR520 resin material system had the highest strength in both the axial and transverse directions and thus has higher damage resistance and stronger fiber resin interface; the next highest strengths were observed in the T700 fiber/E862 resin material system.

For a transverse composite coupon specimen, the axial fiber tows are oriented perpendicular to the direction of loading — unlike the axial tensile test, where continuous axial fibers extended throughout the entire length and along the loading direction. Because there were no continuous axial or bias fibers extending throughout the entire gage section, the primary load-carrying mechanisms in the composite were the matrix and the fiber–matrix interface. The strength differences of different material systems are attributed to the differences between the strength of the matrix and that of the fiber–matrix interface.

Littell et al. (2008) then carried out an extensive experimental characterization of a triaxial braided composite using straight-sided and H-shaped specimens. Four braided materials, consisting of the carbon fiber system impregnated with four different types of polymer epoxy, were tested under uniaxial tension and compression as well as shear loading conditions. Optical strain measurement techniques were applied in the testing. It was reported that different epoxies induce different fiber/matrix bonding conditions, and differences in global behavior can be observed. Some of the experimental data obtained by Littell is incorporated into the data set used in this work.

Kohlman (2012) conducted quasi static tests on single-ply straight-sided triaxial braided composite coupon bars in order to focus more directly on the deformation within an unconstrained unit cell of the braid, where the T700S/E862 material system was used to construct the single-ply laminates. Tensile testing was performed using a servo-hydraulic tension/torsion test frame capable of loading to 222kN (50 kips). Gripping of the specimens was achieved using hydraulic wedge grips with an abrasive surface finish. Out-of-plane displacement of the specimen was carefully examined. Significant periodic out-of-plane displacement behavior was reported along the free edges under transverse tensile loading. The warping was reported to show a periodic antisymmetrical distribution of positive and negative motion at the region where the axial fiber tow is missing. In a location in the same longitudinal direction but on the opposite side, the displacement occurred in the opposite direction. This free edge effect was shown to have a significant influence on the transverse

tensile strength. In order to eliminate these edge effects, a bowtie-shaped specimen was adopted to provide a more reliable strength measurement. Bowtie-shaped specimen is design in a way that all the bias fiber tows, including $+60^\circ$ fiber tow and -60° fiber tow, extend from grip to grip and are fully clamped by the grip. Because free edge effect happens at the location where bias fiber tows ends on straight-sided coupon bar, by using the bowtie-shaped specimen the free edge effect can be eliminated. The experimental results were reported to show a much higher transverse tensile strength in the bowtie-shaped specimens than in straight-sided specimens. Moreover, a more gradual failure and higher failure stress were observed in the bowtie-shaped specimens than in straight-sided specimens.

2.2.2 Experiments conducted under dynamic loading

The fan case in a gas turbine engine is designed not only to provide an air flow path and a structure to support other engine components, but also to contain a fan blade that may accidentally be released during a fan blade-out event. Many impact experiments and numerical simulations have been conducted to simulate a blade-out event and to investigate the impact resistance capability of a composite fan case for turbine engine applications, with the overall goal of increasing aircraft safety.

Impact experiment is conducted by accelerating a projectile used for mimicking the blade into a test object (fan case, composite ring, flat panel and so on). A single-stage gas gun consisting of a pressure vessel and a gun barrel is used to accelerate the projectile. The projectile supported by a sabot is accelerated through the gun barrel toward the test object.

At the end of the gun barrel, the sabot is stopped by a sabot arrester but the project continues to impact the test object. Since conducting blade-out tests at full scale is expensive, impact experiments at different scales have been developed for different purposes, including panel impact tests for evaluation of materials as well as laboratory-scale impact tests of composite fan cases with fan blades or blade-like projectiles that are used to evaluate containment capability. In addition, post-impact structural load tests are sometimes needed to evaluate the capability of a fan case that may be impacted during engine spool down, where the fan blades slow after the engine is shut down (Roberts et al., 2009).

The characteristic properties of projectiles are known to have an effect on the evaluation of different aspects of a composite material. A soft projectile (such as one composed of soft gelatin) can induce a large region of high strain energy density in the composite material before penetration, which is useful for testing a material's resistance to cracking and delamination. A hard projectile (such as a rigid titanium disk) is often used for investigating a material's resistance to local penetration at the contact point (Roberts et al., 2009).

Roberts et al. (2004) studied failure modes at regions of high strain energy density using a soft gelatin projectile, which allowed for transferring more kinetic energy into strain energy in the panel before penetration. The tests were conducted on both a 2DTBC flat panel (a 61-cm $\times$ 61-cm square panel that was fixed along the four straight edges) and a 2DTBC half-ring. The damage was reported to initiate at the center of the back of the

panel because of the fiber tensile failure; it then propagated along the $\pm 60^\circ$ fiber direction, forming an open triangle until the projectile passed through the panel. The size of the damaged area was only slightly larger than the initial impact area. In the experiment, it was found that slippage of the fixed edges out of the test fixture had a significant effect on the penetration threshold. For the composite half-ring, which had its 0° fiber tows aligned circumferentially, the initial damage was also found to result from fiber tensile failure on the back of the impact area. The crack propagates along a direction perpendicular to the 0° fiber tow, continues along the $\pm 60^\circ$ direction for a short distance, and then propagates along the 0° fiber tows, forming two open rectangles. The damage size for the composite half-ring was also reported to be only slightly larger than the impact area, and no delamination was found.

Since a gas turbine engine fan case should be designed to contain a released fan blade during a fan blade-out event, experiments were carried out to investigate the penetration velocity threshold for various materials used for this application—such as T700s/PR520 (Roberts, 2009). It was observed that transverse failure strain for impact test and for quasi-static tests conducted on flat panel with dimensions of 61cm by 61cm by 3.2mm is very close. Furthermore, different damage patterns were observed during the impact tests on a T700/PR520 composite flat panel when struck by gelatin projectiles with different impact velocities. Different damage patterns were induced in systems with different materials.

Pereira and coworkers (Pereira et al., 2010) conducted experiments on a six-layer T700S/E862 2DTBC flat panel to simulate a jet engine blade-out event. In order to investigate how the impact resistance changes with aging, tests were conducted on as-fabricated panels as well as panels subjected to aging cycles with heat and humidity conditions that are similar to those for an in-service engine. In this test, the ballistic impact tests were conducted on several 12-in. $\times$ 12-in. panels held in a circular fixture using a projectile consisting of a thin-walled, hollow aluminum alloy cylinder (made of AL 2024) with a nominal mass of 50 g and a front face with a compound radius. The results revealed that the velocity required to penetrate the aged and unaged panels was in the range of 530 to 550 ft/sec. In addition, no significant reduction in penetration velocity was observed in panels after 344 cycles of aging when compared with as-fabricated panels. From the displacement observed on the back side of the panel when a velocity just below the penetration velocity (544 ft/sec) was used, the deformation was reported to be relatively localized at the center of the panel. When the projectile penetrated the panel, the damage induced by the projectile was diffuse, which is different from the damage pattern produced by cutting or shear failure. Aside from the impact tests, high strain rate properties of as-fabricated condition and panels subjected to aging cycles were measured using a series of Hopkinson bars to quantify the degradation in material properties due to thermal and cycling aging.

Based on the proposed theory that the damage mechanisms in a high-penetration impact event are the same as those in a quasi-static punch-shear test (Sun and Potti, 1996), Xiao et al. (2007) conducted punch-shear tests for two sets of S-2 glass/SC-15 composite plate specimens with two different span ratios held in a circular fixture. A cylindrical punch with a flat front face was employed in the test. For each set of tests, several composite panels with varies number of layers are investigated. The load–displacement behaviors of composite laminates as a function of the number of layers and the span ratios were obtained, which revealed a bi-linear stress–strain behavior up to a maximum load for thick laminates and a linear stress–strain behavior for thin laminates. For the tested material, the load–displacement behavior, energy, and screening image were studied in order to understand how the evolution of damage starts with delamination initiation and propagation and is followed by fiber failure. It was reported that tensile failure of fibers can occur at the bottommost layers during the time when the punch is progressing and the upper layers have undergone shear failure.

2.3 Numerical modeling approaches for composite structures

The material system under study in this work is a two-dimensional triaxial braided composite, a textile composite that exhibits balanced material properties and advanced impact resistance. Textile composites are structured, hierarchical materials by nature (Takano et al., 1999) with three structural levels: micro-, meso- and macro-levels (Lomov et al., 2007). Micro-level defines fiber distribution within representative volume element,

which can be used for calculation of effective material properties; meso-level defines braiding architecture of fiber tows; for macro-level the numerical model is built up without considering detail braiding architecture of fiber tows. If the overall characteristics of the inelastic response are of interest, it would be sufficient to incorporate the model at a macro level. If a particular material failure characteristic is of interest, the analysis may need to be considered at the local meso-scale with relevant details of the fiber tow undulation geometries and damage mechanisms. If the purpose is to design a material to provide certain functionalities, then it would be necessary to consider the structure at micro-scale. The structural levels and the modeling at each level are discussed in greater detail in the following paragraphs.

2.3.1 Micro-scale modeling

A composite, by definition, is composed of two or more different constituents. The micro level of composite modeling refers to the scale at which the analysis is conducted, accounting for the individual constituents that compose the material. In the literature, most of the micro-scale approaches reported to date have been applied for laminar property predictions based on an idealized micromechanical arrangement of a unidirectional fiber reinforced composite material, or the so-called *representative volume element* (RVE). Stiffness predictions are usually more reliable than strength predictions, as material failures are complicated and are accompanied by more uncertainties.

Based on the complexity of the material, different approaches at micro-level analysis can be pursued—from the simple use of the rule of mixture based on a mechanics of materials approach to a complicated, detailed finite element method that accounts for the damage and failure behaviors of various constituents. Aboudi (1991) took mechanics of materials a step forward and developed a method of cell to derive the compliance matrix of a composite based on the constituent properties. In this approach, a circular fiber is approximated by a set of square sub-cells, where each individual sub-cell can only carry the properties of one constituent: either a fiber or a matrix. Next, a closed-form equation could be derived based on an assumption of full plane stress in order to obtain the stress–strain relationship on a laminar level. Sun and Chen (1991) proposed a similar concept using square cell RVE, which they utilized to study the effect of including the plasticity nonlinearity of the matrix constituent.

Aboudi (2010) proposed a new parametric formulation for a high-fidelity generalized method of cells (HFGMC) that employed linear parametric and geometric mapping to transform the arbitrary quadrilateral cell shapes from the physical space to auxiliary uniform square shapes. In this method, a complete quadratic displacement expansion is performed in the mapped space. Using this formulation, the accuracy in the local stress field and in the damage progression were significantly improved.

The abovementioned micro-level approach is an analytical method that focuses more on the effective properties at the ultimate homogenized laminar level. Numerical

finite element methods have been used to explicitly model the composite RVE. By applying appropriate periodical boundary conditions, the effective stiffness properties can be determined. Barbero (2008) presented a finite element method based on micromechanics analysis. Since finite element-based micromechanics are computationally expensive, this method is typically used to verify the accuracy of analytical methods.

Although micromechanics has been widely used for predicting material stiffness, some limitations remain when it is used to predict strength at the micro level. Current research work only covers the method for strength prediction of the “1” direction of the material coordinate system. More investigation is needed on strength predictions for other directions.

2.3.2 Meso-scale modeling

Meso-scale modeling plays a very important role in defining the internal textile geometries as well as the orientation of fiber yarns of a textile composite. Meso-scale models represent the braiding geometry of the internal textile fabric explicitly; the fabric reinforcement and matrix are distinguished, and the material properties will be assigned accordingly. The meso-scale models are usually defined in a unit cell, where the dimension is related to the actual composite structure. For this type of model, the periodic boundary conditions need to be carefully defined. Lomov et al. (2007) produced an in-depth review of meso-scale modeling of textile composites.

The most important aspect of meso-scale modelling is the generation of the explicit fibrous structure that can capture the detailed internal geometries of composites. Meso-scale finite element modeling of a textile composite usually starts with the building of the geometry via computer-aided design (CAD) and the creation of a high fidelity mesh. A few software packages are available for textile geometry generation, and some have the capability to generate a mesh and calculate the material properties.

Sherburn (2007), who worked in the Textile Composites Research Group at the University of Nottingham, developed a user-friendly tool called *TexGen*, which combines geometry building and volume meshing algorithms and is good for handling orthogonal textile structures such as woven textiles. To conduct downstream finite element analysis, it can be linked to Abaqus, where the user is able to define the material properties.

The *WiseTex* suite, developed by Lomov and coworkers (Lomov et al., 2001) in the Composite Material Group at Katholieke Universiteit Leuven (KU Leuven), is capable of modeling the internal geometry of a textile reinforcement—such as 2D/3D woven, two- and three-axial braided and knitted structures—and transferring the data into general finite element (FE) codes. Fiber yarn cross sections and undulations can be explicitly modeled, and process modeling is enabled within the software package itself to account for compaction-induced shifting of the yarns. *WiseTex* has an embedded method of inclusion to conduct calculations for the effective modulus, while also providing the option to create

an FE mesh to perform further numerical solutions outside of the *WiseTex* package. It works with general software code like ANSYS.

Virtual Textile Manufacturing Suite (VTMS), developed at the University of Dayton Research Institute (Tandon, 2011), allows the user to simulate textile processing, including compaction effects, to produce detailed tow morphologies. VTMS can serve as a pre- and post-processor for a stress and damage analysis tool. It is capable of modeling composite processing as well as simulating progressive failure.

The abovementioned software packages can provide generalized functions for the meso-scale modeling of textile composites. Mesh generation is still a considerable obstacle in use of these techniques. To tackle such challenges, numerous research studies have been conducted where specially generated meso-scale FE models are utilized for specific investigations in local material behavior. Some specific studies are summarized in the following paragraphs.

Kim and Swan (2003) developed a voxel-based meshing scheme for the analysis of textile composites that borrowed concepts from image processing. The potential advantage of this method is that generation of the finite element mesh for the textile can be automated, requiring far less time and effort. In this work, effects of mesh refinement as well as different mesh refinement schemes were studied.

A certain amount of work has also been carried out that is related to braided textile composites. Castejon et al. (2001) carried out meso-mechanical FE analysis for braided

composites using solid elements to mesh the fiber yarns and matrix, and failure was taken into account by using the Hashin failure criteria. Song et al. (2007) predicted the strength of 2D-braided textile composites as well as damage degradation based on a meso-mechanical finite element unit cell (single and multi) model. Compressive response was studied in detail in this work. Other attempts to develop a technique to model the geometry have been made by Fang and Liang (2011), who analyzed the damage development of 3D four-directional braided composites based on a meso-scale finite element model by implementing an anisotropic damage model.

Chen (2006) and Goldberg (2011) developed a shell element-based discretization scheme to predict inhomogeneous damage crack patterns in panel impact testing. In this approach, four subcells with a through thickness integration point of the shell element were used to approximate the braiding architecture and define the local fiber orientations.

Li (2010) developed a mesoscale finite element model to further investigate details of the damage and failure behavior of triaxially braided composites. In the proposed meso-mechanical models, the three phases were modeled carefully to include the fiber tows, matrix, and the interface between the constituents. Anisotropic progressive damage law and the cohesive zone method were adopted for simulating the intralaminar and interlaminar failure, respectively. The fiber tow stiffness and strength properties were obtained by using the micro-mechanics method, and manufacturer data for the pure resin were used as the material properties of the matrix. The model was able to successfully

predict the failure process for the 2DTBC, including fiber and matrix damage initiation and propagation as well as the interface delamination process.

Zhang (2013) developed a similar model to investigate the aging response of a braided composite based on the constituent properties. Using this model, the material constants of an infinitely large plate were predicted. Free edge effect was considered, as it is known to lead to premature local damage and failure. The long-term performance was also studied by conducting thermal cycling tests.

Nie (2012) developed a meso-scale model from a new point of view, in which the constituents (fiber and matrix) were modeled using a method that does not involve the sharing of nodes. The interface between constituents was modeled using interface constraints. This model was applied to simulate flat panel impact conditions. Comparing to the simulation results obtained by using macro-mechanical model, meso-mechanical model shows the capability to capture the detailed damage propagation in fiber tows (Nie, 2012).

Lomov and coworkers (Lomov et al. 2007) developed an integrated system that includes modules such as a geometric modeler, along with a meshing engine, a material property processor, an FE solver, and a damage development processor. In this system, the geometrical model of the textile was implemented in the software package *WiseTex*, which builds a detailed internal geometry of the fabric from which the corresponding solid model is generated. In this model, a periodic boundary condition is applied to the unit cell. Yarns

and fibrous plies are considered as unidirectional assemblies of fibers. The stiffness matrix of the material is calculated based on empirical formulas or micro-FE homogenization. The parameters for the failure criterion are calculated from the strength of the unidirectional layer of composite under different loading conditions. A damage mechanics approach is adopted by employing a software package called *SACOM* (developed by M. Zako in Osaka University in the 1990s), where Hoffman's failure criterion is incorporated to define damage initiation and to indicate the damage mode, and the stiffness of damaged elements is subsequently reduced.

2.3.3 Macro-scale modeling

Macro-scale modeling approaches are aimed at building up a constitutive model for the proposed composite material. Constitutive model describes stress and strain relations, which is just mathematical simplifications of a quite complex physical behavior. In macro-scale modeling, anisotropic constitutive equations are usually obtained to describe the composite behavior. A detailed formulation of the constitutive model requires knowledge of a set of effective material parameters, such as material moduli, Poisson's ratios, and the strength of the material in different directions. In some cases, in order to define progressive failure or softening behavior, failure strain or artificial scaling factors are needed. Determination of these parameters needs to be supported by data from corresponding experimental tests, which are costly to conduct. Due to this reason, most of the composite macro-scale models are applied on unidirectional composite lamina with relatively few

material parameters. However, very few macro-scale models can be found for textile composites, due to the difficulties of including complex undulations and the crossing over of fiber yarns.

Macro-level finite element-based approaches have been developed that allow for simulation of the response of a triaxially braided composite in a manner that takes into account the architecture of the braided material as well as maintains the computational efficiency (Chen, 2006; Littell, 2008; Goldberg et al., 2011). In these macro-mechanical models, the composite is discretized into *repeating unit cells*, which are the smallest units that comprise a composite material; each unit cell is then divided into four subcells. The braiding architecture is considered using composite mechanics by treating each subcell separately as a laminated composite with the appropriate fiber layup. The smeared effective stiffness and strength material properties in each subcell can be calculated. Fiber volume fraction is calculated based on information in the manufacturer's database, and the required effective properties can then be obtained using micromechanics-based approaches. The simulation results using these approaches correlated reasonably well with experimental tests.

Cater et al. (2015) further improved macro-level modelling for triaxial braided composites by using an improved semi-analytical discretization scheme. Like the macro-scale model discussed above, a unit cell is divided into four subcells, and each is approximated using a stack of unidirectional composite plies. The difference between

Cater's model and the previous macro-scale models is that in Cater's model, the subcells are discretized by two schemes: one model captures pure matrix regions in each subcell, while the other model absorbs pure matrix pockets into neighboring tow plies in each subcell. In this work, the material properties of a composite ply are calculated using the micromechanics software *Micromechanics Analysis Code based on the Generalized Method of Cells (MAC/GMC)*, as presented in Bednarczyk and Arnold (2002).

In the work of Xiao et al. (2007), a numerical model was developed to simulate plain-weave S-2 glass/SC-15 epoxy composite laminates under quasi-static punch-shear loading within the context of LS-DYNA. In this model, the composite material model MAT_162 (based on Hashin failure criterion and damage mechanics of Matzenmiller) is utilized to account for damage and failure during a high velocity impact. Material properties are calibrated by comparing the corresponding experimental force–displacement curve and the damage profile image. Experimental material properties for a similar material group were utilized as the baseline values for the parameter calibration, and the calibrated material properties was further verified by applying them to a set of plates with several different thickness. It is observed that delamination and fiber fracture due to shear and tension are dominant damage mechanism.

Hoffarth (2016) developed an efficient orthotropic elastic–plastic progressive damage model to simulate the general failure response for a composite material under impact testing. The material model used tabulated experimental data as the direct material

input. Failure and damage were established as function of strain and damage parameters, and mesh size dependence was taken into account.

Tao et al. (2017) carried out finite element analysis of a three-dimensional woven composite structural component. In this work, the micro-scale finite element model was first built to calculate the mechanical properties of fiber tows. Next, the fiber tow-level material properties were further homogenized to material properties at macro-scale by using finite element simulation of the meso-scale representative volume element, which has an explicit fiber tow architecture modeled in *TexGen*. The structural component was modeled with the finite element software Abaqus using a shell element with the material properties obtained by the meso-scale RVE.

2.3.4 Multi-scale modeling

From the above discussion of micro-, meso- and macro-level modeling of textile composites, it is clear that the focus at each level is specific. Due to the advantages of crack resistance inherent in a textile composite, it is a popular candidate for designing large-scale structures to withstand impact loading conditions. A practical modeling approach should be used to achieve balance between accuracy and computational cost.

In recent years, as computational power has increased, large-scale and high-intensity analysis has become feasible. Multiscale approaches aimed at bridging or combining two scales have been widely investigated. Some of these approaches attempt to integrate material design and structural analysis.

Figure 2.3 Multiscale scheme and connection between different scales of modeling

Multiscale Design System (MDS) is multiscale software developed by Fish and Yu (2001b) at Rensselaer Polytechnic Institute. Theoretically, it adopted a nonlocal multiscale continuum damage model and triple-scale asymptotic analysis to derive a closed form expression relating the microscopic, mesoscopic and overall strains and damage. Numerical simulation and capabilities have been demonstrated on a 2D weave. A non-local multi-scale continuum damage model was developed for brittle composite materials in order to consider damage evolution in heterogeneous microphases. In Fish and Yu's work, damage theory based on mathematical homogenization theory was developed. Asymptotic analysis was developed for micro-scale, meso-scale, and macro-scale damage, and a closed-form expression was derived.

Digimat is a nonlinear multi-scale material and structure modeling platform developed by e-Xstream Engineering (2009). It was developed as a design tool to connect the composite manufacturing process with structural analysis. *Digimat* integrates well with commercial finite element analysis codes like LS-DYNA (MSC Software; Newport Beach, Ca.). It is capable of analyzing unidirectional, woven and braided composites.

Depending on how the two different scales communicate or connect, these multiscale analysis approaches can be categorized into three different groups: hybrid analysis schemes, submodeling schemes, or coupled homogenization schemes. The three groups and their features are described below.

- Hybrid analysis schemes
  - Combine different scales of finite element formulations of modeling into a single analysis.
  - Apply fine and comprehensive models at an activity zone of interest (e.g., a meso-scale model) while applying a coarse and computationally effective model at the far-field region (macro-scale model).
  - Apply constraint equations at the boundary and interface between two models at different scales to ensure force and displacement continuities and transitioning.
  - Solve structural response in a single analysis.
- Submodeling schemes

- Perform global analysis with a relatively coarse model and apply smeared material properties over the structural domain.
- Generate the desired sub-model at the region of interest with a fine model and detailed information.
- Extract reaction force and displacement at boundary interfaces from the global model, apply them to the submodel domain as external load and boundary conditions, then solve by a separate analysis to obtain local results.
- The drawback for submodeling schemes is that there is no interactive connection between the global and the local scale other than boundary load and/or displacement. These schemes require that cut boundaries be far away from the area of interest to prevent submodel region response from interfering with cut boundary conditions.
- Coupled homogenization schemes
  - Macro- or structural-level analysis to be conducted with the assumption that the anisotropic homogenous material properties are distributed over the entire domain of the structure.
  - At each macro-scopic material point, the homogenization subroutine at a lower scale level is called to perform a meso/micro calculation to obtain stress and damage field, assess overall material degradation if damage

exists, and feed such information back to the macro-level to finish global integration in a current time increment.

- Such coupling schemes require real-time communication at different scales at each time increment, which could be potentially time consuming. Computation time can be reduced by selectively applying a multi-scale analysis domain or establishing a fine scale material response before performing the structural analysis.

Submodeling approach schemes have matured in most generalized finite element software, like Abaqus. Abaqus provides a very handy tool and user-friendly interface for designers and researchers to define the submodel region as desired by following the established submodeling framework.

A coupled homogenization scheme typically relies on an existing meso/micro homogenization platform and then sets up an interface with commercial finite element code. Many of the software tools discussed for meso-scale modeling already provide some functionality in this respect. *MAC/GMC* links with the Abaqus commercial finite element package to enable linear and nonlinear analysis of composite structures using micromechanics to simulate the fiber and matrix behavior throughout the structure.

Pineda and coworkers (Pineda et al., 2009) developed a multi-scale progressive damage and failure analysis tool for a laminated composite component. This tool can be used to predict composite structure response, which can be widely used for rapid evaluation

of a large number of designs. In this work, a three-level, multiscale progressive damage and failure analysis tool was developed. First, *HyperSizer* analysis software (Collier Research Corporation; Newport News, Va.), is used to provide loading conditions for the structural components. The components are then analyzed using Abaqus at the lamina/laminate level, where damage and failure theories are applied to model the component. At each material point in the finite element model, the analysis can be further refined at the micro-level, where the *MAC/GMC* suite of micromechanics codes is used to perform the micromechanical analysis. The output from all three levels is shared to refine the inputs for each level.

Compared with the submodeling approach and the coupled scheme, a hybrid analysis scheme is more likely to be customized for the study of composites. Nilakantan (2010) developed a multi-scale model to study the woven composite during impact tests, which incorporated the use of multi-level finite elements. This model maintains the explicit geometry of the yarn around the impact zone, while a homogenized resolution has been used for the far field. The central patch of yarn-level resolution used a combination of solid and shell elements for modeling an individual yarn explicitly, while the surrounding homogenized zone uses shell elements.

Nie and Binienda (2012) proposed a combined meso-macro-scale (CMM) finite element multiscale analysis approach that combines the currently existing macro-scale and meso-scale approaches for modeling braided composite materials to make full use of the

advantage of each approach, with the mesoscale model using the embedded unit cell with damage and delamination to describe the details of local deformation and possible material failure, while the macro-scale model captures the global response feature over the entire structure. Later, this method was further improved and verified by using the dynamic submodeling technique (Nie et al., 2012) to improve efficiency and flexibility in implementation.

In order to combine the advantages of computational efficiency of a macro-mechanical model and the accuracy of a meso-mechanical model, a multi-scale modeling methodology is introduced in this work that incorporates a macro-mechanical model at the area far away from the area of interest while obtaining detailed damage propagation information by using meso-mechanical model at the region of interest. This proposed multi-scale modeling approach falls into the category of hybrid analysis schemes.

2.4 Current research

In this work, a multi-scale finite element analysis modeling approach in LS-DYNA has been developed to simulate the impact of triaxially braided composite structures. This approach, referred to as *multi-scale hybrid element analysis* (MHEA), uses meso-scale solid elements to discretize the area around the region of impact and transmit to macro-scale continuum shell elements in the surrounding region.

Within the impact area, a meso-mechanical FE model was used to consider the detailed braiding architecture for the purpose of investigating the mechanical behavior and

local damage initiation and propagation of fiber tows, matrix, and interface between different constituents. The fiber tows and matrix were modeled explicitly and meshed as solid elements. To mimic interface bonding and debonding between fiber tows and between the fiber tow and matrix, tiebreak contacts were defined. With a refined model, it is feasible to study the details of localized behavior that occurs in the complex constituents.

To reduce the computational expense, a macro-mechanical FE model based on an existing subcell approach was applied at the region away from the point of impact to capture the global response of the entire structure. In this subcell model, each unit cell of a triaxial braided composite was comprised of four subcells, and each subcell was represented by a continuum shell element. Fiber tow orientations and failure properties were defined at the through thickness integration points of the shell elements. By incorporating the macro- mechanical FE model, the accuracy and fineness of modeling resolution and the computational expense were well balanced.

Lastly, a surface-based tied contact was adopted in the proposed approach in order to connect solid elements in the meso-scale region and continuum shell elements in the macro-scale region. Impedances have been matched across the meso–macro interface to prevent interfacial reflections of the strain wave.

The MHEA method was examined by applying it to coupon bars in experiments where material damage and failure in both axial direction and transverse direction are studied. The MHEA method was then used to predict the results for impact tests on a flat

circular panel. The MHEA approach combines the advantages of both the meso-scale model (to describe the details of local deformation) and a macro-scale model (to capture the overall response of the entire feature).

CHAPTER III

MULTI-SCALE MODELING METHODOLOGY OF TRIAXIAL BRAIDED COMPOSITE MATERIALS

In this work, a multi-scale finite element analysis modeling approach in *LS-DYNA* has been developed to simulate impacts to triaxial braided composite structures. This approach uses meso-scale solid elements to discretize the area around the impact region and transition to macro-scale continuum shell elements in the surrounding region.

Within the impact area, a meso-mechanical FE model was used to consider the detailed braiding architecture for the purpose of investigating the mechanical behavior and local damage initiation and propagation of fiber tows, the matrix, and the interface between different constituents. The fiber tows and the matrix were modeled explicitly and were meshed as solid elements. To mimic interface bonding and debonding between fiber tows and between a fiber tow and the matrix, contacts based on the cohesive zone method were defined. By employing a refined model, it was feasible to study the details of localized behavior that occur in the complex constituents.

To reduce computational expense, a macro-mechanical FE model based on a subcell approach was applied at a region that was not in the area of the impact zone in order to capture the global response of the entire structure in a computationally efficient manner. In this subcell model, each unit cell of the triaxial braided composite is comprised of four

subcells, and each subcell is represented by a continuum shell element. By incorporating the macro-mechanical FE model, the accuracy and fineness in the modeling resolution were well balanced with the computational expense.

Finally, a surface-based tied contact in *LS-DYNA* was adopted in this multi-scale model in order to connect solid elements in the meso-scale region with the continuum shell elements in the macro-scale region. Impedances were matched across the meso–macro interface to prevent interfacial reflections of the strain wave.

3.1 Composite material system under study

Triaxially braided composites have been widely used in aerospace, automobile and other applications during the past decades due to their light weight, high strength, and lower cost. In the two-dimensional tri-axially braided (2DTBC) composites considered in this study, the 0° tows (axial fiber tows) lay straight and are equally spaced, and they define the axial direction of the composite. The bias fiber tows undulate over and under the axial tows at an angle θ (reported in degrees). In practical aerospace components, composites with a fiber architecture where θ equals 60° are of particular interest, and these are denoted as $(+60^\circ/0^\circ/-60^\circ)$ as shown in Figure 3.1. This composite material is able to conform to different mold shapes during the manufacturing process. A unit cell highlighted in Figure 3.1 is the smallest repeating geometry of a composite material that can represent the composite structure as a whole. The particular braiding architecture results in a quasi-isotropic laminate in the overall material response properties, where the global in-plane

stiffness should be the same in all directions. It has been reported that the $(+60^\circ/0^\circ/-60^\circ)$ 2DTBC can resist crack initiation and propagation as well as delamination during impact (Roberts, 2004).

Figure 3.1 Braiding structure of $(+60/0/-60)$

The material systems used in the 2DTBC considered in this study are Toray T700 carbon fiber (Toray Carbon Fibers America, Inc.) with Epon E862 resin or Cytec Cycom PR 520 RTM resin. The T700 carbon fiber is a high-strength, standard modulus, linear elastic material exhibiting brittle failure behavior when the maximum failure strain and stress is reached. It is “the highest strength, standard modulus fiber available, with outstanding processing characteristics for filament winding, weaving and prepregging” as reported by the manufacturer. The composite materials in this study contains 24,000 fibers in the axial fiber tows and 12,000 fibers in the bias fiber tows. One resin used in the 2DTBC is Epon E862 resin, a low-viscosity, high-flow thermoset resin with a cure temperature of 177°C that is ideal for resin transfer molding (Littell, 2008). The second resin, Cycom PR 520 RTM, is a one-part epoxy resin system with a cure temperature of 180°C that is

reported to have superior damage tolerance and good strain to failure required for composite primary structure applications (Solvay S.A.). The specimens used in this study to obtain experimental results for comparison with numerical results are produced using a resin transfer molding (RTM) process by the manufacturer; the fabric is loaded into a heated, rigid, closed mold and the thermoset resin of choice is injected under pressure. The air in the fabric is pushed out ahead of the resin and the resin begins to cure as it reaches the mold temperature (Kohlman, 2012). The material mechanical properties for each constituent are listed in Table 3.1 as reported by the manufacturers and some researchers (Toray Carbon Fibers America, Inc.; Hexion, Inc.; Solvay S. A.; Nie, 2012).

Table 3.1 Material mechanical properties for constituents

	Toray T700 Fiber	Epon 862	PR 520
Tensile Strength (MPa)	4900	61	82
Tensile Failure Strain ϵ_f (%)	2%	5%	5%
Longitudinal modulus E_{11} (GPa)	230	2.7	4.0
Transverse modulus E_{22} (GPa)	15	2.7	4.0
Longitudinal shear modulus G_{12} (GPa)	24	1.0	0.8
Transverse shear modulus G_{23} (GPa)	5.03	1.0	0.8
Density ρ (g/cm ³)	1.80	1.20	1.25

In this study, a fiber volume fraction of 56% is utilized to manufacture the T700s/E862 and T700s/PR520 composite material systems. Due to the presence of pure matrix pockets in the composite, the fiber volume fraction of the composite is lower than

that within the fiber tows. The fiber volume in the composite can be determined given the fiber volume fraction of the composite material. The fiber volume fraction of the fiber tows is then the ratio of the fiber volume and the volume fraction of the fiber tows.

3.2 Proposed mesomechanical model

Mesoscale models for the composite materials considered in this study are generated by modeling the individual fiber tows, pure resin pockets, and bonding interfaces explicitly. This high fidelity analysis facilitates the accurate simulation of the detailed damage patterns that are observed in the test results.

3.2.1 Mesh for mesomechanical model

In order to have insight into damage initiation and propagation behavior, the composite material architecture is carefully modeled in detail according to the actual fiber tow braiding geometry at the region of interest. Figure 3.2 presents a microscopic image of a section view of a triaxial braided composite with six layers in the through thickness direction, where the areas in black represent the axial fiber tow cross section, the area in light grey represents the bias fiber tows, and the areas in dark grey are resin pockets. From this figure, we can see that the composite material usually contains fiber tows of irregular sizes and shapes, and the plies are misaligned. However, in order to focus on the effect of the braiding geometry on the material response, the meso-scale model in this study is built to capture the major characteristics such as fiber tow undulation and braiding architecture but is idealized in terms of dimensions and arrangements of ordered fiber tows.

For the purpose of this study, the cross section of fiber tows are idealized as a lens with a uniform shape and size, and an ordered arrangement of fiber tows (Figure 3.3) is also assumed in the meso-scale numerical model. In the numerical model, the axial fiber tows are laid straight at equal spacing but without undulation, while the bias fiber tows undulate over and under the axial fiber tows. The cross sections of the fiber tows are assumed to be the same for both axial fiber tows and bias fiber tows, and the area of the cross section is obtained from the average of the cross sectional area of actual fiber tows as measured in experiments. The fiber tows are perfectly aligned in the through thickness direction for each layer.

Figure 3.2 Micro image showing a typical composite cross section with a matrix and fiber tows

Figure 3.3 Idealized architecture of a fiber tow

This meso-scale finite element model is built based on a three-dimensional (3D) representation of the composite geometry. The geometry of the each fiber tow with undulation and yarn-to-yarn braiding is modeled independently, and the volumes are discretized using 3D solid elements in *LS-DYNA* (Figure 3.4). In this figure, a unit cell is presented where the regions in pink and green represent the bias fiber tows and the region in blue represents the axial fiber tow. From this figure, we can see that the braiding architecture can be modeled well.

Figure 3.4 3D discretization of a unit cell of a triaxial braided composite

The remaining volume of the composite is filled with pure resin, where 3D solid elements are also applied to discretize the region (no resin is shown in Figure 3.4). In order to capture debonding and ply delamination of the fiber tows, interfaces are implemented at locations between the fiber tows at locations where the tows cross over each other as well

as at locations between adjacent plies. The cohesive zone method embedded in the TIEBREAK contact in *LS-DYNA* is utilized to simulate fiber tow debonding and ply delamination. A numerical model with accurate fiber tow shapes can provide a straightforward presentation of the damage evolution process.

Therefore, for the meso-mechanical model, three primary phases are considered: fiber tows, resin, and interface. In the following sections, the numerical models for these three phases will be discussed in greater detail.

3.2.2 Mechanical model of the fiber tow

Each fiber tow in a 0° , $+60^\circ$ or -60° orientation could be considered as a unidirectional composite with transversely isotropic material properties. Many micromechanics methods are available for predicting the effective material modulus and strength for a unidirectional composite. For example, the Rule of Mixture approach (Daniel, 2006) is based on the mechanics of material approach and assumes a perfect interface between the fiber and resin. The Chamis approach (Chamis, 1984) adopts a micromechanics model that is more realistic than the Rule of Mixture approach to calculate the effective moduli: it involves interspersing strips of pure matrix with strips that combine fiber and matrix. Effective strength is determined using semi-empirical expressions based on experimental values according to different failure modes. Later, a more sophisticated micromechanical model that incorporates Poisson's ratio effects, known as the Sun and Chen model, was developed. The Generalized Method of Cells (Aboudi, et al., 2012),

which can be used to calculate effective material properties, is another popular way to realize multiscale modeling. In addition to these analytical or semi-analytical methods, the finite element method is often employed to determine the effective material properties for a unidirectional composite or composite unit cells (Liang, 1997; Zhang, 2013). In finite element analysis, a representative volume element under a periodic boundary condition (usually in a hexagonal array or a cubic array), is employed to determine effective composite material properties given the material properties of the constituents. Because of the geometric characteristics, a hexagonal array can achieve a higher fiber volume fraction than a cubic array. Micromechanics methods are able to provide very good predictions for the effective properties of a material.

Composite micromechanics are adopted to generate smeared properties for the fiber tows (Table 3.2) based on the fiber and resin constituent properties. In the current work, a Chamis model based on the mechanics of materials approach is adopted to generate the material properties of a unidirectional composite material. In the approach of Chamis, the representative volume element (RVE) is assumed to be one unit-long block with a square cross section as seen in Figure 3.5. As can be noticed from this figure, the fiber is assumed to have an area of V_f and is assumed to have a square cross section. By assuming perfect bonding at the fiber/matrix interface, the mechanics of materials approach is utilized to calculate the effective material properties of a unidirectional composite as shown in Equation 3.1; the results are summarized in Table 3.2.

Figure 3.5 Cross section of a representative volume element

$$\begin{aligned}
 \text{Longitudinal modulus:} \quad E_{11} &= V_f E_{11}^f + V_m E_m \\
 \text{Transverse modulus:} \quad E_{22} = E_{33} &= \frac{E_m}{1 - \sqrt{V_f}(1 - E_m/E_{22}^f)} \\
 \text{Shear modulus:} \quad G_{12} = G_{13} &= \frac{G_m}{1 - \sqrt{V_f}(1 - G_m/G_{12}^f)} \\
 G_{23} &= \frac{G_m}{1 - V_f(1 - G_m/G_{23}^f)} \\
 \text{Poisson's ratio:} \quad v_{12} = v_{13} &= V_f v_{12}^f + V_m v_m \\
 v_{23} &= V_f v_{23}^f + V_m \left(2v_m - \frac{v_{12}}{E_{11}} E_{22} \right)
 \end{aligned} \tag{3.1}$$

where the superscripts f and m indicate fiber and matrix, respectively; V_f is the fiber volume fraction; and V_m is the matrix volume fraction, which is equal to $1 - V_f$.

For in-plane uniaxial strengths, the effective strengths are considered based on different failure modes: longitudinal tension, longitudinal compression, transverse tension, transverse compression, and in-plane shear (summarized in Equation 3.2). In the Chamis

model, three kinds of longitudinal compressive failure modes are considered: fiber strength–dominated fiber compression failure, resin strength–dominated delamination/shear failure, and resin stiffness–dominated microbuckling failure.

$$\text{Longitudinal tension:} \quad F_{11t} = V_f F_t^f \quad (3.2a)$$

Longitudinal compression:

$$\text{Fiber compression:} \quad F_{11c} = V_f F_c^f \quad (3.2b)$$

$$\text{Delamination/shear:} \quad F_{11c} = 10F_{12s} + 2.5F_t^m \quad (3.2c)$$

$$\text{Microbuckling:} \quad F_{11c} = \frac{G^m}{1 - V_f(1 - G^m/G_{12}^f)} \quad (3.2d)$$

$$\text{Transverse tension:} \quad F_{22t} = [1 - (\sqrt{V_f} - V_f)(1 - E^m/E_{22}^f)]F_t^m \quad (3.2e)$$

$$\text{Transverse compression:} \quad F_{22c} = [1 - (\sqrt{V_f} - V_f)(1 - E^m/E_{22}^f)]F_c^m \quad (3.2f)$$

$$\text{In-plane shear:} \quad F_{12s} = [1 - (\sqrt{V_f} - V_f)(1 - G^m/G_{12}^f)]F_s^m \quad (3.2g)$$

$$\text{For voids:} \quad F^m = \left\{ 1 - [4k_v/(1 - V_f)\pi]^{1/2} \right\} F^m \quad (3.2h)$$

In these equations, F_{11t} , F_{11c} , F_{22t} , F_{22c} and F_{12s} are the computed effective material properties for the unidirectional composite's longitudinal tension, longitudinal compression, transverse tension, transverse compression, and in-plane shear strengths, respectively; E^m and E^f are the Young's modulus of the matrix and fiber, respectively. Equations 3.2(b)–3.2(c) are three compression strengths that result from three longitudinal compression failure modes. A conservative approach is to select the lowest value of the three results as the final compression strength. In addition, it is better to consider the

weakness at the fiber/matrix interface, which is a very important factor causing failure and can be approximated by assuming the presence of voids as shown in Equation (3.2h).

From these equations, we can observe that for a unidirectional composite, the axial tension strength and axial fiber compression fracture are dominated by the fiber strength, while the transverse failure strengths (transverse tension, compression, and shear) are mainly dominated by the resin strength. The final effective elastic constants and failure strengths for both composite systems are summarized in Tables 3.2 and 3.3.

Table 3.2 Effective elastic constants of unidirectional fiber tows

	E_{11} (GPa)	E_{22}/E_{33} (GPa)	G_{12}/G_{13} (GPa)	ν_{12}	G_{23}
T700s/E862	186.8	10.22	6.80	0.296	3.36
T700s/PR520	189.3	11.90	8.50	0.317	4.50

Table 3.3 Failure strength of unidirectional fiber tows

	F_{11t} (MPa)	F_{11c} (MPa)	F_{22t} (MPa)	F_{22c} (MPa)	F_{12s} (MPa)
T700s/E862	3941.7	762.5	57.6	94.5	57.6
T700s/PR520	4543.2	1025.0	78.0	121.5	78.0

The mechanical behavior of the fiber tows is modeled using an existing material model in *LS-DYNA* called *MAT_ORTHOTROPIC_SIMPLIFIED_DAMAGE (also known as MAT_221), where an orthotropic composite material with optional simplified damage can be defined (Livermore Software Technology Corporation, 2007). This model is valid only for 3D solid elements with reduced or full integration. Prior to damage initiation, this material model assumes linear elastic behavior, and the material properties are calculated using the

micromechanical method introduced in previous section. A failure mode-based failure criterion, the *maximum failure strain*, is employed in this material model to capture the onset of damage. This failure criterion is a simple way to predict damage initiation without considering the coupling between strain and stress in different directions. Nine failure modes are available in the maximum failure strain failure criterion, including fiber direction tension/compression failure, matrix tension/compression failure, and shear failure as shown in Equation (3.3). When some or all of the strains meet the failure criteria, the corresponding progressive damage is initiated.

$$\begin{aligned}
\text{Tensile fiber mode:} & \quad \varepsilon_1 > \varepsilon_1^T \ (\varepsilon_1 > 0) \\
\text{Compressive fiber mode:} & \quad |\varepsilon_1| > \varepsilon_1^C \ (\varepsilon_1 < 0) \\
\text{Tensile matrix mode:} & \quad \varepsilon_2 > \varepsilon_2^T \ (\varepsilon_2 > 0); \ \varepsilon_3 > \varepsilon_3^T \ (\varepsilon_3 > 0) \quad (3.3) \\
\text{Compressive matrix mode:} & \quad |\varepsilon_2| > \varepsilon_2^C \ (\varepsilon_2 < 0); \ |\varepsilon_3| > \varepsilon_3^C \ (\varepsilon_3 < 0) \\
\text{In-plane shear mode:} & \quad |\gamma_{12}| > \Gamma_{12} \\
\text{Out-of-plane shear mode:} & \quad |\gamma_{13}| > \Gamma_{13}; \ |\gamma_{23}| > \Gamma_{23}
\end{aligned}$$

The progressive damage process can be realized in the model by applying a reduction to the three Young's moduli (E_a , E_b , E_c) and the three shear moduli (G_{ab} , G_{bc} , G_{ca}) according to the corresponding failure mode in order to improve convergence in the finite element analysis. Therefore, nine internal damage variables (Equation 3.4) are applied to the Young's moduli and shear moduli to account for the degradation in stiffness:

$d_{1t}, d_{2t}, d_{3t}, d_{1c}, d_{2c}, d_{3c}, d_{12}, d_{23}, d_{13}$. In these damage variables, the subscript t stands for tension, and the subscript c stands for compression. The number (1, 2, or 3) in the subscript implies the direction: “1” defines the fiber direction, “2” defines the in-plane transverse direction, and “3” defines the through thickness direction.

$$d_k = \max \left(d_k; D_k^c \left(\frac{\varepsilon_k - \varepsilon_k^s}{\varepsilon_k^c - \varepsilon_k^s} \right)_+ \right) \quad (3.4)$$

where $k = 1t, 2t, 3t, 1c, 2c, 3c, 12, 23, 31$; D_k^c is the critical damage; ε_k is the current strain; ε_k^s is damage threshold at which the failure criteria is satisfied for that damage mode and where progressive damage begins; and ε_k^c is the critical damage threshold.

$$\langle \cdot \rangle_+ \text{ is the positive portion: } \langle x \rangle_+ = \begin{cases} x & \text{if } x > 0 \\ 0 & \text{if } x < 0 \end{cases} \quad (3.5)$$

With the damage variables defined, the flexibility matrix after damage becomes

$$S^{dam} = \begin{pmatrix} \frac{1}{E_a(1-d_{1t/c})} & \frac{-\nu_{ba}}{E_b} & \frac{-\nu_{ca}}{E_c} & 0 & 0 & 0 \\ \frac{-\nu_{ba}}{E_b} & \frac{1}{E_b(1-d_{2t/c})} & \frac{-\nu_{cb}}{E_c} & 0 & 0 & 0 \\ \frac{-\nu_{ca}}{E_c} & \frac{-\nu_{cb}}{E_c} & \frac{1}{E_a(1-d_{3t/c})} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{ab}(1-d_{12})} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{bc}(1-d_{23})} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{ca}(1-d_{31})} \end{pmatrix} \quad (3.6)$$

In addition to the internal damage variables listed above, nine failure strains (ε_f) defined in the context of MAT_221 govern the failure of the element, the state at which the material is said to have undergone complete failure, as shown in Equation (3.7).

Tension fiber mode: $\varepsilon_{1t} \geq \varepsilon_{1tf} \ (\varepsilon_{1t} > 0)$

Compressive fiber mode: $|\varepsilon_{1c}| \geq \varepsilon_{1cf} (\varepsilon_{1c} < 0)$

Tensile matrix mode: $\varepsilon_{2t} \geq \varepsilon_{2tf} (\varepsilon_{2t} > 0); \varepsilon_{3t} \geq \varepsilon_{3tf} (\varepsilon_{3t} > 0)$ (3.7)

Compressive matrix mode: $|\varepsilon_{2c}| \geq \varepsilon_{2cf} (\varepsilon_{2c} < 0); |\varepsilon_{3c}| \geq \varepsilon_{3cf} (\varepsilon_{3c} < 0)$

In-plane shear mode: $|\gamma_{12}| \geq \Gamma_{12f}$

Out-of-plane shear mode: $|\gamma_{13}| \geq \Gamma_{13f}; |\gamma_{23}| \geq \Gamma_{23f}$

When failure occurs, elements are deleted (erosion). Under the control of the NERODE flag in *LS-DYNA*, failure may occur either when only a single integration point has failed, when several integration points have failed, or when all integrations points have failed.

Figure 3.6 Diagram showing parameters for MAT_221

An illustration of the material model used in *MAT_ORTHOTROPIC_SIMPLIFIED_DAMAGE is shown in Figure 3.6. In this figure, ϵ^s indicates the damage threshold, before which the material shows a linear elastic behavior. It can be calculated by dividing the strength by the Young's modulus in the corresponding direction. Beyond this point, damage initiates and follows the progressive damage mechanism as discussed above until the point of critical damage, ϵ^c . The post-damage behavior is determined by the critical damage level D^c , which defines the deduction of slope representing stiffness as compared to the initial stiffness of the material. Finally, the element erosion is controlled by ϵ_f .

3.2.3 Mechanical model for the matrix

Littell (2008) conducted experiments to study the mechanical properties of pure E862 resin and PR520 resin, and he found that the material exhibits an approximate elastic perfectly plastic response at low strain rates, as shown in Figure 3.7.

Figure 3.7 Stress-strain relation for pure E862 resin (figure reproduced from (Littell, 2010))

In the meso-scale model, the stress–strain curve at the lowest strain rate under room temperature conditions was applied to characterize the material property of the matrix while ignoring the effects of the strain rate.

3.2.4 Mechanical model for the fiber/matrix interface

Delamination damage is one of the dominant forms of failure in many composite material systems as a result of impact, and it can significantly reduce the load-carrying capacity of a structure. As mentioned in Chapter 2, there are three main ways to simulate the constituent interface behavior: VCCT, the cohesive zone method, and XFEM. The cohesive zone method is more appropriate when interfacial strengths are relatively weak in comparison to those of the adjoining material, as in a composite laminate. The cohesive zone method, named *Dycoss* within the context of *LS-DYNA*, is implemented through option 9 under `CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK` keyword card, which originates from `MAT_COHESIVE_MIXED_MODE`. In the current study, this option is chosen to apply the cohesive zone method through the contact properties at the interface instead of the material properties of the cohesive element due to the fact that the mesh on two opposite sides of the interface are mismatched, which makes it impossible to create a cohesive element at the interface to connect both sides.

The cohesive zone method is very well explained in the work of Camanho (Camanho, 2002). The cohesive zone method is related to Griffith's theory of fracture when the area under the traction–relative displacement relation in Figure 3.9 is equal to the

corresponding fracture toughness regardless of its shape, and a preset cohesive zone in front of the crack tip is assumed for crack growth modeling. Three fracture modes are found, and they are shown in Figure 3.8: shearing (Mode I), tearing (Mode II), and opening (Mode III).

Figure 3.8 Three types of fracture modes for a material

However, in practice, it is found that delamination usually grows in a mixed mode way that combines all three failure modes, rather than failing in a single pure mode. Although it has specific damage criteria and a specific constitutive equation, the derivative process of a mixed mode constitutive equation is based on pure mode behavior.

Figure 3.9 Traction-separation relation

For each pure mode, a bilinear traction–separation as shown in Figure 3.9 is employed, where a material point falling in the range of A to C is in an elastic undamaged state, while a material point falling in the range of C to E exhibits softened interface stiffness. A high initial stiffness K_i for each mode is used to hold the top and bottom faces of the interface together in the linear elastic range to simulate undamaged cohesive behavior at the interface. A tensile or shear strength (traction at point C) should be known for each loading mode, so that after the interfacial normal or shear tractions attain their respective strength, the stiffness of the interface is degraded until it reaches zero. Point C defines the onset of stiffness softening and also defines the onset separation δ_i^0 by Equation (3.8a). The complete decohesion of the interface point under pure mode loading is governed by the pure mode critical strain energy release rate (G_{IC} , G_{IIC} , and G_{IIIC}), which is equal to fracture toughness and is also equal to the area under the bilinear curve; thus, the corresponding failure separation δ_i^f can be obtained by Equation (3.8b).

$$\delta_i^0 = \frac{t_i^c}{K_i} \quad (3.8a)$$

$$\delta_1^f = \frac{2G_{IC}}{t_1^c}, \delta_2^f = \frac{2G_{IIC}}{t_2^c}, \delta_3^f = \frac{2G_{IIIC}}{t_3^c} \quad (3.8b)$$

where $i = 1, 2, 3$ indicates the three loading modes, and t_i^c represents the pure mode interface strength. The pure mode fracture toughness, G_{IC} , G_{IIC} , and G_{IIIC} , can be measured via experiments. For example, G_{IC} can be measured using the double cantilever beam (DCB) test for Mode I, and end notched flexure (ENF) test can be applied to measure G_{IIC}

for Mode II. However, no method for measuring of Mode III fracture toughness G_{IIIc} has been established due to issues in the experimental measurement of the transverse shear modulus G_{23} . Details regarding the experimental measurement for material fracture toughness can be found in Reeder (1992). The corresponding pure mode irreversible constitutive equation can be defined as Equation (3.9),

$$\tau_i = \begin{cases} K\delta_i & \delta_i^{max} \leq \delta_i^0 \\ (1 - d_i)K\delta_i & \delta_i^0 \leq \delta_i^{max} \leq \delta_i^f \\ 0 & \delta_i^{max} \geq \delta_i^f \end{cases} \quad (3.9)$$

$$d_i = \frac{\delta_i^f(\delta_i^{max} - \delta_i^0)}{\delta_i^{max}(\delta_i^f - \delta_i^0)}, \quad i=1,2,3; \quad d_i \in [0,1] \quad (3.10)$$

where τ_i stands for traction of mode i when the separation is δ_i ; d_i is a variable used to account for the softened stiffness during the softening process after delamination initiates; δ_i^{max} is a state variable that is used to account for the irreversible damage; and K is the undamaged initial stiffness, which is assumed to be the same in all three modes.

Mixed mode delamination can be seen as a combination of the three pure modes, and the strain energy release rate of each mode is a component of the total strain energy release rate at a particular point. However, it has been found that under mixed-mode loading, damage onset usually occurs before any traction component (τ_i) reaches its allowable strength (t_i^c) as is noted in the discussion of the mechanism for pure mode delamination. Therefore, for mixed mode loading, *LS-DYNA* includes a quadratic mixed

mode delamination criterion as shown in Equation (3.11), and the bilinear traction–separation law Equation (3.9) can be used to account for the initiation of delamination.

$$\left(\frac{\langle \tau_3 \rangle}{t_3^c}\right)^2 + \left(\frac{\tau_2}{t_2^c}\right)^2 + \left(\frac{\tau_1}{t_1^c}\right)^2 = 1 \quad (3.11)$$

where $\langle \ \rangle$ is the positive operator, which is applied on a normal opening mode due to the fact that only tension in the normal direction causes the delamination; compression in the normal direction does not contribute to delamination.

In addition, unlike pure mode loading, the separation in mixed mode loading must incorporate all three components. Therefore, the total mixed mode separation can be expressed as follows:

$$\delta_m = \sqrt{\delta_1^2 + \delta_2^2 + \langle \delta_3 \rangle^2} \quad (3.12)$$

The mixed mode total separation corresponding to the initiation state of delamination can be expressed as

$$\delta_m^0 = \begin{cases} \delta_3^0 \delta_1^0 \sqrt{\frac{1+\beta^2}{(\delta_1^0)^2 + (\beta \delta_3^0)^2}} & \delta_3 > 0 \\ \sqrt{(\delta_1^0)^2 + (\delta_2^0)^2} & \delta_3 \leq 0 \end{cases} \quad (3.13)$$

where δ_i^0 is the separation of the i th mode under pure mode loading at the state of onset of delamination and can be determined by Equation (3.8a). β is mode mix ratio and is expressed as $\beta = \sqrt{(\delta_1^0)^2 + (\delta_2^0)^2} / \delta_3$.

After delamination initiates, further loading makes the interface gradually lose its adhesive strength and become reduced to a state of complete decohesion. In *LS-DYNA*, two

widely used criteria are included to account for the complete decohesion of the material point: power law criteria and Benzeggagh–Kenane (B–K) law criteria. For the current material system under study, the B–K law criteria — a semi-empirical equation that has been proven to agree very well with experimental results and is expressed in the form of Equation (3.14) — is utilized due to its ability to consider the variation of fracture toughness as a function of mode ratio in epoxy composites (Benzeggagh, 1996).

$$G_{IC} + (G_{IIC} - G_{IC}) \left(\frac{G_{IIC}^m + G_{IC}^m}{G_{TC}^m} \right)^\eta = G_{TC} \quad (3.14)$$

where G_{IC} , G_{IIC} , and G_{IIIC} are pure mode material fracture toughness of the three modes, and G_{IC}^m , G_{IIC}^m , and G_{TC}^m are the critical strain energy release rates for each component under mixed mode loading corresponding to total decohesion, which can be determined under the respective pure mode as presented in Equation (3.15).

$$\begin{aligned} G_{IC}^m &= \int_0^{\delta_m^{3f}} \tau_3 d\delta_3 \\ G_{IIC}^m &= \int_0^{\delta_m^{2f}} \tau_2 d\delta_2 \\ G_{IIIC}^m &= \int_0^{\delta_m^{1f}} \tau_1 d\delta_1 \end{aligned} \quad (3.15)$$

where δ_m^{1f} , δ_m^{2f} , δ_m^{3f} are the separation of Mode I, Mode II, and Mode III components, respectively, under mixed mode loading; and τ_i are traction components, which can be determined by Equation (3.9). The separation under mixed mode loading corresponding to total decohesion by the B–K law, δ_m^f , will be

$$\delta_m^f = \begin{cases} \frac{2}{K\delta_m^0} \left[G_{IC} + (G_{IIC} - G_{IC}) \left(\frac{\beta^2}{1+\beta^2} \right)^\eta \right] & \delta_3 > 0 \\ \sqrt{(\delta_1^f)^2 + (\delta_2^f)^2} & \delta_3 \leq 0 \end{cases} \quad (3.16)$$

In this model, CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK is used to simulate the behavior of the interface, where a bilinear traction-separation law with a quadratic mixed mode delamination criterion and a damage formulation are included. An example of pure mode traction–separation relation curve is shown in Figure 3.10, with the critical energy release rate defined as the area under the envelope.

Figure 3.10 Pure mode bilinear traction-separation curve

The value of parameters used in the simulation are summarized in Table 3.4. G_{IC} and G_{IIC} are strain energy release rate; η is the exponent of the B–K law, a mixed mode ratio related coefficient. Due to lack of material testing data, fracture toughness values are

those for similar material systems published in the literature, and they are assumed to be the same for the two material systems in this study (Nie, 2014; Alfano, 2001). These will be discussed in more detail in the “Results” chapter.

Table 3.4 Fracture energy for composites in this study

	G_{IC} (J/mm ²)	G_{IIC} (J/mm ²)	η	t_1^c (MPa)	t_2^c (MPa)
T700/E862	0.515	1.937	1.45	122	136
T700/PR520				152	170

3.3 Macromechanical Model

For realistically large structures, it is impractical to use a mesoscale model to analyze the entire structure, as it is too computationally expensive. It is desirable to use the meso-scale model at a specific area of interest, such as the impact zone in a particular ballistic impact test, and to develop a macro-scale model to capture the key features of the overall material behavior in a computationally efficient manner. The macro-scale model is based on homogenized properties.

The macro-scale finite element model is adopted from the “braided through the thickness” analysis approach as developed by Goldberg (Goldberg, 2012). In the current work, only a brief summary of the method is introduced.

Figure 3.11 Braided through the thickness approach and subcells
(a) Top view of unit cell; and (b) braid architecture

A unit cell of a typical triaxial braided fiber preform and its discretization into the subcell based model is shown in Figure 3.11. The top view of the fiber architecture of a unit cell is shown on Figure 3.11(a), where the area in green is the axial fiber tow, the area in dark blue is the $+60^\circ$ fiber tow, and the area in light blue is the -60° fiber tow. The unit cell is divided into four subcells, which are labeled as *A*, *B*, *C* and *D*. In Figure 3.11(b), 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. The material properties of each unidirectional fiber tow can be obtained using the micro-mechanical approach introduced in the previous section. Each subcell can then be modeled as an individual continuum shell element in the finite element model as a smeared continuum shell element, with the effective stiffness properties determined using mechanics of materials-based classical laminate theory using established material properties of unidirectional fiber tows. Details regarding the determination of stiffness and strength properties of each subcell can be found in the work of Goldberg (Goldberg, 2012).

3.4 Connection between meso-mechanical and macro-mechanical models

3.4.1 Constraints at the interface of the meso and macro region

In this proposed multi-scale hybrid element approach, a meso-mechanical model with refined mesh and detailed fiber tow braiding architecture is applied at the impact region in order to obtain more detailed damage evolution results, while a macro-mechanical model is utilized in the remaining area to improve computational efficiency and capture the key feature of the material behavior (Figure 3.12).

Figure 3.12 Multi-scale hybrid element approach scheme

Appropriate definition at the constrained boundary ensures that forces and displacements are sufficiently and smoothly transferred. In the current work, a three-dimensional shell or continuum shell elements are utilized instead of a conventional two-dimensional shell element to develop the model, in order to simplify and ease the

generation of the constraints at the boundary of the meso-scale model and the macro-scale model. Constraints (Figure 3.13) and the modeling of the meso/macro boundary reinforces the continuous displacement field and force transfer. A small amount of OFFSET at contact interface is added in order to avoid strain spurring.

Figure 3.13 Connection between regions of the meso-scale model and the macro-scale model

The size of the mesoscale region can be determined in two steps. First, a full macroscale model for the entire specimen without mesoscale model embedded can be run until the specimen is fully damaged. Then, the damaged area or the area of interest can be replaced by the mesoscale model and rerun the simulation to realize the multiscale simulation.

3.4.2 Validation of interface constraint

This multi-scale hybrid element approach has been validated by conducting static coupon bar tensile tests to verify the effectiveness of the constraint at the interface of the meso-scale region and the macro-scale region. A coupon bar is cut in such a way that the axial fiber tows are perpendicular to the loading direction and the loading direction is

parallel to the longitudinal direction of the bar. The external load is applied at one end as shown in Figure 3.14.

Figure 3.14 Transverse tensile coupon bar

In Figure 3.14, the blue arrow pointing to right shows the loading direction; the meso-mechanical model is applied at the center of the coupon bar, while the macro-mechanical model is applied elsewhere. The four cross sections are labeled as *A*, *B*, *C*, and *D*. The displacement field across the bar and the resultant forces at the four cross sections are studied to verify that the displacement and force are effectively transferred through the interface.

Figure 3.15 Displacement distribution field of the transverse tensile bar

The displacement distribution field of the coupon bar during static tests (Figure 3.15) exhibits a continuous character, which indicates that displacement has been

effectively transferred through the interface of the two regions. The internal force at the four cross sections are displayed in Figure 3.16.

Figure 3.16 Resultant forces at cross sections

The resultant forces at cross sections A and D are plotted in Figure 3.16(a). Since the entire bar is taken as the free body and the reaction forces at ends A and D are plotted, the two forces in Figure 3.16(a) have opposite signs. The coupon bar is then cut at B and C , and the internal forces are plotted in Figure 3.16(b). By comparing the resultant forces at the four sections, it can be noticed that the forces are equal to each other at any time step and equal to the external force, indicating that the constraint applied at the interface can effectively transfer the forces across the boundary.

3.5 Conclusions

In this chapter, a meso-mechanical model is developed within the context of *LS-DYNA* in order to take advantage of the reliable explicit time integration algorithm and the robust contact algorithm of *LS-DYNA* when carrying out the impact simulation. The

material is modeled with detailed braiding architecture in order to capture the details of the damage initiation and evolution process. Fiber tows within this model are considered as a unidirectional composite whose effective material properties can be obtained using the Chamis approach. The interfaces between fiber tows, between a fiber tow and the matrix, and between adjacent layers all play an important role in the impact process, and these interfaces have been implemented using the cohesive zone method.

In order to improve the computational efficiency without losing the capability to capture damage details, the above meso-mechanical model is combined with a macro-mechanical model, which meshes the area using a *TSHELL* element without considering the braiding architecture. In a macro-mechanical model, the effect of the braiding geometry is considered in the calculation of material properties of the *TSHELL* element using classic theory of laminate. Since no detailed geometry is considered, the number of elements can be greatly reduced and the computational efficiency can thus be improved.

Constraint nodes in LS-DYNA are utilized to connect the meso-scale region and the macro-scale region of the composite material. By studying the displacement distribution field and internal forces, the effectiveness of the constraint is verified, giving us confidence in applying this multi-scale hybrid element approach to investigate static and dynamic behavior of the structure.

CHAPTER IV

DAMAGE AND FAILURE STUDY OF MATERIAL SYSTEM T700s/E862

Experiments have shown that the $(+60^{\circ}/0^{\circ}/-60^{\circ})$ 2DTBC can resist crack initiation and propagation as well as delamination during impact (Roberts, 2003) due to its specific braiding architecture. However, the complexity of the architecture makes it difficult to understand the damage and failure mechanisms of this material system. The meso-scale numerical model developed in the previous chapter adopts the realistic architecture of a triaxial braided composite, which enables insight into the process of localized damage evolution as well as the overall specimen failure mechanism. For specimens with large dimensions, such as flat panels used in ballistic impact tests, a multi-scale modeling technique—which combines a macro-scale model defined without detailed braiding architecture that is applied in areas away from the critical impact region and a meso-scale model with realistic braiding architecture that is applied in the critical impact region—is used to balance computational efficiency and the simulation accuracy.

In this chapter, the developed meso-scale numerical model and multi-scale modeling technique are used to study the T700s/E862 composite material system, which is representative of materials used in the aerospace industry. Three aspects of this material system will be considered: tensile testing of a six-layer straight-sided coupon bar, a free-

edge effect study of a single-layer straight-sided coupon bar, and a ballistic impact study of a six-layer flat panel.

4.1 Tensile test of a six-layer T700s/E862 straight-sided coupon bar

To evaluate the developed multi-scale model, quasi-static axial and transverse tensile tests that were conducted on six-layer ($+60^\circ/0^\circ/-60^\circ$) 2DTBC straight-sided coupons were simulated, and the simulation results were compared to the experimental results. The corresponding experiments were conducted by Littell (2008), using specimens that were 30.48 cm long and 3.58 cm wide, with six layers of material in the through thickness direction (the total thickness was 0.3175 cm), as illustrated in Figure 4.1. All tests were conducted under displacement control at rates of 0.0106 mm/sec (0.025 in/min).

Figure 4.1 Specimen configuration for axial and transverse tension test

To maintain an integer number of unit cell regions, the specimen dimensions of the finite element model used for the analyses conducted in this work were 29.9 cm long and 3.7 cm wide, with a coupon thickness of 0.396 cm. Both axial and transverse tension tests were simulated using the proposed MHEA approach. A meso-mechanical model was utilized at the center of the coupon bar, and a macro-mechanical model was applied for all other areas. The dimensions of the continuum shell elements in the macro-mechanical model were defined so as to be compatible with the adjacent subcell dimensions in the meso-mechanical model. As in the corresponding experiments, the axial tensile simulations were conducted using specimens where the axial fibers were aligned along the direction of the applied load, while the transverse tensile simulations used specimens oriented such that the axial fibers were perpendicular to the direction of the applied load. The model specimens were loaded by applying a tensile load at both ends.

Figure 4.2 Simulated axial tensile specimen with MHEA approach

Figure 4.3 Simulated transverse tensile specimen with MHEA approach

Figure 4.4 shows a comparison of the global stress–strain relations between the numerical and experimental results for the case of axial tensile loading. The predicted composite stiffness correlated with experimental results fairly well and indicated a linear response under tensile loading. In the axial tensile test, the axial fiber tows extended from one end of the specimen continuously to the other end throughout the entire length of specimen along the loading direction, forming the primary load transfer path. Therefore, in this case, the constituent interfacial effect is negligible. In Figure 4.5, the simulated transverse tensile stress–strain curve indicated a linear response at low strains and transitioned to a nonlinear response at approximately 0.2% strain. These trends were also observed in the experimental results (Littell, 2008). In contrast, the axial fiber tows in the transverse tensile test are oriented perpendicular to the loading direction, which resulted in no continuous fiber tows extending throughout the entire gage section. As a result of this composite architecture, the matrix material played a much more important role in the material response and carried a higher percentage of the load. In addition, for the transverse tensile loading case, the interfaces between the various fiber tows and between the fiber tows and the matrix pockets also played a significant role in the material response and carried a significant load. Due to the fact that they carried a significant amount of load, the matrix and the interfaces developed a high level of damage, which likely played a significant role in causing the nonlinearity observed in the stress–strain curves. From these comparisons, we can see that the MHEA approach was able to capture the overall global

Figure 4.4 Correlation of stress and strain in the axial tensile test.

Figure 4.5 Correlation of stress and strain in the transverse tensile test

behavior of the composite, including accurately predicting the initial stiffness of the composite and the nonlinearity in the stress–strain curves. However, in the transverse tensile test, the simulation results shows little discrepancy from the experimental results in terms of stiffness prediction, where the simulation gives a smaller stiffness than the experimentally measured value. The numerical model is built based on the assumption that all fiber tows are perfectly aligned in both the through thickness direction and the lateral direction (Figure 3.3), which is not the case in the realistic structure of the composite (Figure 3.2), where the fiber tows next to each other form a more compact structure. The difference in the geometry between the numerical model and a realistic composite material might be one of the main reasons for the discrepancy in the stiffness observed in Figure 4.5. Further investigation of the influence of geometry imperfections on the material properties is needed to better understand the cause of this discrepancy.

(a) Failure pattern of axial tensile bar   (b) Failure pattern of transverse tensile bar

Figure 4.6 Predicted failure pattern of coupon bar tensile test by numerical model:
(a) axial tensile test; (b) transverse tensile test.

The meso-scale finite element model used in this analysis also provided additional insights into the mechanisms of the damage and failure that occurred in the composite. As shown in Figure 4.6(a), the model predicted that in the axial tensile test, the fibers would

exhibit catastrophic breaking at the global failure stress. Meanwhile, as shown in Figure 4.6(b), in the simulation of the transverse tensile test, the failure was predicted to occur due to interfacial failure in the interface between the fiber tows. In the study by Littell (2008), it was also reported that the transverse strength of a triaxial braided composite material system significantly depends on the matrix material and the fiber–matrix interface of the composite.

For the axial tensile test, the simulation results shows that damage starts from the bias fiber tows in matrix mode at a global strain of 1.2% (which is the strain found in experiments where the global stress–strain curve turns to nonlinear) while the axial fiber tows are found to be intact with no damage (Figure 4.7(a)). Moreover, the degradation of transverse stiffness is found starting at the early stage of global strain of about 0.9%, but no damage can be observed until the global strain reaches 1.2%. Approaching the failure strain, which is predicted to be 1.77%, matrix mode damage in the bias fiber tows becomes more and more dense, while the axial fiber tows remain intact. The axial fiber tows were predicted to collapse suddenly when the global strain reached 1.77%, with the entire coupon bar failing (as shown in Figure 4.7(b)).

(a) Matrix mode damage of axial fiber tows (left) (b) Matrix mode damage of axial fiber tows (left) and transverse fiber tows (right) at strain of 1.2% transverse fiber tows (right) at failure strain of 1.77%

Figure 4.7 Matrix mode damage in the axial tensile test predicted by numerical model

(a) 25% of ultimate strength (b) 50% of ultimate strength (c) 75% of ultimate strength (d) Ultimate strength

Figure 4.8 Delamination damage in the axial tensile test predicted by numerical model

(a) Matrix mode damage of axial fiber tows (top) and transverse fiber tows (bottom) at strain of 0.5%   (b) Matrix mode damage of axial fiber tows (top) and transverse fiber tows (bottom) at failure strain of 1.7%

Figure 4.9 Matrix mode damage in the transverse tensile test predicted by the numerical model

(a) At stress of 50% of ultimate strength

(c) At ultimate strength

(b) At stress of 75% of ultimate strength

(d) At failure strain of 1.7%

Figure 4.10 Delaminated nodes in transverse tensile test predicted by the numerical model

Figure 4.8 shows the predicted delaminated nodes (which are indicated as triangles) in axial tensile tests at a global stress of 25% of ultimate strength (Figure 4.8(a)), 50% of ultimate strength (Figure 4.8(b)), 75% of ultimate strength (Figure 4.8(c)) and ultimate stress (Figure 4.8(d)). In these figures, delaminated nodes regularly spread in Subcells A and C along the bias fiber tow direction, and no delaminated nodes are found in Subcells B and D except for several places at the two ends of the coupon bar. The reason for this delamination pattern in the axial tensile test might result from the fact that the axial fiber tows are the primary load transferring path, which causes relative movement between the axial fiber tows and the adjacent bias fiber tows when an external axial load is applied. The number of delaminated nodes increases steadily throughout the loading period. However, the numerical model predicts a different delamination process that starts at the beginning of the loading and continues throughout the entire loading period; in contrast, in the experiment, no delamination occurs until the global strain reaches 75% of the ultimate strength (Littell, 2008). The number of delamination nodes is very small at the beginning of the loading, and it could be hardly noticed in the experiment using optical measurement and an ultrasonic through-transmission technique. However, when stress becomes larger and larger—say, greater than 75% of the ultimate strength—the accumulated delamination starts to be noticeable using the optical and ultrasonic equipment.

Figure 4.9(a) shows the matrix mode damage in transverse tension predicted at a global strain of 0.5%, capturing damage initiation that occurred at the edge of axial fiber

tows, which leads to the nonlinear behavior seen in the global stress–strain curve in the transverse tensile test. At this strain level, the simulation results indicate degradation of transverse stiffness starting in the bias fiber tows without any deformation in the transverse direction. As the strain reaches the numerically predicted global failure strain of 1.7% (shown in Figure 4.9(b)), the axial fiber tows split throughout the coupon bar. While the bias fiber tows also split in the transverse direction in some areas, the splitting area is not as large as that for the axial fiber tows.

Similar to the axial tensile test, the process of delamination in the transverse tensile test is also studied, as shown in Figure 4.10. From this figure, it can be noticed that four states of the meso-scale part of the coupon bar are extracted, corresponding to global stresses of 50%, 75%, and 100% of ultimate strength as well as the state corresponding to the failure strain. The simulation results reveal that the delamination in the transverse tensile test first occurs in Subcells B and D (Figure 4.10(b)), which is the opposite of the case for the axial tensile test, where the delamination is mainly focused in Subcells A and C before gradually expanding to Subcells A and C (Figure 4.10(c)). From there, the delamination quickly extends along the direction of the bias fiber tows, as shown in Figure 4.10(d), to produce the main damage pattern shown in Figure 4.6(b). The results obtained by the optical measurement of the surface strain field of the specimen and the digital microscopic images of the cross section also demonstrated that in transverse tensile tests, it is the matrix cracking that causes fiber tows to split, initiating damage propagation in the

composite coupon and thus causing the nonlinear response on the stress–strain curve (Littell, 2008). Accordingly, the highlighted area in the optical image (Figure 4.11) shows a region with high transverse strain, indicating the location where the fiber tows split.

Figure 4.11 Fiber tow splitting in the transverse tensile test (figure reproduced from (Littell, 2008))

Figure 4.10(a) also demonstrates that the delamination at the free edge can be observed as early as when global stress reaches 50% of ultimate strength, where matrix cracks begin to grow as seen in Figure 4.9(a). The matrix material and fiber tow interfaces were previously observed to have a significant effect on the transverse tensile strength of these braided composites (Li, 2010; Littell, 2008). The predicted interfacial failure in the transverse tensile test is hypothesized to occur due to the fact that in the transverse tensile test, the loads are transferred through multiple interfaces and shear stresses are introduced that tend to straighten out the bias tows in the direction of loading. In these figures, we can see that damage is initiated at the free edges, in the areas within the red circles. The failure is thus initiated at the free edges, and the free edge stress triggers delamination early in the

material loading process. The free edge effects in the transverse tension test are an important damage mechanism that will be discussed in more detail in the following section. Overall, the MHEA approach is found to accurately capture the overall material behavior as well as provide insight into the local patterns and progression of damage.

In summary, for the transverse tensile test, the damage is initiated at the free edge and then propagates as the axial fiber tows split in matrix mode damage. Meanwhile, as the material approaches the failure strain, the delamination throughout the coupon bar expands and leads to the final failure of the coupon bar.

4.2 Free edge effect of a single-layer T700s/E862 straight-sided coupon bar

Previous experimental research has revealed that single-ply straight-sided coupons exhibit significant periodic out-of-plane displacement behavior along the free edges under tensile loading, which is a result of the locally asymmetric nature of the braided structure. Static axial and transverse tensile tests of single-layer specimens were conducted by Kohlman (2012). Digital image correlation (DIC) methods were applied during the experiments to produce out-of-plane displacement contours of the region near the center of the tensile coupons for transverse loading and axial loading, as shown in Figures 4.12(a) and 4.12(b) respectively. It was reported that under transverse tensile loading, the coupons displayed warping with an anti-symmetrical distribution of positive and negative displacements as shown in Figure 4.12(a), where the axial fibers run from left to right. The areas of out-of-plane displacement alternate between a positive and a negative

displacement along the edges of the coupon where axial fiber were not represented as shown in the figure. Figure 4.12(b) presents the out-of-plane displacement contours at the center of the coupon for the axial tension test. In this case, high out-of-plane displacements occur in a direction parallel to the axial fibers, with the displacement manifesting in Subcells B and D, as reported in Kohlman (2012). The magnitude of the surface displacements in the axial tension test were approximately three times smaller than those in the transverse tension test.

Figure 4.12 DIC of out-of-plane displacement contour at global strain of 1% (figure reproduced from (Kohlman, 2012))

The corresponding numerical simulations for single-ply braided composite coupons under both transverse tension and axial tension loading conditions were carried out under conditions where the right end was fixed and displacements were applied to the left end of the model as shown in Figure 4.13. For the case of transverse tensile loading, the numerical simulation correctly captured the trend of the alternating positive and negative displacements along the free edges that were observed in the experiments. The regions of

maximum positive or negative out-of-plane displacements were predicted to occur in the regions along the free edge where bias fiber tow tips are located and where axial fibers were not present. Figure 4.13(a) also shows that the bias fiber tow tips located at Subcells B and D (where no axial fiber tows were present) experience larger out-of-plane displacement than the fiber tips located at Subcells A and C (where bias fiber tow tips are supported by the axial fiber tows beneath them). The regions of high displacement are most likely due to the fact that bias fibers do not extend through the entire length of the coupon but terminate at one of the free edges. Load can be transferred through the interaction of bias tows in the regions where they cross over one another, which was observed in Li (2010) and Zhang (2015) and can lead to the high out-of-plane displacements observed in their results.

The delamination of a coupon bar (Figure 4.10) indicates that damage initiates at the free edge where the tips of fiber tows are located. The tips of fiber tows, as we know, are the locations where stress concentrates and where damage is easily induced. Because of these high out-of-plane displacements along the free edges and the high stresses that result from these displacements, delamination and shear failures can initiate at the free edges and propagate through the length of the coupon along the bias fibers. This delamination procedure results in a delamination band along the bias fiber tow direction (shown in Figure 4.10(d)), which leads to the final failure pattern in the transverse tensile test (Figure 4.14). Moreover, in Figure 4.14(b), which shows a side view of the damaged

single layer coupon bar in the transverse tensile test, it can be seen that after damage initiates at the point of the high out-of-plane displacement along the free edge, it quickly extends through the gage area of the coupon bar, and the coupon bar exhibits a tension-induced twisting behavior.

Figure 4.13 Numerical results of out-of-plane displacement at a global strain of 1% (a) out-of-plane displacement of transverse tension, and (b) out-of-plane displacement of axial tension.

Numerical simulations of the axial tension test for the single-layer material are shown in Figure 4.13(b). In general, the alternating regions of high and low out-of-plane displacements in the middle of the coupon bar observed in the experiments are captured by the simulations, but the simulations have an out-of-plane displacement that is much smaller in magnitude. However, the locations of the high displacement regions predicted in the

(a)

(b)

Figure 4.14 Numerical prediction of failure pattern in single layer transverse tensile test
(a) top view, and (b) side view

simulations varied along the length of the coupon. Basically, the phenomenon where the area of maximum out-of-plane displacement alternates between a positive and a negative value along the edges of coupon bar is also present in the axial tensile test, where it occurs at the left and the right ends of the coupon bar in Subcells B and D. These regions, where the bias fiber tows intersect and the axial fiber tows are not present (Figure 4.13(b)), extend from the ends to the center of the coupon bar.

The similar alternating positive and negative out-of-plane displacement pattern at the left and the right ends was also observed in the work of Cater (2013). However, in Cater's work, the transition of the displacement origin from the left and right ends to the center of the coupon bar is not captured. The phenomena where asymmetric out-of-plane

displacement occurs on the opposite edges, which is observed in transverse tensile tests on free edges along the longitudinal direction (Figure 4.13(a)), is also captured in the axial tensile test. However, in contrast to the work by Cater (Cater, 2013) and Zhang (Zhang, 2014), the numerical model developed in this work predicted a different out-of-plane displacement evolution mechanism, which seems to be a combination of the results obtained by the other two researchers, as will be discussed in detail below.

At the beginning of the loading (Figure 4.15(a)), the asymmetric out-of-plane displacement is captured at two places: 1) the free edge along the longitudinal direction, and 2) at the left and the right ends of the coupon bar. From this figure, we can see that at the free edge, a similar asymmetric out-of-plane displacement pattern as the one in transverse tensile test can be observed, with maximum positive and negative out-of-plane displacements presenting at the locations where bias fiber tows end at the free edge, which is also captured in the work of (Zhang, 2014). Meanwhile, the out-of-plane displacement at the left and right ends also displays an asymmetric character that alternates in the positive and negative direction, as observed in the work of Cater (2013).

Figure 4.15 Free edge effect in axial tensile test
at a global strain of (a) 0.14% and (b) 0.5%

As the external load increases, the asymmetric out-of-plane displacements at the left and the right ends tend to extend toward the middle of the coupon bar, where they meet each other. Because of the asymmetric character of the displacements, in order for them to be compatible at the location where they meet, the maximum out-of-plane displacement slightly changes the path where they extend from the two ends to the middle of the coupon bar, and they ultimately form the pattern shown in Figures 4.13(b) and Figure 4.15(b). Therefore, the out-of-plane pattern observed in the experiment is highly local geometry dependent.

4.3 Ballistic impact of a flat panel composed of a T700s/E862 system

The multi-scale hybrid element approach has been applied to a flat panel of triaxial braided composite to investigate the material behavior during the damage process.

4.3.1 Experiment setup and numerical model

Ballistic impact of a flat panel has been studied experimentally by Pereira et al. (2010) in order to measure the impact properties of a triaxial braided composite. In this study, impact tests were conducted on flat panels using a hard aluminum projectile designed to induce damage similar to that in a jet engine fan case during a fan blade-out event. From previous experience, for a hard projectile with a sharp edge, local through-thickness shear failure due to the cutting action of the projectile is expected to be the primary initial failure during the impact process. A relatively soft gelatin projectile was utilized in order to mitigate the local initial through-thickness cutting failure, and a large,

24-in. by 24-in. composite panel held in a square fixture with a 20-in. aperture was used in an experiment by Ruggeri (2009). The design of an experiment with a soft projectile and a larger composite panel can help to reduce the local concentration of stress and cutting action that was observed in the experiments using hard projectiles with sharp edges; this test setup is commonly used to investigate the mechanism of impact energy absorption of a braided composite. However, the use of large composite panels can lead to an increase of the cost of a composite test specimen. In addition, the square fixture always induces tear failure along the edge.

To mitigate the issues with using large panels and square fixtures in impact testing of composites, Pereira et al. (2010) developed a new design for an impact experiment. In the new experimental design, a smaller-sized composite flat panel (12-in. by 12-in.) is held in place using circular fixture, and this smaller panel size helps to reduce the cost of the specimen. The projectile used is a thin-walled, hollow 2024 aluminum alloy cylinder having a front face with a compound radius designed to produce a similar deformation profile and failure mode as that in Ruggeri (2009). A single-stage gas gun consisting of a pressure vessel and a gun barrel is used to accelerate the projectile, which is supported in a cylindrical can-shaped sabot. At the end of the barrel, the sabot is stopped by a sabot arrester, and the projectile goes on to strike the composite panel (Roberts et al., 2009). The impact velocity and the exit velocity of the projectile is detected using calibrated high-speed digital video cameras. Moreover, full-field displacement and strain measurements

on the back side of the panel are obtained by an ARAMIS digital image correlation system in conjunction with a calibrated pair of Photron model SA1.1 cameras (Pereira et al., 2010).

The flat panel used in Pereira's experiment consists of six layers of a 2D triaxial braided composite with a size of 12-in. by 12-in. and composed of a T700s/E862 system, which is impacted by a thin-walled hollow 2024 aluminum alloy cylinder having a front face with a compound radius. The composite panel is held by 28 bolts that extend through the front clamp of the fixture, the specimen itself, and the rear plate of the fixture. A series of twelve impact tests were conducted with various impact velocities in order to determine the penetration threshold.

In the present work, the numerical model is developed accordingly using finite element software LS-DYNA by applying the MHEA methodology described in previous section. The setup of the numerical model is shown in Figure 4.16, where the object in blue is the aluminum projectile, the orange area in the center of the panel with a size of 7.4 cm by 9.6 cm is modeled using a meso-scale fine mesh model that covers the impact zone, while the remaining region is modeled using a macro-scale model. The size of the panel is 29.6 cm by 26.7 cm, with a 25.4-cm-diameter open area in the circular fixture (Figure 4.16(a)). In the experiment, the composite panel is well secured with no slipping at the boundary; in the numerical model, for each node on the boundary, all three translation degrees of freedom are fixed to simulate the fixed boundary condition that was present in the experiment. The aluminum projectile has standard material properties, and it can be

modeled in *LS-DYNA* using the material model MAT_PIECEWISE_LINEAR_PLASTICITY. A series of various initial velocities for the projectile, ranging from 160 m/s ~ 170 m/s were used as the impact velocity in the numerical simulation in order to predict the penetration threshold for the particular composite panel. In the experiment, the composite panel is very well fixed without slipping at the boundary; in the numerical model for each point on the boundary, all three translation are fixed to simulate the fixed boundary condition as in the experiment.

(a) Top view

(b) Side view

Figure 4.16 Panel impact model using the MHEA approach

4.3.2 Impact analysis results

Figure 4.17 Velocity history for projectiles with three different initial velocities

A series of simulations with projectiles having various impact velocities were conducted, and the resulting velocity history of the projectiles in three representative cases are shown in Figure 4.17. For the lowest impact speed of 157 m/s, the projectile struck the panel and then bounced back with a negative velocity, indicating that an initial velocity of 157 m/s is too low for a projectile to penetrate the composite panel. When the initial velocity of the projectile is increased to 165 m/s, it was found that, after impact, the velocity of the projectile was reduced to 0 m/s, which indicates that the projectile was stopped inside the panel after impact. Therefore, an initial projectile velocity of 165 m/s was found from the numerical modeling to be the penetration threshold (Figure 4.17), which falls within the range of 161 m/s~168 m/s reported by Pereira et al. (2010). If the impact velocity is

further increased to 175 m/s, the projectile will penetrate the composite panel with a residual velocity of about 20 m/s.

In the experiment, the penetration velocity for all panels fall in the range of 530 to 550 ft/sec (162 m/s ~ 168 m/s). Photographs of the damage on the front face and rear face of the panel that were induced by a projectile with a velocity of 554 ft/sec (169 m/s) are shown in Figure 4.18.

Figure 4.18 Penetrated composite panel
(a) Front view, (b) rear view (figure is reproduced from Pereira et al. (2010))

Figure 4.19 Simulation predicted damage pattern in a composite panel after impact with initial velocity of 165m/s

Figure 4.20 Simulation predicted damage pattern in a composite panel after impact with initial velocity of 100m/s

Accordingly, Figure 4.19 shows the damage pattern of the composite panel after impact that is predicted by the simulation. The simulation, which uses an impact velocity that is near the threshold velocity, predicts a large amount of bulging and results in penetration of the panel, similar to the results observed in the experiments (Pereira et al., 2010) as shown in Figure 4.18. The *front side* is the surface of the panel that faces the direction of the approaching projectile, while the *back side* of the panel is the surface on the opposite side.

Figure 4.20 shows the damage pattern after impact with the initial velocity of 100m/s. We can see from the numerical modeling results that the damage is also quite localized, and it appears to be diffuse as opposed to the cutting or shearing failure generated by a hard projectile with a sharp edge. Moreover, damage in the layer at the back side (6th layer) is found to be more severe than the damage in the layer at the front side (1st layer).

For this impact with relative low initial velocity, the front surface looks intact only with slightly damage of the matrix, while the surface on the back side shows more severe damage.

The damage evolution process for the two impact velocities (165m/s and 100m/s) are further studied in more detail as following. Figure 4.21 shows axial fiber tow damage evolution process along with time under the impact velocity of 100m/s, in which all the six layers are displayed separately in the same column that is labeled by the corresponding time step. The damage evolution process results for -60° fiber tows and $+60^\circ$ fiber tows in the same simulation are presented in Figure 4.22 and Figure 4.23. With the low impact velocity (100m/s), the projectile is not able to penetrate the composite panel and the flat panel only get slightly damaged on the backside surface. Figure 4.21 demonstrates that during impact, the axial fiber tow damage starts from the *back side* of the flat panel (the 5th layer and the 6th layer). As time goes by, axial fiber tows on the other layers are getting damages gradually from *back side* to *front side* as shown in Figure 4.21. For bias fiber tows (-60° fiber tows and $+60^\circ$ fiber tows) in Figure 4.22 and Figure 4.23, similar to axial fiber tows, damage grows from the backside layers. By the end of the impact process, the most severe damage is found on the *back side* surface of the composite flat panel, and the second severe damage is on the 5th layer (the layer adjacent to the backside surface layer), and so on. Leave the bias fiber tows on the first several layers intact, meaning bias fiber tows on the first several layers are not damaged at all (Figure 4.20). The numerical results imply

that for impact with speed as low as 100m/s, the damage are found on the backside of the composite panel.

Figure 4. 24 ~ Figure 4.26 show the damage evolution process of axial fiber tows, -60° fiber tows and $+60^\circ$ fiber tows under impact speed of 157m/s, which reveals a different damage mechanism as that in the impact with initial speed of 100m/s shown in Figure 4.21, Figure 4.22, and Figure 4.23. Different from the low speed impact, the impact with initial velocity of 157m/s indicates that the damage in axial fiber tows starts from the front side of the composite panel and followed by the quickly collapse of the axial fiber tows in all the other layers, in contrast to the case with 100m/s initial velocity where damage starts from backside of the panel and gradually extend to the front side layers.

Figure 4.21 Axial fiber tows damage evolution at impact velocity of 100m/s

Figure 4.22 -60° fiber tows damage evolution at impact velocity of 100m/s

Figure 4.23 +60° fiber tows damage evolution at impact velocity of 100m/s

Figure 4.24 Axial fiber tows damage evolution at impact velocity of 157m/s

Figure 4.25 -60° fiber tows damage evolution at impact velocity of 157m/s

Figure 4.26 +60° fiber tows damage evolution at impact velocity of 157m/s

The reason for the different damage mechanisms of the impact with lower impact velocity (100m/s) and the impact with higher impact velocity (157m/s) might be that during lower velocity impact there is enough time for the fiber tows to relax and for the strain wave to propagate from the impact point to the backside layer; while during higher velocity impact there is not enough time for the fiber tows to relax and for the strain wave to propagate so the material at impact point immediately reaches its failure strain and failed.

One of the common failure mechanisms in fiber-reinforced composite laminates is *delamination*, a failure mode which presents at the interface of two adjacent layers due to the low interface strength, resulting in separation of the layers. Since there is a lack of reinforcement in the through thickness direction, the composite relies on the relatively weak matrix to carry the load in that direction. Delamination as a result of impact or material defects can cause a significant reduction in the compressive strength of a structure as well as the in-plane strength.

Time step (ms)	Top view	Side view
$t = 0.01$		
$t = 0.012$		
$t = 0.013$		
$t = 0.02$		

Figure 4.27 Delamination propagation of a composite panel using a projectile with an impact velocity of 165m/s

Delamination is one of the major failures in composite structures, and it is typically responsible for the initial or early damage during an impact event. As mentioned previously, tiebreak contact is applied in the numerical model to define tow-to-tow and layer-to-layer initial bonding as well as the resulting damage and failure behavior. To understand the delamination pattern, the delamination propagation history for a composite panel during the early stages of the impact process is developed, and the history included the time points between 0.01 and 0.02 ms after the firing of the projectile (Figure 4.27). The projectile is propelled with an initial velocity of 165 m/s (Figure 4.19). The column at the center of Figure 4.27, shows a top view of the composite panel, looking down at the front side the panel, where the black dots denote detached nodes, indicating locations where delamination has occurred. Laterally, the area of delamination becomes larger as time progresses. The column on the right shows the side view of the delaminated panel. It shows that delamination begins from the top layer at the front side of panel, at a location right under the impact footprint, and it propagates in the through thickness direction as the panel deflects. According to the time of the impact event, it was found that delamination occurred as soon as the projectile touched the panel, and it expanded rapidly at a very early stage of impact.

CHAPTER V

DAMAGE AND FAILURE STUDY OF MATERIAL SYSTEM T700s/PR520

The previous chapter discussed the capability and improved efficiency of the proposed analytical approach in predicting the damage and failure progression of triaxial braided composites under both static and impact dynamic loading conditions. The developed meso-mechanical model, which explicitly captures braiding architecture, enables the study of damage in individual fiber bundles as well as the load transfer mechanism between the bonding interfaces of braided fiber bundles.

With increased interest in and long-term development of polymer matrix composites in various industries, a wide range of polymers have been adopted for infusing and binding the carbon reinforcement for final shape after resin consolidation. When a new analytical method is developed, the method must be investigated for material systems with different fibers and resins in order to ensure its universal application.

In this chapter, the focus will be on the application of the developed analytical approach to an alternative material system that employs the same carbon fiber braids (Toray T700s) but is infused with a different resin (Cytec PR520). Compared to E862 resin, PR520 is a toughened resin that is reported to provide superior mechanical properties. Relative impact experiment were also carried out. (Pereira, 2016) The material mechanism included in the proposed meso-mechanical model will reflect the change in resin from one

material system to another. To numerically analyze the T700s/PR520 material system, the same building block approach used for the T700s/E862 system is adopted to develop the model parameters, and further prediction will be performed using flat panel impact test results. This study will aid in understanding the robustness of the developed model for different material systems.

5.1 Experimental measurements of different polymer and composite materials

In polymer-based carbon reinforced composites, it is widely known that the carbon fibers are the primary factor responsible for the overall composite stiffness, strength and toughness. Use of a polymer matrix as a binder is typically considered in the design process to ensure shape and surface quality of the composite component as well as specific properties to meet environmental requirements such as corrosion resistance and good mechanical properties at high temperatures. However, the effect of a polymer matrix on the mechanical properties of the composite—such as bonding strength, strain rate—is also considerable.

To understand the nature of the behavior of the polymer matrix and its effects on the mechanical properties of composites, extensive experimental testing has been conducted at NASA Glenn Research Center in the last decade that targets widely used polymer resin materials. The results for these tests are available in Littell (2010), which describes a series of tests that included constituent testing of pure polymer resins and static testing of standard composite coupons. Four thermosetting polymer resins were

considered in the study: Epon 862 (as discussed in the previous chapter), Cytec PR520, Cytec 5208 and Hexcel 3502. Table 5.1 presents the mechanical properties of bulk polymers obtained from static tensile tests of the four resins.

Table 5.1 Polymer resin mechanical properties (data obtained from Littell (2010))

	Young's Modulus (GPa)	Yield stress (MPa)
Epon 862	2.7	61
Cytec PR520	4.0	82
Cytec 5208	3.8	50
Hexcel 3502	3.6	33

Resin transfer molding (RTM) was applied to infuse the T700s Toray fiber fabrics to create six-layer flat panels with a triaxial braided architecture. The fabricated panels were cut into test specimens of the desired coupon specimen dimensions for mechanical property measurement. Figures 5.1 and 5.2 show the results for Young's modulus and strength obtained in the static axial test and transverse tensile test, respectively, for standard straight-sided triaxial braided composite specimens (Littell, 2010).

From the test results for the resins and composites, it can be noticed that of the four materials, E862 (mentioned in previous chapter) provides a moderate constituent modulus and strength and yields moderate composite mechanical properties. The toughened resin PR520 produced the highest constituent modulus and strength and the highest composite mechanical properties.

Figure 5.1 Measured axial stiffness and strength of triaxial braided composites systems with four different polymer resins

Figure 5.2 Measured transverse stiffness and strength of triaxial braided composites systems with four different polymer resins

One important observation from the composite coupon tests results (Figure 5.1 and Figure 5.2) is that the various resin systems show little influence on the elastic modulus of the composite in both the axial and transverse directions. The axial moduli are all around 48 GPa, and the transverse moduli are about 42 GPa. However, the strengths of the composite specimens vary significantly from one resin system to another. This finding provides the motivation to further investigate composites with different resin systems in the current study.

5.2 Characterization of meso-mechanical model for the T700s/PR520

When leveraging the existing meso-mechanical model for a different resin system, a model parameter characterization process that is similar to the one for the T700s/E862 system is employed. Since the fiber tow braiding architecture is the same for both systems, the geometry and mesh for the resulting composites will be the same. The use of a different resin is expected to result in variation of in the material properties of the pure resin pockets and the fiber tows as well as the interfaces between constituents—especially those properties that are more sensitive to the matrix, such as fiber tow transverse and through-thickness properties.

For the macro-mechanical model, the relative effective material properties were derived by using the same procedure as discussed in Chapter 3. For a meso-mechanical model of a T700s/PR520 material system, the properties of the pure resin phase are directly obtained from the test results presented in Table 5.1. Since a fiber tow can be considered

as a unidirectional composite, micromechanics for a composite were applied to calculate the fiber tow elastic modulus and the initial value of the strength properties using the method discussed in Chapter 3. Strain, which determines the initiation of damage used as input for MAT221 in LS-DYNA, will be calculated by dividing the strength over the elastic modulus for each specific strain component. The resulting failure strains typically need to be calibrated with the results from existing experimental tests.

For cohesive bonding properties of interface between constituents, correlation to experiment tests is applied to determine the interface toughness where interface failure dominates, as no experimental data are available. It was reported in Littell (2010) that toughened PR520 provides better interface bonding properties than E862, so the interface strength of the material system for T700s/E862 should be scaled up to reflect the stronger interface property of the T700s/PR520 material system; all other parameters in the interface model will remain the same. (Please refer to Table 3.4 for the material properties of the T700s/PR520 material system). In the next section, detailed coupon prediction results based on the hybrid element model will be presented.

5.3 Static coupon testing of the T700s/PR520 material system

Numerical analysis results were compared with the experimental results for the T700s/PR520 material system for both the axial tensile test and the transverse tensile test.

5.3.1 Straight-sided coupon axial tensile test

Straight-sided coupon specimens with six layers in the through-thickness direction were used in the static tensile test. To see the numerical configuration, please refer to Figure 4.1. Axial tensile test results are shown in Figure 5.3. The image on the left shows the failure pattern after the collapse of the coupon bar. The curve on the right shows a comparison of stress–strain relationship for the experimental results to the simulation results.

Figure 5.3 Stress–strain relation for the axial tensile test

Figure 5.4 Matrix mode failure at (a) 1.5% strain and (b) 2.0% strain

The overall behavior in the axial tensile test for the T700s/PR520 system is very similar to that of the T700s/E862 system. Based on results presented in Chapter 4, the critical failure in this loading condition is controlled by the axial fiber tows that extend from end to end and form the primary load transfer path. Since the experimental results and the numerical results indicate an approximately linear stress–strain relation until failure, matrix cracking and delamination are hardly noticeable in the stress–strain curves.

Prior to catastrophic failure, the main damage observed is fiber tow transverse damage or so-called *matrix mode failure*. Figure 5.4 shows the accumulation of matrix cracking with increments in load or strain. The matrix cracking starts in the bias tows at a global strain of 1.5%, as shown in Figure 5.4(a); at this point, the axial tows are intact and are primarily in a state of axial tension. When the global strain reaches the failure strain

(2.0%), the axial fiber tow suddenly collapses. Meanwhile, from Figure 5.4(b), it can be seen that the bias fiber tows have split.

The failure mechanism for a T700s/PR520 composite material system is similar to that of the T700s/E862 material system, which is dominated by axial fiber tows that extend from end to end, forming the load transfer path. The coupon bar failed immediately after the axial fiber tows collapsed, and the failure strain of the coupon bar was close to the pure fiber failure strain of 2.1%.

5.3.2 Straight-sided coupon transverse tensile test

Transverse tensile test results for straight-sided coupons are shown in Figure 5.5, with the damage pattern for the coupon bar after failure shown on the left and the stress–strain relation curve shown on the right. As can be noticed from this figure, the stress–strain curves for the experiment and the numerical analysis show very good agreement, and the nonlinearity prior to peak loading was captured.

With the toughened resin PR520, the transverse tensile strength of the coupon bar is improved by about 25%–30% as compared to a coupon bar with E862 resin. The predicted peak load obtained from the numerical analysis is slightly lower than that obtained in the experiment. For the material system T700s/PR520, the coupon bar exhibited brittle cleavage damage (Figure 5.5), where the coupon bar failed due to fracture of the bias fiber tows, splitting of the axial fiber tows, and failure at the interface.

Figure 5.5 Stress–strain relation for the transverse tensile test

The difference in the damage and failure mechanism of the T700s/PR520 coupon bar under transverse tensile loading is obvious, in comparison to that of the T700s/E862 coupon. For the T700s/E862 material system, failure is dominated by failure at the constituent interface and the splitting of fiber tows, during which the fiber tows are pulled apart and slide against each other prior to final failure, as shown in Figure 4.6(b). In contrast to the T700s/E862 material system, the material system of T700s/PR520 exhibits less fiber tow shearing or sliding, and the final failure of the coupon bar results from bias fiber tow fracture failure (Figure 5.6(c)). Obviously, the difference in the failure mechanism in the transverse tensile test between the two material systems results from the different interface interaction. The stronger interface in the material system of T700s/PR520 leads to a higher

load transfer rate from the interface onto the bias fiber tows, making the bias fiber tows the primary load-carrying constituent.

Matrix cracking or fiber tow transverse failure is still the primary mode of failure prior to peak load. Transverse failure accumulation with strain is shown in Figure 5.6. Under transverse loading, the initial transverse failure occurs in the axial fiber tows at a global strain about 0.5% in matrix mode failure, as these fiber tows are perpendicular to the direction of loading (Figure 5.6(a)). Meanwhile, it can be observed that the location of initial matrix cracking occurs at edge of the axial fiber tow and coincides with the location where the bias tows cross. Such areas tend to have stress concentration due to the effects of local geometry. As global strain increases to 1.0%, additional matrix cracks appear in the axial fiber tows; meanwhile, the free edge effects and scissoring between bias tows cause matrix cracking on the bias tows along the free edges (Figure 5.6(b)). After peak load (Figure 5.6(c)), transverse failure only occurs on the bias tows at the breaking interfaces. This is more directly related to in-plane shear failure.

(a) Matrix failure at 0.5% strain

(b) Matrix failure at 1.0% strain

(c) Matrix failure at 1.5% strain

Figure 5.6 Matrix failure progression: (a) 0.5% strain (b) 1.0% strain and (c) 1.5% strain

Figure 5.7 Fiber tensile failures at 1.5% strain

Bias fiber tow axial tensile failure is part of the ultimate failure that occurs in the transverse loading test (Figure 5.7). With strong interface strength, the load can be successfully transferred to the bias fiber tows, and the bias fiber tows can carry a large portion of the applied load in the direction of the fibers. After achieving the fiber strength, the bias fiber tows fracture in the direction of the fiber and produce a higher failure strength as shown in the stress–strain relation (Figure 5.5). Delamination, sliding and scissoring motion between the bias tows all contribute to shear loads. As discussed in previous chapters, the testing of straight-sided specimens provides a good opportunity to examine the effects of the resin.

Investigation of delamination progression was conducted for the T700s/PR520 material system, as shown in Figure 5.8, where detached nodes (denoted as small triangles) at the interface of constituents are plotted at global strains of 0.5%, 1%, and 1.5%. In the

transverse tensile test, the constituent interface plays an important role in transferring the load from fiber tow to fiber tow. The T700s/PR520 material system utilizes toughened

(a) Delamination at 0.5% strain

(b) Delamination at 1.0% strain

(c) Delamination at 1.5% strain

Figure 5.8 Delamination progression (a) 0.5% strain (b) 1.0% strain and (c) 1.5% strain

resin that provides higher interface strength to better transfer the load between the fiber tows. The stronger interface also leads to a different mode of failure for the straight-sided coupon bar.

At the early stage of loading, delamination is only observed along the free edge of the specimen (Figure 5.8(a)), and no significant damage can be found throughout the specimen. As the global strain grows to 1.0%, the limited delamination in the specimen is only present at the edge region with confined progression into the center of the specimen. This is also the location where damage initiates. When the coupon bar is about to fail, delamination mainly concentrates at the fracture edge (Figure 5.8(c)).

5.3.3 Summary of static coupon tests

In summary, the static coupon tests on the T700s/PR520 system that are described in this section followed the same procedure as that used to accomplish the material characterization of T700s/E862. The test results indicate that the material behavior of T700s/PR520 is consistent with the behavior of a toughened resin. With the developed hybrid element model, global stress–strain behavior as well as damage patterns in static coupon tensile tests are well captured. By applying this model to various material systems (T700s/E862 and T700s/PR520), the developed numerical model has been demonstrated to be capable of accurately predicting the failure strength and failure mechanism of braided composites in the static tensile test.

5.4 Simulation of impact on a flat panel of T700s/PR520 material

Impact testing of a T700s/PR520 composite was conducted in the Ballistic Impact Laboratory at NASA Glenn Research Center in accordance with ASTM Standard D8101/D8101M-17 (Revilock, 2018). In these tests, a 12-in. $\times$ 12-in. flat panel of a T700s/PR520 composite system with a thickness of 0.125 in (i.e., a six-ply material) was fixed on a circular aperture with a 10-in.-diameter opening for impact testing. In the test setup, the composite panel was held by 28 bolts that extend through the fixture front clamp, the specimen, and the rear fixture plate. A projectile made from a thin-walled, hollow AL 2024 cylinder and having a front face with a compound radius was accelerated by a single-stage gas gun (consisting of a pressure vessel and a gun barrel) and impacted with the composite panel. An ARAMIS digital image correlation (DIC) system and a calibrated pair of Photron model SA1.1 cameras were utilized to obtain full-field displacement and strain measurements on the back side of the panel, as shown in Figure 5.9 (Revilock, 2018).

In the FE model, all three translation degrees of freedom for each node on the circular boundary are fixed to simulate the fixed boundary conditions applied in the experiment. A preliminary analysis of a T700s/PR520 impact simulation shows significant fiber failure in the macro-scale region around the meso–macro constraint interfaces. In order to avoid deviation in the prediction, the size of the meso-scale zone is expanded, with one more unit cell in the direction perpendicular to the axial fiber tow.

Figure 5.9 A T700s/PR520 flat panel in a circular fixture with DIC paint is applied to the back side of the specimen

Figure 5.10 Multi-scale model of a flat panel of T700s/PR520 material

After expansion, the meso-scale model zone contains 90 unit cells (18 unit cells in the axial direction and 5 unit cells in the transverse direction), with a height of 96 mm and a width of 92 mm according to Figure 5.10. The area in orange is the meso-scale model

region, and the rest area is the macro-scale model region with circularly fixed boundary condition denoted as denoted in black in Figure 5.10. Overall, the meso-scale model zone consists of 1,266,510 nodes and 706,320 solid elements. The correlated material model obtained from the coupon bar tensile test is applied in the impact simulation. A macro-scale model that does not consider failure and that consists of 2,840 TSHELL elements and 5,984 nodes is applied for the remaining area of the panel.

5.4.1 Impact results for the T700s/PR520 circular panel

A series of impact tests with various impact velocities were conducted to determine the penetration threshold and damage patterns of the T700s/PR520 composite panel system. The *penetration threshold* is achieved when the projectile has a residual velocity of 0 m/s after it penetrates the panel. For this material system, the experimental results reveal a penetration threshold between 165.3 m/s (542.1 ft/s) and 166.6 m/s (546.5 ft/s). Figure 5.11 shows the pattern of damage in the flat panel following impact by a projectile with an initial velocity of 165.3 m/s (542.1 ft/s), where the projectile is contained by the panel and extensive damage—such as fiber breakage, fiber tow splitting, and material loss under impact footprint—develops on both the front side and back side of the panel (Revilock, 2018). When the velocity is increased slightly, to 166.6 m/s (546.5 ft/s), penetration is complete: a through-hole is created and material is lost from the impact zone.

Figure 5.11 Damaged panel under an impact velocity of 165.3 m/s (542.1 ft/s), where the projectile is contained with extensive damage to the front side and back side

Figure 5.12 Damaged panel under an impact velocity of 166.6 m/s (546.5 ft/s), where the projectile penetrates the panel, creates a through-hole, and has a residual velocity of 30.4 m/s (99.8 ft/s).

5.4.2 Prediction results for a T700s/PR520 circular panel

Numerical analysis was conducted with the purpose of predicting the penetration threshold and damage patterns for a circular panel constructed from a T700s/PR520 material system. The multi-scale model predicted a penetration threshold at about 175 m/s (574.1 ft/s), which is slightly higher than the penetration threshold obtained in the experiment (165 m/s). Considering that the current numerical model didn't account for the thermal effect due to impact during which the temperature of the composite panel increases (Johnston, 2017), it is expected that the velocities will be slightly higher than the experimental data. Figure 5.13 shows a comparison between the experiment results and the numerical prediction for the residual velocity of a projectile with respect to the initial impact velocity (Revilock, 2018). In this figure, the dashed line indicates the residual velocity from the experiment, and the solid line is the numerical prediction.

Figure 5.13 Comparison of experimental results with the numerical prediction for the residual velocity of a projectile striking a T700s/PR520 circular panel

The rigid body velocity of the projectile in the direction of the impact, as shown in Figure 5.14, illustrates how the penetration velocity threshold is determined in the finite element analysis. For an impact velocity of 165 m/s (shown in green), the velocity of the projectile decreases to zero at a time point of approximately 0.35 ms; after that, the projectile bounced back (i.e., it had a negative velocity). For the case of an impact with an initial velocity of 175 m/s, the residual velocity of approximately 10 m/s (32.8 ft/s) is fairly close to 0 m/s, indicating the penetration threshold for the T700s/PR520 material system. With a higher impact velocity, 195 m/s, the projectile penetrates the panel and has a relatively higher residual velocity—approximately 48 m/s (157.5 ft/s). Compared to the T700s/E862 material system, the T700s/PR520 system exhibits a penetration velocity threshold that is about 10 m/s higher than that for the T700s/E862 system.

Figure 5.14 Predicted projectile velocity time history for impact velocities of 165 m/s (541.3 ft/s), 175 m/s (574.1 ft/s), and 195 m/s (639.6 ft/s) when striking a T700s/PR520 circular panel

Numerical predictions for damage patterns for the T700s/PR520 panel are shown in Figures 5.15, 5.16 and 5.17 for impact velocities of 165 m/s (541.3 ft/s), 175 m/s (574.1 ft/s) and 195 m/s (639.6 ft/s), respectively. These images show the region of impact only, as it is not practical to perform analysis for a time duration that is long enough to capture the vibration in other areas of the panel. It is difficult to compare the numerical predictions to the exact damage patterns as shown in the post-test images. Still, in comparing the predictions with the test results presented in Figures 5.11 and 5.12, it can be seen that finite element analysis is able to predict the impact damage in the area around the footprint of the cylindrical projectile, such as the significant amount of fiber damage that occurs locally around the impact area, the delamination propagation within the entire meso-scale model region, and the general amount of material loss.

(a) At $t = 0.11$ ms: front side (*left*) and back side (*right*)

Front side

Front side

Back side

Back side

(b) At $t = 0.69$ ms: image from experiment (*left*) and numerical prediction (*right*)

Figure 5.15 Panel damage at impact velocity of 165 m/s (541.3 ft/s)

(a) Front side of damaged panel in finite element analysis

(b) Back side of damaged panel in finite element analysis

Figure 5.16 Prediction of panel damage under an impact velocity of 175 m/s (574.1 ft/s)

(a) Front side of damaged panel in finite element analysis

(b) Back side of damaged panel in finite element analysis

Figure 5.17 Prediction of panel damage under an impact velocity of 195 m/s (639.6 ft/s)

At the same time, some discrepancies can be observed when comparing the test results to the finite element predictions. For the case of a panel impacted by a projectile at a velocity of 165 m/s, the finite element analysis predicts less vertical cracking than the cracking observed in the test results, and the crack tends to grow in a horizontal direction with more bias toward failure. This is especially true when comparing cases just past the threshold for penetration: the test results show a velocity of 166.6 m/s (Figure 5.12), whereas the numerical analysis predicts a velocity of 175 m/s (Figure 5.15). In the image in Figure 5.12, the tested panel has an obvious vertical crack extending over nearly the entire diameter of the impact area, and a large area of material loss can be observed on the back side of panel. However, due to the size of the meso-scale model, the finite element model is not able to capture the long vertical crack.

The progression of damage in the panel in the finite element analysis when using an impact velocity of 175 m/s (574.1 ft/s) is illustrated in Figure 5.16. The top row shows a prediction of the damage on the back side of the panel at 0.1 ms, 0.2 ms, 0.3 ms and 0.4 ms, while the bottom row shows a prediction of the impact damage on the front side. It can be observed that the initial damage to the back side of the panel occurs at 0.1 ms; more failure occurs on the back side as the panel deflects at 0.2 ms; the impact side fiber tows start to break and a hole begins to form at 0.3 ms; at 0.4 ms, all fiber tows in the impact zone fail, producing a penetration hole.

(a) Back side damage in finite element analysis

(b) Impact side damage in finite element analysis

Figure 5.18 Predicted damage progression in a panel under an impact velocity of 175 m/s (574.1 ft/s)

Overall, finite element analysis slightly over-predicts the penetration threshold and more damage observed. At lower velocity impacts, the finite element analysis tends to over-predict the damage to the panel.

5.4.2 Summary

This chapter presented the use of the proposed multi-scale approach on an alternative material system, with the same fiber material (T700s fibers) and architecture used in the previous system but infused by the toughened resin PR520. Coupon-level correlation assisted the material model characterization for the new material system. The

recalibrated model parameters showed the correct trend from a weak (E862) to a strong (PR520) resin system. Critical model parameters were identified as fiber tensile failure strain and interface strength. The correlated model was found to predict the results of the impact testing very well. The penetration threshold was correctly captured, and the damage pattern was predicted reasonably well. The study presented in this chapter enhances the fidelity of the proposed multi-scale modeling approach.

Using the same testing protocol, strain rate properties were measured for the two selected material systems (T700s/ E862 and T700s/PR520). The following chapter, which presents a model parameter sensitivity study, will briefly cover the use of the proposed meso-scale approach in a numerical study of the effect of strain rate. While the impact capability of panels with and without environmental effects (aged in environmental chamber) would also be of interest, the environmental effect is out of scope for the current study; thus, the related data will not be considered. However, a meso-scale modeling approach can provide a good fundamental platform to study the effect of the environment on the microscale properties of composite panels, and such a study could be conducted in the future.

CHAPTER VI

MODEL PARAMETER STUDY AND NUMERICAL INVESTIGATION

Based on the correlation between the experiment and the numerical analysis detailed in the previous two chapters, the proposed numerical modeling approach has been shown to be capable of capturing the damage mechanism in various material systems, and the critical numerical parameters for the model were identified by correlating the numerical results with the primary failure modes in composite structures under various loading conditions.

Since the proposed modeling approach has been validated against existing experimental data, it can be used as a tool or virtual test platform for the purpose of investigating material behaviors that are difficult to achieve in the physical world. However, as is the case for other methods mentioned in the current literature, some assumptions have been made in the developed numerical model that may affect its application. It is important to understand the effects of such assumptions by conducting sensitivity studies for various model parameters. This chapter presents some numerical studies based on the verified attributes of the proposed approach.

6.1 Resin strain rate effects

During dynamic impact loading, composite materials are loaded at different rates at different locations and time steps. Composite materials are usually strain rate sensitive,

which means that the elastic modulus and strength of the material increase with an increase in the strain rate. The composite materials under study are made from carbon fiber and epoxy resin. Carbon fibers are rate-insensitive as compared to other types of reinforcing fibers, such as those made from fiberglass, metal, and other materials. If loading rate effects exist, they may be masked by the scattering of the test data (Naito, 2013).

In the proposed meso-mechanical model, pure resin pockets were explicitly modeled, and these pockets occupy approximately 30% of the total volume of the material. The existing experimental data for composites materials show obvious strain rate effects of the resin (Littell, 2008). Meanwhile, the material properties of the resin affect the properties of the fiber tows and determine the bonding properties of the interface between the fiber tows. This section discusses a numerical investigation of the effect of the properties of the resin.

Figure 6.1 Stress–strain curves for the resin of E862 obtained from experiments conducted at various loading rates (figure reproduced from Littell, 2008)

To incorporate the experimental data into a finite element material model, it is assumed that the E862 resin behaves as an elastic-perfectly plastic material at each loading rate. Figure 6.2 shows curves of the approximate stress–strain relation and the corresponding experimental data at three strain rates. Impact testing at the penetration threshold ($v = 165 \text{ m/s}$) is selected to conduct the sensitivity study. The sensitivity study was conducted as a simplified numerical investigation, which involved performing an analysis at each strain rate level rather than developing a rate-dependent material model.

Figure 6.2 Assumed material behavior in finite element analysis at different strain rates

The FE analysis predictions for a circular panel with an initial impact velocity of 165 m/s using three distinct resin strain rates are shown in Figure 6.3. The results for a strain rate of $10E-5$ are indicated in red, those for a medium strain rate of $10E-3$ are indicated in green, and those for the highest strain rate of $10E-1$ are indicated in blue. From this figure, it can be observed that changing the strain-rate–dependent material properties

for a pure resin in a meso-mechanical model resulted in little change to the damage behavior of the panel under impact loading. This is expected, as compared to the fiber tows, a pure resin is relatively weak in terms of stiffness and strength; thus, it has little influence on the overall behavior of the panel under dynamic impact loading.

Figure 6.3 Finite element results for projectile velocity of 165 m/s with composites having different pure resin strain rate properties

However, these results also indicate that if the mechanical behavior of the triaxial braided composite under impact loading is strain-rate-dependent, the contribution from the phase of the pure resin is less than the contribution from the phase of the fiber tow and the phase of the interface between fiber tows. Since no experimental data are available for a unidirectional composite made from carbon fiber and E862 (or PR520) resin, the current study will not consider the strain rate effect resulting from the fiber tow and the interface between fiber tows. This aspect should be further investigated in future work, as the data

becomes available. However, to study the strain-rate effect resulting from the phase of the fiber tow in a multiscale analysis framework, one potential method is to use a micro-mechanical model that can generate strain-rate-dependent material properties for fiber tows in the meso-mechanical model in order to conduct an impact analysis.

6.2 Fiber tensile failure strain

In the proposed meso-mechanical model, a progressive damage modeling approach was adopted to simulate damage and failure of the fiber tows in a triaxial braided composite.

Figure 6.4 Progressive damage model of a fiber tow in a stress–strain relationship

In the progressive damage model, the material exhibits linear elastic behavior until it reaches the peak stress, known as the *tensile strength*. Further loading will lead to the initiation and accumulation of damage, which degrades the initial elastic modulus as shown in Figure 6.4.

In material model *MAT_ORTHOTROPIC_SIMPLIFIED_DAMAGE (also known as *MAT_221), two critical parameters control the overall softening behavior. One parameter is the critical damage threshold in tension, $epsclt$, corresponding to the tensile strength of the material; the other is failure strain in tension, $epsltf$, which defines the final failure (an element will become eroded as soon as the strain reaches the final failure strain in the tensile test). Failure strain, $epsltf$, is a numerical parameter, ideally determining the amount of energy dissipation that occurs during the process of fiber tensile failure. It is an empirical parameter, and calibration is needed for the building block framework.

Figure 6.5 Sensitivity of the failure strain of a fiber tow (2.00%, 2.11%, 2.15% and .50%) with respect to the projectile velocity based on a test using an impact velocity of 165 m/s

Numerical analysis of the impact was carried out using an impact velocity of 165 m/s. The effect of fiber tow failure strain on panel impact prediction in terms of projectile

velocity is shown in Figure 6.5. Panel damage patterns resulting from various failure strains are shown in Figure 6.6, where the centerline cross-sectional view of the damaged panel is presented.

The results shown in Figures 6.5 and 6.6 show a clear trend: the higher the fiber tow failure strain, the stronger the impact resistance of the panel. The results for a simulation using a fiber tow failure strain of 2.11% show the best correlation with the experimental results, predicting a penetration threshold with a residual velocity that is approximate to zero. With a lower fiber tow failure strain (2.0%), the projectile penetrates the composite panel with a residual velocity that is close to 30 m/s. At intermediate fiber tow failure strains (2.15% and 2.50%), projectiles bounce back but induce different levels of damage on the panels. A simulation using a higher failure strain (2.50%) predicts far less damage, and the fiber failure is more severe on the back side of the panel.

(a) fiber failure strain = 2.00%

(b) fiber failure strain = 2.11%

(c) fiber failure strain = 2.15%

(b) fiber failure strain = 2.50%

Figure 6.6 Cross-sectional cut of damaged panels with different failure strains

This sensitivity study shows that damage energy stored in the progressive damage material model is important in the numerical prediction results.

6.3 Effect of bonding interface

In the proposed meso-mechanical model, the interface between fiber tows exists as an individual material phase in addition to the woven fiber tows and the pure resin pockets. These interfaces play an important role in ensuring that load and deformation are successfully transferred between constituents when the interface is intact (without damage); at the same time, the model is also expected to capture delamination through its fracture mechanics-based formulation. In other words, the purpose of incorporating the bonding interface into the model is to capture the initial bonding stiffness, the crack initiation and propagation, as well as the contact of the fiber tows when debonding occurs.

In LS-DYNA, interface definition in the current meso-mechanical model is based on a type of tiebreak contact where the cohesive failure model is adopted. It is well known that the cohesive model is a model based on fracture mechanics and damage mechanics, where damage initiation criteria is utilized to trigger the irreversible damage process and where a damage propagation law is adopted to account for the crack growth. Fracture toughness or the critical strain energy release rate defines the capability of the composite to dissipate the fracture energy. In the current work, an initial value for the fracture toughness is presumed, and it is calibrated using specific experimental data. To better

understand the sensitivity, numerical studies were conducted for these parameters. The following paragraphs provide details about these numerical studies.

Interface strength is usually considered to be related to the adhesive capability of the pure resin. For the T700s/E862 material system, straight-sided transverse tensile specimens show an interface-dominated failure mechanism. Based on this observation, interface strength-related values can be correlated.

Figure 6.7 Sensitivity of interface strength (50%, 100% and 150% of baseline values) on penetration velocity based on tests using an impact velocity of 165 m/s

A sensitivity study was conducted for a flat panel with an initial velocity of 165 m/s by adjusting the value for the interface strength. Figure 6.7 shows the resulting projectile velocity history, while Figure 6.8 shows the damage predicted by the meso-scale model region at different time steps. From the projectile velocity history in Figure 6.7, it can be

noticed that a stronger interface exhibits less interface damage and prevents the projectile from penetrating the panel. Figure 6.8 shows obvious differences in damage between panels with stronger and weaker interface strength, where the panel with the higher interface strength shows less severe delamination.

Figure 6.8 Delamination and damage progression in the meso-mechanical region for materials with different interface strengths

6.4 Summary

In the work described in this chapter, the effects of several critical model parameters on the panel impact analysis were investigated through a sensitivity study. These parameters included pure resin strain rate properties, fiber tensile failure strain, fiber failure strength, and interface strength. The goal of the sensitivity study is to further demonstrate the capability of the model to capture desired failure modes as well as to illustrate the robustness of the model in predicting critical damage. The sensitivity study furthers the application of the proposed approach, along with the model correlation and validation discussed in the previous chapters.

CHAPTER VII

SUMMARY AND FUTURE WORK

In this chapter, the work that has been described in the previous chapters will be reviewed to highlight the details of the methodology employed in this study and to illustrate the significant findings. Based on the current understanding of the developed methodology, potential future work to improve the prediction accuracy of the developed numerical model and to better understand the damage and failure behavior of the triaxial braided composite will be proposed.

7.1 Summary

This work focused on a numerical study of the elastic nature and physically-based damage behavior of single-layer of six-layer triaxial braided composites under both static and dynamic impact loading conditions by using a multiscale hybrid element finite element model. This multiscale model was developed by incorporating a meso-scale model with a precisely defined braiding architecture and a macro-scale model with smeared material properties. Two material systems, T700s/E862 and T700s/PR520, representing composites with an intermediate resin and a toughened resin, respectively, were intensively investigated through both quasi-static tests and dynamic impact tests.

The development of the meso-scale finite element model was discussed in detail in Chapter 3, including the setup of the detailed fiber tow braiding geometry, the application of a material model that incorporates a progressive damage mechanism and an implementation of the interface interaction behavior. In addition, the methodology to

realize the macro-mechanical model was also presented in a step-by-step manner to demonstrate the procedure for obtaining the smeared material properties for the laminates. The boundary between the meso-scale model and the macro-scale model was handled carefully and was proved to be capable of successfully transferring the displacement and force through the meso- and macro-boundaries.

The multiscale model was calibrated and verified by comparing the experimental results for the stress–strain relation of a six-layer straight-sided coupon bar tensile test to the results of the numerical modeling. With this multiscale model, fairly accurate predictions of the elastic and damage behavior of the textile composite in static tensile tests were achieved, making us confident in moving forward with an investigation of the composite failure behavior under dynamic impact loading conditions. Because the multiscale model was proven to enhance the computational efficiency as well as to accurately predict the mechanical behavior of a triaxial braided composite, it can be utilized for a comprehensive study of the static and dynamic performance of a composite material.

Chapter 4 described the use of the developed multiscale model to study the T700s/E862 composite material system under both static and dynamic impact loading conditions to gain a comprehensive understanding of the mechanical behavior of this material system. A six-layer straight-sided coupon bar under static axial and transverse tensile loading was first simulated to study the global and local damage evolution of the triaxial braided composite. The numerical predictions of the stress–strain relation correlated reasonably well with the experimental data. It was also found that the critical failure mode varied from axial fiber tow fracture in the axial tensile test to interface shear

failure and fiber tow splitting failure in the transverse tensile test. Meanwhile, the local damage initiation and propagation in each failure mode as well as the interface delamination behavior were characterized and associated with the experimental observations, which provided significant insight into the damage and failure mechanism of the textile composite.

A meso-scale model with T700s/E862 material properties was then applied to study the free edge effect of a single-layer straight-sided coupon bar. Both the axial tensile test and the transverse tensile test were simulated, and out-of-plane displacement was found to be mainly related to the braiding architecture of the local fiber tow. The out-of-plane displacement at the center of the coupon bar in the axial tensile test displayed a different pattern from that in the transverse tensile test, which was also noticed in the corresponding experiment. Investigation of the free edge effect of a single-layer coupon bar is useful, because it shows how the local fiber tow braiding architecture of the composite results in local stress and strain concentration and initiates damage.

After a comprehensive investigation of static tensile tests, the multiscale finite element model was finally utilized to simulate the impact test of a composite panel made from T700s/E862. The penetration velocity threshold, which is a crucial factor during fan blade out in aerospace applications, was accurately predicted. The detailed modeling of the fiber tow braiding architecture in the meso-scale model enabled the damage initiation and evolution process of the composite flat panel to be well captured under impact loading. Delamination, which is one of the important failure modes during the impact process, was also examined using the multiscale model. By conducting an impact simulation, the

proposed multi-scale model was proven to be capable of predicting the penetration velocity threshold, damage pattern, and damage evolution process.

In order to validate the universality of the developed multiscale finite element model, numerical modeling was conducted on an alternative composite material system, T700s/PR520, following the same procedure as the one for the T700s/E862 material system, as described in Chapter 5. First, tensile tests on six-layer straight-sided coupon bar were simulated, and the stress–strain relation was found to correlate very well with the experimental data. The predicted penetration velocity threshold from the impact test simulation correlated reasonably well but was about 10 m/s higher than the value reported for the experiments.

Chapter 6 described sensitivity studies of the presumed parameters that were conducted to gain a better understanding and control of these presumed parameters in order to achieve the best simulation results. The effect of the strain rate of the pure resin, the failure strain of fiber tows, and the interface toughness on the penetration velocity threshold were investigated and discussed. The results of the numerical analysis indicated that the impact damage behavior of the panel was barely affected by varying the strain rate–dependent material property of the pure resin. A study of the effect of the strain rate from the fiber tows on the impact damage behavior of the panel was recommended for future work, as this could have a greater influence than the strain rate of the pure resin phase. The effect of the failure strain as an empirical parameter was evaluated, and it was found that the panel’s impact resistance increased monotonically with an increase in the failure strain.

The same conclusion was drawn for the results for the interface: a stronger interface will show less interface damage and will prevent a projectile from penetrating the panel.

Through the comparison of the two material systems, so far based on the simulation result, the T700s/PR520 has higher penetration velocity and is more recommended for using in the structure which need better penetration resistant property.

7.2 Future work

The novel multiscale model was able to incorporate the real fiber tow braiding architecture and obtain good numerical results when compared to existing experimental data. However, when generating the geometry of the textile composite, certain assumptions were made in order to reduce the difficulty of geometry generation, such as assuming that all fiber tows have the same cross-sectional dimensions throughout the entire length of the fiber tows, that fiber tows are ideally aligned in both the through-thickness direction and the lateral direction, and that axial fiber tows are straight, with no undulations. The energy absorbed by a real textile composite having an irregular fiber tow braiding architecture would be different from that of a composite with an idealized braiding structure. Therefore, it would be interesting to improve the meso-scale model by incorporating an irregular fiber tow geometry, which could be achieved by scanning the textile composite using some technique. By scanning the textile composite prior to testing, the model prediction of the energy absorbed by the composite will reflect the true energy absorbed during the impact test.

An advanced cohesive zone method was implemented in the current meso-scale model in order to simulate interface behavior (delamination and fiber–matrix debonding),

and fairly good results were obtained. During this process, empirical values for the material fracture toughness were used because of a lack of corresponding experimental data. Although the numerical modeling showed reasonably good results, experiments to determine fracture toughness are necessary to set the simulation on a solid physical foundation. Experimentally, the mixed mode material fracture toughness could be obtained by using the mixed mode bending apparatus, which combines a double cantilever beam method with an end notch flexure method.

Finally, since some resins are strain rate-sensitive and temperature-sensitive materials, it is necessary to investigate how the material behavior changes under different environmental conditions, especially in high-velocity dynamic tests such as ballistic impact tests. Although the strain rate effect was considered in the pure resin region in the current work, the strain rate effect in fiber tows—which are the major load-carrying constituent in a braided composite—should also be carefully examined. One way to study the strain rate effect was proposed in this dissertation. Another potential method to investigate the strain rate effect resulting from the phase of the fiber tows in a multiscale analysis framework is to use a micromechanical model, which can generate strain rate-dependent material properties for fiber tows in the meso-mechanical model when conducting an impact analysis.

REFERENCES

- Aboudi, J. (1991) *Mechanics of Composite Materials: A Unified Micromechanical Approach*. Elsevier, New York.
- Aboudi, J. (2010) The generalized method of cells and high-fidelity generalized method of cells micromechanical models—a review. *Mechanics of Advanced Materials and Structures*, Vol. 11, pp. 329–366.
- Aboudi, J., Arnold, S.M., and Bednarczyk, B.A. (2012) *Micromechanics of Composite Materials: A Generalized Multiscale Analysis Approach*. Elsevier, Inc.
- Altair (2018) *Multiscale Designer*, v. 2018. Troy, Michigan.
- Azzi, V.D., and Tsai, S.W. (1965) Anisotropic strength of composites. *Experimental Mechanics*, Vol. 5, pp. 283–288.
- Barbero, E.J. (2008) *Finite Element Analysis of Composite Materials*. CRC Press, Boca Raton, Florida.
- Barenblatt, G.I. (1962) Mathematical theory of equilibrium cracks in brittle failure. *Advances in Applied Mechanics*, pp. 55–129.
- Belytschko, T., and Black, T.E. (1999) Elastic crack growth in finite element with minimal remeshing. *International Journal for Numerical Methods in Engineering*, Vol. 45, pp. 601–620.
- Bednarczyk, B.A., and Arnold, S.M. (2002). “MAC/GMC 4.0 User’s Manual – Keywords Manual”. NASA Technical Memorandum, NASA/TM-2002-212077/VOL2.
- Benzeggagh, M.L., and Kenane, K. (1996) Measurement of mixed-mode delamination fracture toughness of unidirectional glass/epoxy composites with mixed-mode bending apparatus. *Composites Science and Technology*, Vol. 56, pp 439–449.
- Camanho, P.P. (2002) Failure criteria for fibre-reinforced polymer composite. Universidade do Porto. Available at <https://paginas.fe.up.pt/~stpinho/teaching/feup/y0506/fcriteria.pdf>.
- Camanho, P.P., and Davila, C.G. (2002) Mixed-mode decohesion finite elements for the simulation of delamination in composite materials. NASA/TM-2002-211737. NASA Langley Research Center, Hampton, Virginia.

Castejon, L., Miravete, A., and Cuartero, J. (2001) Analytical formulation of (00, $\pm\alpha$ 0) braided composites and its applications in crashworthiness simulations. *Mechanics of Composite Materials and Structures*, Vol. 8, pp. 219–229.

Cater, C., Xiao, X., Goldberg, R.K., and Kohlman, L.W (2015) Experimental and numerical analysis of triaxially braided composites utilizing a modified subcell modeling approach. *American Society for Composites–30th Technical Conference, ACS 2015*. DE Stech Publications.

Chamis, C.C. (1969) Failure criteria for filamentary composites. NASA rrrrote, NASA TN D-5367, Lewis Research Center, Cleveland, Ohio.

Chang, F.K., and Chang, K.Y. (1987) A Progressive Damage Model for Laminated Composites Containing Stress Concentrations. *Journal of Composite Materials*, Vol. 21, pp. 834–855.

Chen, J. (2006) Material modeling of strain rate dependent polymer and 2D tri-axially braided composite. Ph.D. Dissertation. The University of Akron, Akron, Ohio.

Christensen, R.M. (1997) Stress Based Yield/Failure Criteria for Fiber Composites. *Journal of Solid and Structures*, Vol. 34, No. 5, pp. 529–543.

Cui, W., and Wisnom, M.R. (1993) A combined stress-based and fracture mechanics-based model for predicting delamination in composites. *Composites*, Vol. 24, pp. 467–474.

Daniel, I.M., and Ishai, O. (2006) Engineering Mechanics of Composite Materials. Oxford University Press, New York and Oxford.

Dassault Systèmes, (2018) ABAQUS 6.14 Online Documentation. Waltham, Mass., USA. <http://abaqus.software.polimi.it/v6.14/index.html> Accessed on March 30.

de Moura, M.F., Goncalves, J.P., Marques, A.T., and de Castro, P.T. (1997) Modeling compression failure after low velocity impact on laminated composites using interface elements. *Journal of Composite Materials*, Vol. 31, pp. 1462–1479.

Dudgale, D.S. Yielding of steel sheets containing slits. *Journal of Mechanics and Physics of Solids*, Vol. 8, pp. 100–104, 1960.

Engblom, J.J., and Yang, Q. Finite element based sub-laminate damage model for intraply cracking. *Journal of Reinforced Plastics and Composites*, Vol. 14, pp. 233–254, 1995.

e-Xstream engineering (2009) *Digimat* v. 3.2.1. Hautcharage, Luxembourg. Available at <http://www.e-xstream.com>

Fang, G., and Liang, J. (2011) A review of numerical modeling of three-dimensional braided textile composites. *Journal of Composite Materials*, Vol. 45, pp. 2415–2436.

Fish, J., and Yu, Q. (2001a) Two scale damage modeling of brittle composites. *Composites Science and Technology*. Vol. 61, pp. 2215–2222.

Fish, J., and Yu, Q. (2001b) Multiscale damage modeling for composite materials: Theory and computational framework. *International Journal for Numerical Methods in Engineering*, Vol. 52, pp. 151–191.

Frantziskonis, S. (1988) Distributed damage in composites, theory and verification. *Composite Structures*, Vol. 10, pp. 165–184.

Goldberg R. K., Blinzler B. J., and Binienda W. K. (2011) Investigation of a macromechanical approach to analyzing triaxially-braided polymer composites. *AIAA Journal*, Vol. 49, pp 205–215.

Griffith, A.A. (1921) The phenomena of rupture and flow in solids. *Philosophical Transactions of the Royal Society of London*, Vol. 221, pp. 163–198.

Hashin, Z., and Rotem, A.A. (1973) Fatigue failure criterion for fibre reinforced materials. *Journal of Composite Materials*, Vol. 7, pp. 448–464.

Hashin, Z. (1980) Failure criteria for unidirectional fiber composites. *Journal of Applied Mechanics*, Vol. 47, pp. 329–334.

Herakovich, C.T. (1998) *Mechanics of Fibrous Composites*. Wiley, New York.

Hexion, Inc.. Collumbus, Ohio, USA. <http://www.hexion.com/en-us/brand/epon>.

Hill, R.A. (1948) theory of the yielding and plastic flow of anisotropic materials. *Proceedings of the Royal Society, Series A*, Vol. 193, pp. 281.

Hinton, M.J., Kaddour, A.S., and Soden, P.D. (2004) A further assessment of the predictive capabilities of current failure theories for composite laminates: comparison with experimental evidence. *Composites Science and Technology*, Vol. 64, pp. 549–588.

Hoffarth, C. (2016) A generalized orthotropic elasto-plastic material model for impact analysis. Ph.D. Dissertation. Arizona State University, Tempe, Arizona.

Hoffman, O. (1967) The brittle strength of orthotropic materials. *Journal of Composite Materials*, Vol. 1, pp. 200–206.

Irwin, G.R. (1956) Onset of fast crack propagation in high strength steel and aluminum alloys. Proceedings of the Second Sagamore Ordnance Materials Conference, Durham, N.C., December 5–7. Vol. 2, pp. 289–305.

Irwin, G.R. (1957) Analysis of Stresses and Strains Near the End of a Crack Traversing a Plate, *J. Appl. Mech.*, Vol. 24, pp. 361- 364.

Johnston, J.P. Pereira, J.M., Ruggeri, C.R., and Roberts, G.D. (2017) High speed thermal imaging on ballistic impact of triaxially braided composites. *32nd Annual Technical Conference of the American Society for Composites (ASC)*, West Lafayette, US.

Juhasz, T.J., Rolfes, R., and Rohwer, K.A. (2001) New strength model for application of a physically based failure criterion to orthogonal 3D fiber reinforced plastics. *Composite Science and Technology*, Vol. 61, pp. 1821–1932.

Kim, H.J., and Swan, C.C. (2003) Voxel-based meshing and unit-cell analysis of textile composites. *International Journal for Numerical Methods in Engineering*. Vol. 56, pp. 977–1006.

Kimiyoshi N. (2014) Effect of Strain Rate on Tensile Properties of Carbon Fiber Epoxy-Impregnated Bundle Composite. *Journal of Materials Engineering and Performance*. Vol. 23 (3), pp 708–714.

Kohlman, L.W. (2012) Evaluation of test methods for triaxial braid composites and the development of a large multiaxial test frame for validation using braided tube specimen. Ph.D. Dissertation. The University of Akron, Akron, Ohio.

Li, X., Binienda, W.K., and Goldberg, R.K. (2010) Finite element model for failure study of two-dimensional triaxially braided composite. NASA/TM-2010-216372. NASA Glenn Research Center, Cleveland, Ohio.

Li, X., Binienda, W.K., and Goldberg, R.K. (2011) Finite-element model for failure study of two-dimensional triaxially braided composite. *Journal of Aerospace Engineering*, Vol. 24, pp. 170–180.

Liang, J., Du, S., and Han, J. (1997) Effective elastic properties of three dimensional braided composites with matrix micro-cracks. *Acta Mater. Compos. Sinica*, Vol. 14, 101-107.

Littell, J. (2008) The Experimental and analytical characterization of the macromechanical response for triaxial braided composite. Ph.D. Dissertation. The University of Akron, Akron, Ohio.

Livermore Software Technology Corporation (2007) LS-DYNA Keyword Manual v. 971, Livermore, California.

Lomov, S.V., Huysmans, G., Luo, Y., Parnas, R., Prodromou, A., and Verpoest, I. (2001) Textile composites: Modeling strategies. *Composites: Part A*, Vol. 32, pp. 1379–1394.

Lomov, S.V., Ivanov, D.S., Verpoest, I., Zako, M., Kurashiki, T., Nakai, H., and Hirose, S. (2007) Meso-FE modelling of textile composites: Road map, data flow and algorithms. *Composites Science and Technology*, Vol. 67, pp. 1870–1891.

Maimi P., Camanho, P.P., Mayugo, J.A., and Davila, C.G. (2007) A continuum damage model for composite laminates: Part I – Constitutive model. *Mechanics of Materials*, Vol. 39, pp. 897–908.

Matzenmiller, A., Lubliner, J., and Taylor, R.L. (1995) A constitutive model for anisotropic damage in fiber-composites. *Mechanics of Materials*, Vol. 20, pp. 125–152.

Moës, N., Dolbow, J., and Belytschko, T. (1999) A finite element method for crack growth without remeshing. *International Journal for Numerical Methods in Engineering*, Vol. 46, pp. 131–150.

Needleman, A. (1987) A continuum model for void nucleation by inclusion debonding. *Journal of Applied Mechanics*, Vol. 54, pp. 525–531.

Nie, Z., and Binienda, W.K. (2012) A meso-macro mechanical approach to analyzing triaxially braided composites with failure subjected to impact. Presented at Thirteenth ASCE Aerospace Division Conference on Engineering, Science, Construction, and Operations in Challenging Environments, and the 5th NASA/ASCE Workshop On Granular Materials in Space Exploration, Pasadena, California, April 15–18.

Nie, Z., Binienda, W.K., Goldberg, R.K., and Li, X. (2012) Applying the submodeling technique to ballistic impact of textile composite structures. NAFEMS North America Conference, Washington, D.C., September 11–12.

Nie, Z. (2014) Advanced mesomechanical modeling of triaxially braided composites for dynamic impact analysis with failure. Ph.D. Dissertation. The University of Akron, Akron, Ohio.

Nilakantan, G., Keefe, M., Bogetti, T.A., Adkinson, R., and Gillespie, J.W. (2010) On the finite element analysis of woven fabric impact using multiscale modeling techniques. *International Journal of Solids and Structures*, Vol. 47, pp. 2300-2315.

Paley, M and Aboudi J. (1992) Micromechanical analysis of composites by the generalized cells model. *Mechanics of Materials*, Vol. 14, pp. 127–139.

Paris, F. (2001) A study of failure criteria of fibrous composite materials. NASA/CR-2001-210661. NASA Langley Research Center, Hampton, Virginia.

Pathak, H., Singh A., Singh I.V., and Yadav, S.K. (2013) A simple and efficient XFEM approach for 3D cracks simulations. *International Journal of Fracture*, Vol. 181, pp. 189–208.

Pereira, M.J., Roberts, G.D., Ruggeri, C.R., Gilat, A., and Matrka, T. (2010) Experimental techniques for evaluating the effects of aging on impact and high strain rate properties of triaxial braided composite materials. NASA/TM-2010-216763. NASA Glenn Research Center, Cleveland, OH.

Pereira, M.J., Revilock, D.M., Ruggeri, C.R., Roberts, G.D., Kohlman, L.W., and Miller, S.G. (2016) The effects of hygrothermal aging on the impact penetration resistance of triaxially braided composites. 31st American Society for Composites Technical Conference, Williamsburg, VA, US.

Pineda, E.J., Waas, A.M., Collier, C.S., and Bednarczyk, B.A. (2009) Multiscale model for progressive damage and failure of laminated composites using an explicit finite element method. 50th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference, Palm Springs, California, May 4–7.

Puck, A., and Schurmann, H. (1998) Failure Analysis of FRP Laminates by means of Physical Based Phenomenological Models. *Composites Science and Technology*, Vol. 58, pp. 1045–1067.

Raju, I.S. (1987) Calculation of strain-energy release rates with higher order and singular finite elements. *Engineering Fracture Mechanics*, Vol. 28, pp. 251–274.

Reeder, J.R. (1992) An evaluation of mixed-mode delamination failure criteria. NASA/TM-104210. NASA Langley Research Center, Hampton, Virginia.

Revilock, D. (2018) personnel communication. NASA Glenn Ballistic Impact Lab, Cleveland, OH.

Roberts, G.D., Pereira, M.J., Revilock, D.M., Binienda, W.K., Xie, M., and Braley, M. (2004) Ballistic impact of braided composites with a soft projectile. NASA/TM-2004-212973. NASA Glenn Research Center, Cleveland, Ohio.

- Roberts, G.D., Pereira, M.J., Braley, M.S., Arnold, W.A., Korner, J.D., and Watson, W.R. (2009) Design and testing of braided composite fan case materials and components. NASA/TM-2009-215811. NASA Glenn Research Center, Cleveland, Ohio.
- Ruggeri, C. (2009) “High strain rate data acquisition of D braided composite substructures.” M.Sc. Thesis, Dept. of Civil Engineering, The University of Akron, Akron, Ohio. December.
- Rybicki, E.F., and Kanninen, M.F. (1977) A finite element calculation of stress intensity factors by a modified crack closure integral. *Engineering Fracture Mechanics*, Vol. 9, pp. 931–938.
- Sherburn, M. (2007) Geometric and mechanical modeling of textiles. Ph.D. Dissertation. University of Nottingham, Nottingham, U.K.
- Solvay S. A.. Neder-Over-Heembeek, Brussels, Belgium.
http://www.cyttec.com/sites/default/files/datasheets/CYCOM_PR520_RTM_031912.pdf
- Song, S., Waas, A.M., Shahwan, K.W., Xiao, X., and Faruque, O. (2007) Braided textile composites under compressive loads: Modeling the response, strength and degradation. *Composites Science and Technology*, Vol. 67, pp. 3059–3070.
- Sun, C.T., and Chen, J.L. (1991) A micromechanical model for plastic behavior of fibrous composites. *Composites Science and Technology*, Vol. 40, pp. 115–129.
- Sun, C.T., and Potti, S.V. (1996) A simple model to predict residual velocities of thick composite laminates subjected to high velocity impact. *International Journal of Impact Engineering*, Vol. 18, pp. 339–353.
- Takano, N., Uetsuji, Y., Kashiwagi, Y., and Zako, M. (1999) Hierarchical modelling of textile composite materials and structures by the homogenization method. *Modelling and Simulation in Materials Science and Engineering*, Vol. 7, pp. 207–231.
- Talreja, R. (1985) A continuum mechanics characterization of damage in composite materials. *Proc. Royal Soc. London A*, 399(1817), pp. 195–216.
- Talreja, R. (2006) Multi-scale modeling in damage mechanics of composite materials. *Journal of Material Science*, Vol. 41, pp. 6800–6812.
- Tandon, G.P. (2011) Characterization of thermo-oxidation in laminated and textile composites. In: *Long-Term Durability of Polymeric Matrix Composites*, Pochiraju, K.V., Tandon, G.P., and Schoeppner, G.A., eds. Springer, New York.

Tao, W., Liu, Z., Zhu, P., Zhu, C., and Chen, W. (2017) Multi-scale design of three dimensional woven composite automobile fender using modified particle swarm optimization algorithm. *Composite Structures*, Vol. 181, pp. 73–83.

Toray Carbon Fibers America, Inc. Decatur, AL, USA.
http://www.toray.us/united_states/network/cfa/index.html

Tsai, S.W. (1965) Strength characteristics of composite materials. NASA CR-224. National Aeronautics and Space Administration, Washington, D.C.

Tsai, S.W. and Wu, E.M. (1971) A general theory of strength for anisotropic materials. *Journal of Composite Materials*, Vol. 5, pp. 58–80.

Williams, K.V. and Vaziri, R. (2001) Application of a damage mechanics model for predicting the impact response of composite materials. *Computers and Structures*, Vol. 79, pp. 997–1011.

Xiao, J.R., Gama, B.A., and Gillespie, Jr., J.W. (2007) Progressive damage and delamination in plain weave S-2 glass/SC-15 composites under quasi-static punch-shear loading. *Composite Structures*, Vol. 78, pp. 182–196.

Zhang, C., Binienda, W.K., Morscher, G.N., Martin, R.E., and Kohlman, L.W. (2013) Experimental and FEM study of thermal cycling induced microcracking in carbon/epoxy triaxial braided composites. *Composites: Part A*, Vol. 46, pp. 34–43.

Zhang, C. (2014) A meso-scale finite element model for simulating free-edge effect in carbon/epoxy textile composite. *Mechanics of Materials*, Vol. 76, 1-19.