NONLINEAR STRAIN RATE DEPENDENT COMPOSITE MODEL
FOR EXPLICIT FINITE ELEMENT ANALYSIS

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

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

Xiahua Zheng
May, 2006
NONLINEAR STRAIN RATE DEPENDENT COMPOSITE MODEL
FOR EXPLICIT FINITE ELEMENT ANALYSIS

XIAHUA ZHENG
Dissertation

Approved:  Accepted:

Advisor  Department Chair
Wieslaw K. Binienda  Wieslaw K. Binienda

Committee Member  Dean of the College
Atef F. Saleeb  George K. Haritos

Committee Member  Dean of the Graduate School
Kevin L. Kreider  George R. Newkome

Committee Member  Date
Pizhong Qiao

Committee Member
Ernian Pan

Committee Member
Robert K. Goldberg
ABSTRACT

With the increasing application of fiber reinforced polymer matrix composites in the aerospace industry, a composite model that has the capability to capture the nonlinear, strain rate dependent deformation behavior of the material is desired for finite element analysis.

Polymers are believed to be the major contributors to the nonlinearity and rate dependence of the composites. In this study, the nonlinear, rate dependent constitutive polymer model developed by Goldberg has been modified to incorporate the rate dependence of the elastic modulus and a simple damage model is proposed to improve its unloading prediction. The polymer constitutive equations are then implemented within a strength of material based micromechanics method in order to predict the nonlinear, strain rate dependent deformation of the composite. Strain rate dependent failure criteria and post failure progressive damage model are incorporated into the composite model.

The polymer and the composite models are implemented into a commercially available explicit finite element code, LS-DYNA, as user defined materials (UMATs). The deformation behaviors of several representative polymers and two polymer matrix composites of various fiber configurations are simulated in LS-DYNA with the UMATs for a wide range of strain rates, and the numerical results agree well with the experimental data. To expand the application of the unidirectional lamina based
composite UMAT, braiding/weaving with through-thickness integration points method is proposed with examples to simulate the deformation behavior of textile composites.
DEDICATION

To my daughter Vivian
on her first Birthday

My wish for her is to grow up healthy and happy
ACKNOWLEDGEMENTS

First, I would like to express my deep gratitude to my advisor Dr. Wieslaw K. Binienda for his generous support and encouragement for the past years. He has always provided me with the best opportunities and help whenever I needed it.

Secondly, I would like to thank Dr. Robert K. Goldberg for the valuable comments and suggestions he provided me throughout the course of the research. Dr. Gary D. Robert has also been very supportive and provided inspiration and direction in this study.

Acknowledgements are also given to my committee members, Dr. Atef F. Saleeb, Dr. Kevin L. Kreider, Dr. Pizhong Qiao and Dr. Ernian Pan, for evaluating my work and providing me with helpful recommendations.

Special thanks are given to Jingyun Cheng, Mijia Yang and Jialai Wang for their friendship and help.

Greatest thanks are given to my husband Han Xu for more love and support than I could ever expect. I feel so lucky I have met you and became your wife! I would also like to express my gratefulness to my parents and parents-in-law who are always ready to come to my side and share my burden.
TABLE OF CONTENTS

LIST OF TABLES...........................................................................................................xi
LIST OF FIGURES........................................................................................................xii

CHAPTER

I  INTRODUCTION........................................................................................................1
  1.1 Strain rate effect on composites .................................................................2
  1.2 Composite modeling.....................................................................................4
  1.3 Explicit Finite Element code.......................................................................5
  1.4 Objective of this study...................................................................................5
  1.5 Outline of the dissertation............................................................................6

II  NONLINEAR STRAIN RATE DEPENDENT POLYMER AND COMPOSITE MODELS........................................................................................................8
  2.1 Nonlinear, strain rate dependent polymer constitutive equations..............8
    2.1.1 Mechanical properties of polymers.......................................................8
    2.1.2 Polymers as composite matrix...............................................................11
    2.1.3 Background on polymer modeling.......................................................12
    2.1.4 Original Goldberg’s polymer constitutive equations............................14
    2.1.5 Material constants determination.......................................................17
    2.1.6 Parametric study on the polymer model.............................................20
2.2 Modifications on the Goldberg’s polymer model

2.2.1 Strain rate dependent initial elastic modulus

2.2.2 Evolution of $\alpha$

2.2.3 Polymer damage model to improve unloading behavior

2.3 Composite micromechanical model and damage model

2.3.1 Composite micromechanical model

2.3.2 Discussion on the composite unit cell selection

2.3.3 Rate-sensitive fibers

2.3.4 Composite rate-dependent failure criteria

2.3.5 Composite progressive post failure model

III EXPLICIT FINITE ELEMENT METHOD IN LS-DYNA

3.1 LS-DYNA overview

3.2 Explicit Finite Element Method

3.2.1 Lagrangian formulation

3.2.2 Governing equations

3.2.3 Spatial discretization

3.2.4 Lumped mass matrix

3.2.5 Central difference method

3.3 Implicit method vs. explicit method

3.4 Time step control

3.5 Belytschko-Tsay shell formulation

3.5.1 Co-rotational coordinates
5.2.1 Off-axis IM7/977-2 predictions ................................................................. 119
5.2.2 Failure envelope prediction for IM7/977-2 .............................................. 122
5.2.3 Angle-ply IM7/977-2 predictions .............................................................. 123
5.2.4 Off-axis AS4/PEEK predictions ................................................................. 126
5.2.5 Angle-ply AS4/PEEK predictions .............................................................. 130
5.2.6 Example for progressive composite post failure model......................... 131

5.3 Braiding/weaving with through-thickness integration points method............ 132
5.3.1 Unit cell dimensions ............................................................................... 136
5.3.2 Integration rules ...................................................................................... 138
5.3.3 Material type assignment ........................................................................ 141
5.3.4 Material constants determination ............................................................. 141
5.3.5 Coupon tests simulation ............................................................................ 145

VI CONCLUSIONS ......................................................................................... 148
REFERENCES .................................................................................................. 151
APPENDIX ....................................................................................................... 158
LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Material constants for polymer modeling</td>
<td>105</td>
</tr>
<tr>
<td>2</td>
<td>Mechanical properties for fibers</td>
<td>119</td>
</tr>
<tr>
<td>3</td>
<td>Fiber volume ratios for thickness slices in $0^\circ \pm 60^\circ$ system</td>
<td>138</td>
</tr>
<tr>
<td>4</td>
<td>Fiber volume ratios for thickness slices in $0^\circ \pm 45^\circ$ system</td>
<td>138</td>
</tr>
<tr>
<td>5</td>
<td>Integration rules for the $0^\circ \pm 60^\circ$ and $0^\circ \pm 45^\circ$ systems</td>
<td>139</td>
</tr>
<tr>
<td>6</td>
<td>Material properties for T700 carbon fiber</td>
<td>142</td>
</tr>
<tr>
<td>7</td>
<td>Material properties for Hexply M36 epoxy</td>
<td>143</td>
</tr>
<tr>
<td>8</td>
<td>Material constants in Material 54 for Sub-cells A &amp; C</td>
<td>144</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Typical stress-strain curves of polymers under uniaxial tension</td>
</tr>
<tr>
<td>2.2</td>
<td>Schematic demonstration of polymer stress-strain curves under different testing temperatures</td>
</tr>
<tr>
<td>2.3</td>
<td>Schematic demonstration of strain rate effects on polymer stress-strain curves</td>
</tr>
<tr>
<td>2.4</td>
<td>Determination of $n$ and $Z_1$</td>
</tr>
<tr>
<td>2.5</td>
<td>Parametric studies with $Z_0$ and $Z1$</td>
</tr>
<tr>
<td>2.6</td>
<td>Parametric studies with $\alpha_0$ and $\alpha_1$</td>
</tr>
<tr>
<td>2.7</td>
<td>Hydrostatic stress effects</td>
</tr>
<tr>
<td>2.8</td>
<td>Uniaxial loading-unloading curves by Xia et al.</td>
</tr>
<tr>
<td>2.9</td>
<td>Determination of effective elastic modulus</td>
</tr>
<tr>
<td>2.10</td>
<td>Determination of polymer damage constant</td>
</tr>
<tr>
<td>2.11</td>
<td>Unit cell and slices in the micromechanical model</td>
</tr>
<tr>
<td>2.12</td>
<td>Square and hexagonal-packed array</td>
</tr>
<tr>
<td>2.13</td>
<td>Unit cells for square and hexagonally packed arrays</td>
</tr>
<tr>
<td>2.14</td>
<td>Effect of $m$ on the damage growth in the MLT model</td>
</tr>
<tr>
<td>2.15</td>
<td>Effect of $m$ on the damage growth in Yen’s model</td>
</tr>
<tr>
<td>3.1</td>
<td>Lagrangian and Eulerian mesh</td>
</tr>
</tbody>
</table>
5.2 Experimental and LS-DYNA predicted shear stress-shear strain curves for PR520 resin at strain rates of 420/s, 1.76/s and $7.0 \times 10^{-5}$/s .................................................................107

5.3 Experimental and LS-DYNA predicted tensile stress-strain curves for PR520 resin at strain rates of 510/s, 1.4/s and $5.0 \times 10^{-5}$/s ......................108

5.4 Experimental and LS-DYNA predicted shear stress-shear strain curves for 977-2 resin at strain rates of 518/s, 1.91/s and $9.0 \times 10^{-5}$/s .........................109

5.5 Experimental and LS-DYNA predicted tensile stress-strain curves for 977-2 resin at strain rates of 365/s, 1.31/s and $5.7 \times 10^{-5}$/s .........................110

5.6 Strain rate effect on polymer deformation behavior ........................................111

5.7 Experimental and LS-DYNA predicted shear stress-strain curves for E-862 resin at strain rates of 404/s, 1.5/s and $7.5 \times 10^{-5}$/s .........................112

5.8 Experimental and LS-DYNA predicted tensile stress-strain curves for E-862 resin at strain rates of 455/s, 1.4/s and $5.7 \times 10^{-5}$/s .........................113

5.9 Experimental and LS-DYNA predicted compressive stress-strain curves for PEEK resin at strain rates of 0.1/s, 0.01/s, $1 \times 10^{-3}$/s and $1 \times 10^{-5}$/s ...............115

5.10 Experimental and LS-DYNA predicted tensile stress-strain curves for PEEK resin at strain rates of $1 \times 10^{-3}$/s, $1 \times 10^{-4}$/s and $1 \times 10^{-5}$/s ..................116

5.11 Hydrostatic stress effects on PEEK ............................................................117

5.12 Comparison of the unloading behavior between the original Goldberg’s polymer model and the modified model with damage ........................................118

5.13 Experimental and LS-DYNA predicted shear stress–strain curves for IM7/977-2 [10°] at strain rates of 320/s and 0.56/s .........................................121

5.14 Experimental and LS-DYNA predicted shear stress–strain curves for IM7/977-2 [45°] at strain rates of 405/s and 1.2/s .........................................121

5.15 Experimental and LS-DYNA predicted shear stress–strain curves for IM7/977-2 [90°] at strain rates of 405/s and 1.09/s .........................................122

5.16 Model predictions of the failure envelops for the unidirectional IM7/977-2 composite at various strain rates ..................................................123
5.17 Experimental and LS-DYNA predicted stress-strain curves for IM7/977-2 $[\pm 45^\circ]_s$ at strain rates of 604/s, 2.1/s and $9 \times 10^{-5}$/s ..........................125

5.18 Experimental and LS-DYNA predicted stress-strain curves for off-axis AS4/PEEK at strain rate of $1 \times 10^{-5}$/s .................................................127

5.19 Experimental and LS-DYNA predicted stress-strain curves for off-axis AS4/PEEK at strain rate of 0.1/s ..................................................128

5.20 Experimental and LS-DYNA predicted stress-strain curves for AS4/PEEK $[0^\circ]$ at strain rate of $1 \times 10^{-5}$/s, $1 \times 10^{-3}$/s and 0.1/s .........................129

5.21 Experimental and LS-DYNA predicted stress-strain curves for AS4/PEEK $[\pm \theta^\circ]_s$ at strain rate of 0.01/s .............................................130

5.22 Experimental and LS-DYNA predicted stress-strain curves for AS4/PEEK $[\pm 15^\circ]_s$ at strain rate of 0.1/s .............................................131

5.23 Effect of damage exponent $m$ on the post failure prediction ..................132

5.24 Unit cell for braided composites ..................................................134

5.25 Braided fiber preform .................................................................135

5.26 Unit cell for $0^\circ \pm 60^\circ$ braided composite system ..............................137

5.27 Unit cell for $0^\circ \pm 45^\circ$ braided composite system ..............................137

5.28 Simulated tensile and compressive stress-strain curves of M36 epoxy .......143

5.29 Coupon tensile tests in axial direction .............................................146

5.30 Coupon tensile tests in transverse direction ......................................147

5.31 Experimental and predicted coupon tensile tests ...............................147
CHAPTER I

INTRODUCTION

Due to their weight saving advantage, fiber-reinforced polymer matrix composites are increasingly being applied to structural components in the aerospace industry. In their applications to some primary aircraft structures, such as the radome and the fan containment system of jet engines, the mechanical properties of these materials under high strain rate impact conditions have attracted more attention in the composites community.

Unlike metals whose mechanical behavior has been intensively studied and well understood, fiber reinforced composites often lack detailed dynamic mechanical properties. Experiments on composite structures could be expensive and difficult to set up due to the complex composite configurations and interactions between the fiber and matrix. As an alternative approach, finite element simulations have become an important tool in assisting the composite structural design. Then a reliable composite model which
has the capability of properly capturing the nonlinearity and rate sensitivity is required for high strain rate simulations with finite element software.

In this Chapter, background on the strain rate dependent composite modeling and the description of the remaining chapters will be presented.

1.1 Strain rate effect on composites

In aerospace applications, the composite would be loaded at strain rates up to several hundred per second. These strain rates can be considered as high strain rates. In automobile crashworthiness analysis, the strain rates studied are usually in the magnitude of 1 to 10 per second which is often referred to as moderate strain rates. In the quasi-static experiments, the strain rates around $1 \times 10^{-5}$ per second are categorized as low strain rates.

The experimental techniques employed for determining the behavior of composite materials at high strain rate under various loading conditions were reviewed by Hamouda and Hashmi. Each technique has its applicable strain rate range. For example, conventional testing machine is used for strain rate lower than 0.1/s; for strain rate of 0.1-100/s, there is servo-hydraulic machine; for high strain rate at about 100-10^4/s, Split Hopkinson Pressure Bar (SHPB) is widely used by researchers; for even higher strain rate, gas gun apparatus for direct impact is employed. Different techniques also require different specimen geometry and have their own limitations and advantages that drive researchers searching for modifications to make the technique best serve their need.
The deformation of polymer composites is often assumed to be linear elastic and independent of strain rate in transient dynamic finite element codes used for impact analysis. However, many researchers have observed through their experiments that the modulus and strength of the composites increase with increasing strain rate.

There are several factors affect the rate-dependence of composites such as their configurations and loading conditions, but first of all it depends on the rate dependence of each constituent. As for the fiber phase in the composite, the rate sensitivity depends on the type of the fiber. Of the two most commonly used fibers, carbon fiber is known to be rate insensitive; while the glass fiber is rate sensitive.

Polymers are known to have a strain rate dependent deformation response that is nonlinear above about one or two percent strain. Traditionally, viscoelasticity models have been used to describe the polymer behavior. However, due to the increasing mechanical expectation in practical industry, more sophisticated constitutive laws are required to describe the material deformation response. Despite the differences in the microstructure, similarities in the deformation behavior have been observed between metals and polymers. It raised substantial interest in the research community in adapting well-established plastic and viscoplastic constitutive equations for metals to model the nonlinear, strain rate sensitive deformation of polymers and polymer matrix composites. Through the continuous effort in NASA Glenn Research Center, Goldberg has modified the Bodner-Partom viscoplastic state variable model, which was originally developed for metals, to analyze the nonlinear, strain rate dependent deformation behavior of the polymer matrix materials. The effects of hydrostatic stresses
on the nonlinear response, which unlike in metals are significant for polymers, were properly accounted for in his model. The polymer constitutive equations were incorporated into the composite micromechanical model based on mechanics of materials. In this study, Goldberg’s polymer and composite model will be modified and implemented into a finite element software as user defined material models.

1.2 Composite modeling

With the information provided by the experimental tests, a variety of methods have been developed to model the rate dependent response of fiber reinforced polymer matrix composites. Basically the methods used fall in two categories: macromechanical approaches and micromechanical approaches.

In the macromechanical approach, the composite material is modeled as an anisotropic, homogeneous material, without paying any attention to the individual constituents. For example, Sun and coworkers\textsuperscript{13,14} developed a macromechanical, transversely isotropic viscoplasticity models to analyze the nonlinear deformation of polymer composites. The advantage of this approach is its simplicity: it is easier to implement and computational time efficiency.

In micromechanics, the effective properties and response of composites are computed based on the properties and response of the individual constituents. A representative volume element (RVE) or unit cell composed of fiber and matrix is chosen as representative of the composite as a whole. Then the analyzing of the composite is carried out on the unit cell\textsuperscript{15,16,17}. Though more complicated to implement, this approach
gives more realistic representation of the composite. It has more flexibility in modeling the deformation and damage of the composite. In this study, the micromechanical approach is used for the modeling of composites.

1.3 Explicit Finite Element code

Commercial finite element software has become an indispensable tool in engineering design and analysis, such as ANSYS, ABAQUS and LS-DYNA. Due to its explicit time integration methodology, LS-DYNA\textsuperscript{18} excels in analyzing large deformation dynamic response of structures. It is rapidly gaining favor in aerospace and automobile industries.

LS-DYNA has a large library of material options which have been widely used in the automobile and aerospace industries. However, the material models for composite materials currently included within LS-DYNA are elastic before failure and do not consider the effect of strain rate on the material response, which is an important feature in polymer matrix composites, especially under high strain rate impact. Recently, Yen\textsuperscript{19} implemented a new composite failure model into LS-DYNA. This model uses scale factors to account for the effect of strain rate on the initial elastic moduli and the layer strength values. However, the model is only applicable to solid elements and the deformation behavior is simply linear elastic..

1.4 Objective of this study

The objective of this study is to adopt for finite element calculation a nonlinear, strain rate dependent composite model that can be used to adequately capture the
nonlinear, rate-dependent deformation and damage behavior of composite structures under high strain rate loadings.

In this study, the nonlinear, rate dependent polymer and composite micromechanical model developed by Goldberg\textsuperscript{11,12} will be modified and implemented into LS-DYNA as user defined material models (UMATs). The composite UMAT will also incorporate rate-dependent failure criteria and progressive damage. The composite model will be representing for a single unidirectional composite layer. By defining the through-thickness integration points, the composite UMAT can simulate the multilayered, woven or braided composites. The polymer and composite UMATs will then be verified with experimental tests on various polymer and composite types.

1.5 Outline of the dissertation

In Chapter II, background on the mechanical properties of polymers, polymer types as composite matrix and polymer modeling will be reviewed first. Then the original Goldberg’s nonlinear, rate-dependent polymer constitutive equations will be introduced with parametric studies. Next, the modifications and improvements on the polymer model will be explained in detail. Finally, after a brief description on the micromechanical composite model developed by Goldberg with discussions and modifications, rate dependent failure criteria and progressive damage model added to the composite model will be presented.

An overview of LS-DYNA will be given in Chapter III. Several aspects in its explicit finite element methodology will be studied, such as its shell formulation and time
step calculation. The procedure in developing a stand alone explicit finite element code will be described consequently.

In Chapter IV, the theory and applications of currently available composite models in LS-DYNA will be examined and compared with examples. By pointing out the inadequacy in capturing the nonlinear, strain rate dependent deformation behavior of composites by these models, the necessitation for a new composite model will be justified. Then the procedure in implementing user defined material models (UMATs) will be explained in detail.

The implemented polymer and composite UMATs in LS-DYNA will be examined and verified with experimental results in Chapter V. Correlation studies on the polymer UMAT will be conducted with several types of polymers, including PR520 and 977-2 toughened epoxies, one standard epoxy E862 and a thermoplastic PEEK. Verification of the composite UMAT will be carried out with off-axis and angle plies of two types of carbon fiber reinforced polymer matrix composites, namely IM7/977-2 and AS4/PEEK. In ordered to simulate the braided or woven composites with the unidirectional composite UMAT, braiding/weaving with through-thickness integration points method will be proposed with examples at the end of this chapter.

Finally, the study will be summarized in Chapter VI and future work will also be discussed.
2.1 Nonlinear, strain rate dependent polymer constitutive equations

2.1.1 Mechanical properties of polymers

The deformation behavior of polymer material can take several different forms. One typical response of polymers under uniaxial tension is shown in Figure 2.1a. After elastic deformation in region R1, the material yields and necking occurs. With the expansion of necking, the strain increases while the stress keeps constant (region R2). Then work hardening commences in region R3 till failure occurs. However, not all polymers undergo the elongation and necking process as in Figure 2.1a. Another type of polymer response is shown in Figure 2.1b, in which the stress never decreases and the specimen fractures before the material yields.
Several important factors control the polymer’s mechanical behavior. The first one is the testing temperature $T$ relative to the material’s glass transformation temperature $T_g$. $T_g$ is an important characteristic property of polymers. Under the same testing temperature, polymers with higher $T_g$ tend to behave brittle (curve A in Figure 2.2), while those with lower $T_g$ behave ductile (curve D in Figure 2.2). In other words, if the testing temperature $T$ is much lower than the material’s $T_g$ (curve A in Figure 2.2), the specimen fractures before yielding occurs. Increasing testing temperature $T$, the stress reaches a maximum, the material yields, and then the stress goes down till fracture (curve B in Figure 2.2). If $T$ is close to $T_g$ (curve C in Figure 2.2), after yielding occurs, the strain keeps increasing without the increase in stress, then work hardening occurs till material fails. This is similar to the typical curve as shown in Figure 2.1a. When testing temperature $T$ is above $T_g$ (curve D in Figure 2.2), the material does not yield. The strain increases dramatically with a slight increase of stress, showing a long platform in the stress-strain curve. Then the curve goes up rapidly till fracture.
The second factor is the strain rate. Many experimental works have shown that strain rate has great effect on the polymer’s deformation behavior\textsuperscript{20,21,22,23}. With the increasing of the strain rate, the modulus and the yielding stress of polymers increase, while the ultimate strain decreases, as being illustrated in Figure 2.3. The polymer’s ductility decreases and behaves more brittle. In a tensile test, increasing the strain rate has the similar effect as decreasing the temperature. In addition, tests under high strain rate may also cause a temperature increase in the specimen which consequently changes the polymer’s stress-strain curves. Arruda et al.\textsuperscript{25} studied the coupling of strain rate and temperature, showing a thermal softening of the material at large strains.
Another important factor influencing the polymer’s deformation behavior is the hydrostatic pressure. In the classical plasticity theory, it is generally assumed that hydrostatic pressure has no effect on the plastic deformation of material. However, polymers, as well as soils, concretes and rocks, are found to be pressure sensitive\textsuperscript{25,26,27}. Polymers tend to have higher initial modulus and yield stress with higher hydrostatic pressure. One of the examples is that the yield stress in compression is higher than the yield stress in tension.

2.1.2 Polymers as composite matrix

Although in general the term of composite material could apply to any combination of individual materials or phases, it most often refers to a material made of thin but strong fibers which have been impregnated with a polymer matrix. The matrix
material serves several functions:\textsuperscript{28} it holds the fibers together to form a solid part; it protects the fibers and transmits the stress from one fiber to another; in unidirectional composites, the matrix properties control the transverse properties.

Among the matrix materials, polyesters are economic, easy to handle and mold and have good mechanical properties. They are most commonly used with fiberglass. When higher strength or higher chemical resistance is required, epoxy resins are used. They are almost exclusively employed for carbon, boron and Kevlar fibers. The thermosetting epoxy has high glass transition temperature which gives it high strength and high modulus as demonstrated by Curve A in Figure 2.2. However, its brittleness is undesirable for the application of impact resistance. To improve its ductility and extend its failure strain, many researches have been carried out to add thermoplastics into the thermosetting epoxy. These modified epoxies are called toughened epoxies. Poly ether ether ketone (PEEK) is a thermoplastic used as matrix material which has received considerable interest. The use of thermoplastics allows molding techniques which are unavailable for the other thermosetting polymers.

2.1.3 Background on polymer modeling

As has been stated in Section 2.1.1, the deformation response of polymers is nonlinear, strain rate dependent and hydrostatic pressure dependent. Traditionally, linear viscoelasticity models, such as the Kelvin or Voigt model and the Maxwell model, have been used to predict the response of polymers.\textsuperscript{8,26} These models utilize combinations of Hookean springs and Newtonian dashpots to idealize the strain rate dependent behavior
of materials. Then the linear theories are adapted to the nonlinear conditions. For example, in a model developed by Cessna and Sternstein, nonlinear dashpots were incorporated into the constitutive equations\textsuperscript{29}.

A more sophisticated technique in polymer constitutive modeling utilizes a molecular approach by adapting the Eyring equation\textsuperscript{30}. In this method, the deformation of polymers is assumed to be a thermally activated rate process involving the motion of molecular chains over potential energy barriers. Another approach\textsuperscript{31} to predict the polymer deformation is based on the flow-theories of Argon\textsuperscript{32}, which assumes that the deformation is due to the unwinding of molecular kinks. In both of these two approaches, state variables are used to represent the resistance to molecular flow caused by a variety of mechanisms. The state variable values evolve with stress, inelastic strain and inelastic strain rate.

An alternative approach has adapted the well-established plastic and viscoplastic constitutive equations for metals to model the nonlinear, strain rate sensitive deformation of polymers and polymer matrix composites\textsuperscript{9,10,13,33,34}. This method is based on the fact that despite the differences in the micro-structure, similarities in the phenomenological deformation behavior have been observed between metals and polymers\textsuperscript{35}. Recently, Goldberg et al. have modified the Bodner-Partom viscoplastic state variable model\textsuperscript{36}, which was originally developed for metals, to analyze the nonlinear, strain rate dependent deformation of the polymer matrix materials\textsuperscript{12}. The effects of hydrostatic stresses on the nonlinear response, which unlike in metals are significant for polymers, are accounted for in this model.
In state variable models, a single unified strain variable is defined to represent all inelastic strains\textsuperscript{37}. Furthermore, in the state variable approach there is no defined yield stress. Inelastic strains are assumed to be present at all values of stress. The inelastic strains are just assumed to be very small in the “elastic” range of deformation. State variables, which evolve with stresses and inelastic strains, are defined to represent the average effects of the deformation mechanisms.

As will be explained in the following sections, Goldberg’s polymer state variable model will be modified and utilized in this study to predict the polymer’s nonlinear, strain–rate dependent deformation behavior. Then the polymer model is implemented into the composite micromechanical model to account for the composite’s nonlinear, strain rate dependent behavior.

2.1.4 Original Goldberg’s polymer constitutive equations

In Goldberg’s model, the polymer is assumed to be isotropic, even though some degree of material anisotropy may be developed due to preferred orientations of molecular chains at finite deformations. Temperature and moisture effects are not considered, as only room temperature data are currently available. The nonlinear strain recovery observed in polymers during unloading is not simulated, and small strain theory is assumed to apply. Phenomena such as creep, relaxation and high cycle fatigue are not accounted for within the equations.

The isotropic compliance matrix is used to relate the strains to the stresses using the following equation from which the stress tensor $\sigma_{ij}$ can be calculated:
where $\varepsilon_{ij}^E$ is the elastic strain tensor, $\varepsilon_{ij}^I$ is the inelastic strain tensor, and $\varepsilon_{ij}$ is the total strain tensor which equals to the summation of $\varepsilon_{ij}^E$ and $\varepsilon_{ij}^I$. $E_m$ is the Young’s modulus and $\nu_m$ is the Poisson’s ratio.

From $\sigma_{ij}$, the deviatoric stress components $S_{ij}$ can be directly derived. The components of the inelastic strain rate tensor $\dot{\varepsilon}_{ij}^I$, as shown in Equation 2.2, are defined as a function of $S_{ij}$, the second invariant of the deviatoric stress tensor $J_2$ and an isotropic state variable $Z$, which represents the resistance to molecular flow.

$$\dot{\varepsilon}_{ij}^I = 2D_0 \exp \left[ -\frac{1}{2} \left( \frac{Z}{\sigma_e} \right)^{2n} \right] \left( \frac{S_{ij}}{\sqrt{J_2}} + \alpha \delta_{ij} \right)$$  \hspace{1cm} (2.2)

where $D_0$ and $n$ are both material constants, with $D_0$ representing the maximum inelastic strain rate and $n$ controlling the rate dependence of the material. The effective stress $\sigma_e$ is defined as

$$\sigma_e = \sqrt{3J_2} + \sqrt{3}\alpha \sigma_{kk}$$  \hspace{1cm} (2.3)
where $\alpha$ is a state variable controlling the level of the hydrostatic stress effects and $\sigma_{kk}$ is the summation of the normal stress components which equals three times the mean stress.

The evolution equations for the two state variables $Z$ and $\alpha$ can then be computed using the following equations:

$$
\dot{Z} = q(Z_1 - Z)\dot{e}_e^I \\
\dot{\alpha} = q(\alpha_1 - \alpha)\dot{e}_e^I
$$

(2.4)

(2.5)

where $q$ is a material constant representing the “hardening” rate, and $Z_1$ and $\alpha_1$ are material constants representing the maximum value of $Z$ and $\alpha$, respectively. The initial values of $Z$ and $\alpha$ are defined by the material constants $Z_0$ and $\alpha_0$. The term $\dot{e}_e^I$ in Equation 2.4 and Equation 2.5 represents the effective deviatoric inelastic strain rate, which is defined as

$$
\dot{e}_e^I = \sqrt{\frac{2}{3}}\dot{e}_y^I\dot{e}_y^I \\
\dot{\varepsilon}_y^I = \dot{\varepsilon}_y^I - \dot{\varepsilon}_m^I\dot{\delta}_y
$$

(2.6)

where $\dot{\varepsilon}_m^I$ is the mean inelastic strain rate, which is equal to $(\dot{\varepsilon}_y^I + \dot{\varepsilon}_{2z}^I + \dot{\varepsilon}_{3z}^I)/3$. In many state variable constitutive models developed to analyze the behavior of metals\textsuperscript{37}, the total inelastic strain and strain rate are used in the evolution laws and are assumed to be equal to their deviatoric values. As discussed by Li and Pan\textsuperscript{38}, since hydrostatic stresses contribute to the inelastic strains in polymers, indicating volumetric effects are present, the mean inelastic strain rate is accounted for in Equation 2.6, unlike in the inelastic analysis of metals.
2.1.5 Material constants determination

The material constants in the above state variable constitutive equations that need to be determined include \( D_0, n, Z_o, Z_i, \alpha_o, \alpha_i \) and \( q \). The procedure in determining those parameters has been described by Goldberg et al\(^{12} \). More details on the general approach can be found in Stouffer and Dame\(^{37} \) and Bodner\(^{36} \).

It is often necessary to determine the inelastic strain rate as part of the process to find the material constants. The inelastic strain can be determined from Equation 2.1 using the stress and strain histories, elastic modulus and Poisson’s ratio. The inelastic strain rate is then determined from the slope of a sliding spline fit of several time-inelastic strain pairs.

The parameter \( D_0 \) is correlated with the maximum inelastic strain rate. Typically, it is assumed to be equal to a value of \( 10^4 \) times the maximum expected strain rate. For example, \( D_0 = 10^6 \text{ sec}^{-1} \) or greater for high rate loading and wave propagation applications.

To determine the values of \( n \) and \( Z_i \), Equation 2.2 is simplified to the case of pure shear loading, so that the hydrostatic stress constant \( \alpha \) does not go into the picture:

\[
\frac{\dot{\gamma}^i}{2} = D_0 \exp\left[ -\frac{1}{2} \left( \frac{Z}{\sqrt{3} |\tau|} \right)^{2n} \right] \frac{\tau}{|\tau|} \tag{2.7}
\]

where \( \dot{\gamma}^i \) is the engineering inelastic shear strain rate, \( \tau \) is the shear stress. Rearrange Equation 2.7 as follows:

\[
-2 \ln \left( \frac{\dot{\gamma}^i}{2D_0} \right) = \left( \frac{Z}{\sqrt{3} |\tau|} \right)^{2n} \tag{2.8}
\]
Then the natural logarithm of both sides of the resulting expression is taken. The values of the inelastic shear strain rate, shear stress, and state variable \( Z \) at “saturation”, which is the point where the stress-strain curve flattens out and becomes horizontal, are substituted into Equation 2.8, leading to the following

\[
\ln\left[-2 \ln\left(\frac{\dot{\gamma}_0}{2D_0}\right)\right] = 2n \ln(Z_1) - 2n \ln(\sqrt{3}\tau_s) \tag{2.9}
\]

where \( \tau_s \) equals the saturation shear stress, \( \dot{\gamma}_0 \) is the total engineering shear strain rate in a constant strain rate shear test, which is assumed to equal the inelastic strain rate at “saturation”. With a set of shear stress-shear strain curves obtained from different constant strain rate tests, data pairs of the total strain rate \( \dot{\gamma}_0 \) and the saturation shear stress \( \tau_s \) are taken. For each strain rate, the data values are substituted into Equation (2.9), and represent a point on a master curve. A least squares regression analysis is then performed on the master curve. As suggested by Equation 2.9 and shown in Figure 2.4, the slope of the best-fit line is equal to \(-2n\). The intercept of the best-fit line is equal to \(2n \ln(Z_1)\).

To determine the value of \( Z_0 \), Equation 2.8 is rearranged as follows

\[
Z = \left[-2 \ln\left(\frac{\dot{\gamma}^\prime}{2D_0}\right)\right]^{\frac{1}{2n}} \sqrt{3}\tau
\tag{2.10}
\]

As the initial value of \( Z \), \( Z_0 \) corresponds to the value at the point where the stress-strain curve becomes nonlinear, which is the approximate point where the curve appreciably deviates from a linear extrapolation of the initial data. Therefore the values of \( \tau \) and \( \dot{\gamma}^\prime \)
at the “starting” point of nonlinearity are substituted into Equation 2.10 to yield $Z_0$. The inelastic shear strain rate $\dot{\gamma}'$ is approximated with the shear strain rate used in the constant strain rate test divided by 100.

$$\ln \left[ -2 \ln \left( \frac{\dot{\gamma}_0}{2D_0} \right) \right]$$

![Figure 2.4 Determination of $n$ and $Z_1$](image)

To determine the value of $q$, Equation 2.4 is integrated for the case of pure shear loading, leading to the following expression:

$$Z = Z_1 - (Z_1 - Z_0) \exp \left( \frac{-q}{\sqrt{3}} \dot{\gamma}' \right)$$  \hspace{1cm} (2.11)

where $\dot{\gamma}'$ is the inelastic shear strain. At saturation, the value of the internal stress $Z$ is assumed to approach $Z_1$, resulting in the exponential term approaching zero. Assuming that saturation occurs when the following condition is satisfied

$$\exp \left( \frac{-q}{\sqrt{3}} \frac{\dot{\gamma}_s'}{\sqrt{3}} \right) = 0.01$$  \hspace{1cm} (2.12)

The equation is solved for $q$, where $\dot{\gamma}_s'$ is the inelastic shear strain at saturation.
To obtain the values of $\alpha_1$ and $\alpha_0$, Equation 2.3 is used in combination with stress-strain data from constant strain rate uniaxial tensile tests and constant strain rate shear tests. It is assumed that at a particular strain rate the effective stress at saturation under uniaxial tensile loading is equal to the effective stress at saturation under pure shear loading. Assuming the value of $\alpha$ at saturation is equal to $\alpha_1$ yields

$$
\sigma_s(1 + \sqrt{3}\alpha_1) = \sqrt{3}\tau_s \tag{2.13}
$$

where $\sigma_s$ and $\tau_s$ are the tensile and shear stresses at saturation, respectively. Similarly, assuming the value of $\alpha$ at the point the stress-strain curve becomes nonlinear is equal to $\alpha_0$ gives

$$
\sigma_{nl}(1 + \sqrt{3}\alpha_0) = \sqrt{3}\tau_{nl} \tag{2.14}
$$

where $\sigma_{nl}$ and $\tau_{nl}$ are the tensile and shear stresses at the point where the respective stress-strain curves become nonlinear. Then the values of $\alpha_1$ and $\alpha_0$ can then be determined from these equations.

2.1.6 Parametric study on the polymer model

To better illustrate the functions of the material constants in the polymer constitutive equations, parametric studies were carried out in this section. The toughened epoxy 977-2 was examined with the implemented polymer constitutive model in LS-DYNA, as will be explained in the following chapters. Details such as the material constants and single test element used in the simulation can be found in Chapter V. The influences of the material constants $Z_0$, $Z_1$, $\alpha_0$, and $\alpha_1$ on the tensile stress-tensile strain
curve were examined respectively by varying the targeting constant and keeping the remaining ones unchanged. A constant strain rate of 365/s was used.

It can be seen from Figure 2.5 that, as the initial value of Z, $Z_0$ influences the starting point of nonlinearity, with a larger $Z_0$ resulting in a stiffer behavior. However, $Z_0$ doesn’t affect the maximum stress. As the maximum value of Z, $Z_1$ controls the maximum stress in such a way that a larger $Z_1$ yields a higher maximum stress.

The effects of $\alpha_0$ and $\alpha_1$ representing respectively the initial and maximum values of hydrostatic stress constant $\alpha$ are plotted in Figure 2.6. Similar to $Z_0$ and $Z_1$, $\alpha_0$ influences the starting point of nonlinearity and $\alpha_1$ affects the maximum stress. For the tensile tests, a larger $\alpha_0$ gives a lower stress/strain curve in the nonlinear part, and a larger $\alpha_1$ shows smaller maximum stress. However, for compressive loading, the trend will be opposite. Though the effects of $\alpha_0$ and $\alpha_1$ are similar to those of $Z_0$ and $Z_1$, they are in a much milder magnitude.
Figure 2.5 Parametric studies with $Z_0$ and $Z_1$
Figure 2.6 Parametric studies with $\alpha_0$ and $\alpha_1$. 
To better demonstrate the effect and importance of the state variable $\alpha$, compressive and tensile stress-strain curves are plotted in Figure 2.7. The curves of compression were inversed and superimposed onto those of tension. It can be seen from the figure that, with the application of $\alpha$, the stress is significantly larger in compressive loading than in tensile loading, which is consistent with experimental observations. Without $\alpha$ modification, the compressive and tensile response of the polymer can not be differentiated; indicating the hydrostatic stress effect which is typical for polymers is not accounted. This effect will be further examined with experimental results in Chapter V.

![Figure 2.7 Hydrostatic stress effects](image-url)
2.2 Modifications on the Goldberg’s polymer model

To better account for the strain rate dependence and hydrostatic stress effect, several opportunities are addressed in this section to improve the original Goldberg’s polymer model. The modifications include the strain rate dependent initial elastic modulus, the evolution law of $\alpha$ and the application of damage model to improve the unloading behavior which has been found of great importance in an impact test.

2.2.1 Strain rate dependent initial elastic modulus

In Goldberg’s polymer model, the elastic modulus is considered as a material constant that is rate independent. However, the elastic modulus has been found to increase with increasing strain rate in both polymers and polymer matrix composites. Comparing the dynamic and static value of the initial modulus, Tay et al.\textsuperscript{22} found a two to three folds increase of the initial modulus of epoxy and an approximately three to five times increase in the initial modulus of glass fiber reinforced epoxy. Hsiao and Daniel\textsuperscript{3} observed an increase in the dynamic modulus and strength over the static values in a [90º] unidirectional carbon/epoxy laminate, primarily due to the rate dependence of the matrix. To account for this phenomenon, the constitutive equations need to be modified in order to include a strain rate dependent elastic modulus instead of a constant modulus.

In their study of gelatin gum gels, Teratsubo et al.\textsuperscript{21} proposed a linear relationship between the natural logarithms of modulus and the inverse of strain rate. Albérola et al.\textsuperscript{39} observed a strain rate threshold at 1 s$^{-1}$ for amorphous PEEK. For strain rates lower than 1 s$^{-1}$, the modulus is almost constant; while for strain rate higher than 1 s$^{-1}$, the modulus
tends to markedly increase. In a recently developed model, Yen\textsuperscript{19} utilized the Johnson-Cook model\textsuperscript{40} commonly used to describe high strain rate behavior in metals to account for the effect of strain rate on the elastic moduli and strength values of a composite layer. Similarly, the revised equation used in this study to compute the effect of strain rate on the elastic modulus of polymer is as follows:

\[
E = E_0 \left( 1 + C \ln \frac{\dot{\varepsilon}}{\dot{\varepsilon}_0} \right)
\]  

(2.15)

where \( C \) is a scaling material constant, \( E \) is the current elastic modulus, \( E_0 \) is the reference elastic modulus, \( \dot{\varepsilon}_0 \) is the reference effective strain rate and \( \dot{\varepsilon} \) is the applied effective strain rate. The effective strain rate is defined as:

\[
\dot{\varepsilon} = \sqrt[3]{\frac{2}{3} \left[ (\dot{\varepsilon}_{11} - \dot{\varepsilon}_m)^2 + (\dot{\varepsilon}_{22} - \dot{\varepsilon}_m)^2 + (\dot{\varepsilon}_{33} - \dot{\varepsilon}_m)^2 + 2\dot{\varepsilon}_{12}^2 + 2\dot{\varepsilon}_{23}^2 + 2\dot{\varepsilon}_{13}^2 \right]}
\]

(2.16)

where, \( \dot{\varepsilon}_m = \frac{1}{3}(\dot{\varepsilon}_{11} + \dot{\varepsilon}_{22} + \dot{\varepsilon}_{33}) \).

In the polymer model developed here, Equation 2.15 is utilized to modify \( E_n \) in Equation 2.1 based on the applied effective strain rate. Since the elastic modulus has been found to be strain rate insensitive under 1/s, the reference strain rate \( \dot{\varepsilon}_0 \) is assumed to equal 1/s in the polymer model. Hereby, only the strain rate constant \( C \) needs to be determined in order to obtain the rate-dependent elastic modulus. By using two sets of stress-strain curves obtained at one low strain rate and one high strain rate, the constant \( C \) can be easily determined. The strain rate effect on the elastic modulus will be further explained with an example in Chapter V.
2.2.2 Evolution of $\alpha$

The hydrostatic stress effect is accounted for by the state variable $\alpha$. In the original Goldberg’s polymer model, $\alpha$ evolutes with the effective inelastic strain rate $\dot{\varepsilon}^e$ as being stated in Equation 2.5. It is noted in the definition of $\dot{\varepsilon}^e$ in Equation 2.6 that the shear components of the elastic strain rate are included. In the case of pure shear loading such definition will result in a non-zero $\dot{\varepsilon}^e$, and consequently evolution of $\alpha$. This is not realistic since for pure shear loading the hydrostatic stress is not present. Furthermore, for multi axial loading where tensile/compressive and shear stresses are both present, the hydrostatic stress effect will be exaggerated.

To solve this discrepancy, the evolution law for $\alpha$ is rewritten by removing the shear components as following:

$$
\dot{\alpha} = q(\alpha_i - \alpha)(\dot{\varepsilon}^e)^\gamma
$$

(2.17)

where

$$
(\dot{\varepsilon}^e)^\gamma = \sqrt{\frac{2}{3}} \dot{\varepsilon}_i^j \dot{\varepsilon}_j^i \quad (i=j=1, 2, 3)
$$

(2.18)

2.2.3 Polymer damage model to improve unloading behavior

Numerous constitutive models can be found in the literature to simulate the mechanical behavior of polymers in the loading path. However, few authors are interested in predicting the unloading behavior. Under a complicated loading condition, such as in an impact test, the stress wave propagation induces loading and unloading in the material. Therefore, it is important to examine and correctly reproduce the polymer’s stress-strain diagram with strain reversal.
Experimental works\textsuperscript{41,42,43} have revealed that the unloading behavior of polymers has two features, even with small strains. Firstly, the slope of the stress–strain diagram at the beginning of the unloading path is smaller than the slope at the beginning of loading; secondly the concavity of the stress–strain curve changes during the unloading path. For example, Xia et al.\textsuperscript{43} found a decreased modulus in the unloading branch of their uniaxial tensile loading/unloading test with epoxy resin as shown in Figure 2.8. Similar unloading curve was also obtained by Brusselle-Dupend et al.\textsuperscript{44} with semicrystalline polymer polypropylene.

![Figure 2.8 Uniaxial loading-unloading curves by Xia et al.\textsuperscript{43}](image)

To improve the unloading behavior of their polymer constitutive models, Brusselle-Dupend et al.\textsuperscript{44} re-evaluated implicitly the apparent elastic modulus of the amorphous phase at the beginning of unloading. With modified modulus the constitutive model’s performance in the unloading path was much improved. Similarly, the model...
developed by Xia et al. was able to distinguish between the loading and unloading cases and a modified constant modulus was applied to the unloading path upon the loading-unloading switch instant.

In both of the above papers, the loading-unloading switch instant needs to be recognized, and then a new modified elastic modulus is applied to the unloading path. An alternative and straightforward way was proposed by Du Bois. He claimed that plastics have no constant elastic modulus and the unloading behavior can be approximated with a simple damage model decreasing Young’s modulus. This study will utilize Du Bois’ approach to improve the unloading performance of Goldberg’s polymer model.

In the modified Goldberg’s polymer model, the elastic modulus is replaced by an effective modulus as:

\[ E_{\text{eff}} = (1 - d)E \]  

(2.19)

where \( E \) is the undamaged elastic modulus and \( E_{\text{eff}} \) is the effective modulus, \( d \) is the damage variable.

With the increasing of inelastic strain, micro cracks are initiated and increase which result in the reduction of the effective elastic modulus. Therefore it is reasonable to relate the effective elastic modulus with the inelastic strain. Then the damage variable \( d \) can be defined as

\[ d = 1 - \exp(-C_i e'_e) \]  

(2.20)

where \( C_i \) is the polymer damage constant, and \( e'_e \) is the current effective inelastic strain defined by
$$e'_c = \sqrt[3]{2\delta_{ij}} e'_y$$
$$e'_y = e'_y - \varepsilon'_y \delta_{ij}$$

(2.21)

It is assumed that damage is an irreversible process. Therefore, $d$ has a monotonically increasing value. By choosing the exponential term in Equation 2.20, it is ensured that the damage variable $d$ ranges from 0 to 1. Note that, since in this polymer constitutive model, inelastic strain is assumed to be present all the time, there is no starting point of the modulus reduction.

If a series of uniaxial unloading tests are available, $d$ and $e'_y$ pairs can be determined. To determine $d$ from the unloading tests, Equation 2.19 is rewritten as

$$d = 1 - E_{eff} / E$$

(2.20)

As shown in Figure 2.9, $E_{eff}$ can be approximated by the slope of the line connecting the starting and the ending point of the unloading branch. The effective inelastic strain $e'_y$ can be calculated from the inelastic strain $e'$ shown in the figure. $d$ and $e'_y$ pairs can be obtained from loading-unloading tests of various strain limits, and plotted in one diagram, as demonstrated in Figure 2.10. By applying curve fitting to the data points on the diagram with a least squares regression analysis, the value of $C_1$ is obtained. Note that Figure 2.9 and 2.10 are both for demonstrative purpose.

The polymer damage model is not only important for the unloading prediction of polymers, it is found to be essential for the accurate deformation prediction of composites. Due to the thermo history during the preparation of the polymer matrix composite and the presence of fibers, there maybe more defects in the in-situ polymer
matrix than in the neat polymer. Therefore the damage constant may be chosen differently for the polymer matrix in the composite from that for the pure polymer.

Examples on the unloading prediction of polymers will be given in Chapter V. The importance of polymer damage model on the deformation prediction of composites will also be further demonstrated in that chapter.

Figure 2.9 Determination of effective elastic modulus

Figure 2.10 Determination of polymer damage constant
2.3 Composite micromechanical model and damage model

2.3.1 Composite micromechanical model

To compute the nonlinear, strain rate dependent deformation response of polymer matrix composites based on the response of the individual constituents, a micromechanical model has been developed by Goldberg et al.\textsuperscript{46}. In this model, the unit cell is defined to consist of a single fiber and its surrounding matrix. The fibers are assumed to be transversely isotropic and linear elastic with a circular cross-section. The matrix is assumed to be isotropic, with a nonlinear, rate dependent deformation response computed using the previously described polymer constitutive equations.

The unit cell is divided up into an arbitrary odd number of rectangular, horizontal slices, as shown in Figure 2.11. The top and bottom slices in the unit cell are composed of pure matrix. The remaining slices are composed of two subslices; one composed of fiber material and the other composed of matrix material. Due to symmetry, only one-quarter of the unit cell is analyzed. To be noted in the figure is the fact that a standard principal material coordinate system notation is used, namely, the 1-axis is aligned with the fiber direction, and the 2- and 3-axes are perpendicular to the fibers. When considering a composite lamina, it will be further assumed that the 2-axis is in the plane of the layer and the 3-axis is perpendicular to that plane which is the thickness direction.

In the fiber direction, the strains are assumed to be uniform in each subslice, and the stresses are combined using volume averaging. The in-plane transverse normal stresses and the in-plane shear stresses are assumed to be uniform in each subslice, and the strains are combined using volume averaging. The out-of-plane strains are assumed to be uniform in each subslice. The volume average of the out-of-plane stresses
in each subslice is assumed to be equal to zero, enforcing a plane stress condition on the global level for the slice.

The effective properties, effective stresses and effective inelastic strains are computed independently for each slice. For those slices composed of both fiber and matrix material, micromechanics equations are developed based on the uniform stress and uniform strain assumptions. For the slices composed of pure matrix, the stresses and inelastic strains in matrix are computed using the matrix elastic properties and the inelastic constitutive equations. The standard transversely isotropic compliance matrix (or isotropic in the case of matrix) is used to relate the local strains to the local stresses in the fiber and the matrix. The responses of each slice are combined using laminate theory to obtain the effective response of the corresponding lamina.

The composite model is developed for a single unidirectional lamina. Besides all of the material constants used in the polymer constitutive model, the fiber properties are required for this composite model, including the tensile and shear moduli and Poisson’s ratio, the fiber volume fraction and the number of fiber slices to be used in the unit cell.
2.3.2 Discussion on the composite unit cell selection

The selection of composite unit cell is based on the assumption of the geometric arrangement of the fibers and matrix within a composite. The unit cell selection used in this study is based on the square-packed array, as illustrated in Figure 2.12a. Another popular assumed arrangement is referred to as the hexagonal-packed array (Figure 2.12b). The names of the arrays are derived from the form of the polygons that connect the centers of neighboring fibers. Unit cells can then be determined based on these arrays and some examples are shown in Figure 2.13.

Unit cell selection is the first step in the composite model development in micromechanics. Some variations have been observed in the prediction of the deformation behavior of composites using different unit cell selections\textsuperscript{16,47}. This effect
may be studied in the future. Another important point to note here is that there is a maximum fiber volume ratio that the arrays can represent. The square-packed array can have a maximum fiber volume ratio of 78.54%, while the limit for the hexagonal-packed array can reach 90.69%. Due to the simplicity in the implementation and consequently computational efficiency, the unit cell based on the square-packed array was chosen in this study.
Figure 2.12 Square and hexagonal-packed array

(a) Square-packed array

(b) Hexagonal-packed array
2.3.3 Rate-sensitive fibers

In the original Goldberg’s composite model, all the strain rate effects are attributed to the rate sensitivity of the polymer matrix, and the fibers are considered to be rate independent. It is true for carbon fibers whose dynamic properties are the same as the quasi-static ones. However, as the other most commonly used fibers, the glass fibers exhibit rate sensitivity. Testing on composites also indirectly provided evidence of the rate sensitivity of different fibers. In their testing on unidirectional carbon/epoxy laminates, Hsiao and Daniel observed that the stiffening in the dynamic stress-strain
curves with increasing strain rate is lowest in the longitudinal case and highest in the transverse and shear cases; while Zhao and Gary\textsuperscript{50} claimed the failure strength in the fiber direction is more rate sensitive than the transverse direction after their compressive SHPB testing on glass fiber/epoxy composites. These phenomena indicate that the rate sensitivity of fibers influences the dynamic behavior of composites of different configurations.

To account for the rate dependence in rate-sensitive fibers, the Johnson-Cook model\textsuperscript{40} as in Equation 2.15 can be used to modify the initial elastic modulus of fibers. The strain rate in the fiber direction is used for $\dot{\varepsilon}$ in the equation instead of the effective strain rate defined by Equation 2.16.

2.3.4 Composite rate-dependent failure criteria

Numerous composite failure criteria can be found in literatures. Paris\textsuperscript{51} did a thorough survey on the failure criteria of fibrous composite materials. Hinton et al\textsuperscript{52} organized a “World-Wide Failure Exercise” which tested 12 of the leading theories for predicting failure in composite laminates against experimental evidence. One category referred to as the failure-mode-based failure criteria has gained more trust, such as Hashin\textsuperscript{53} and Chang\textsuperscript{54} failure criteria. In this study, Hashin criteria will be utilized with modifications to account for the strain rate effect.

Based on observations of specimen failure in tension with different fiber orientations, Hashin concluded that there are only two mechanisms of failure: fiber or matrix failure. He proposed four expressions of failure criteria for the 3D case and then particularized for the 2D case as following:
Fiber tensile failure \( (\sigma_{11}>0) \)

\[
\left( \frac{\sigma_{11}}{X_T} \right)^2 + \left( \frac{\sigma_{12}}{X_S} \right)^2 = 1
\]

(2.23)

Fiber compressive failure \( (\sigma_{11}<0) \)

\[
\left| \frac{\sigma_{11}}{X_C} \right| = 1
\]

(2.24)

Matrix tensile failure \( (\sigma_{22}>0) \)

\[
\left( \frac{\sigma_{22}}{Y_T} \right)^2 + \left( \frac{\sigma_{12}}{X_S} \right)^2 = 1
\]

(2.25)

Matrix compressive failure \( (\sigma_{22}<0) \)

\[
\left( \frac{\sigma_{22}}{2X_S} \right)^2 + \left[ \left( \frac{Y_C}{2X_S} \right)^2 - 1 \right] \left[ \frac{\sigma_{22}}{Y_C} + \left( \frac{\sigma_{12}}{X_S} \right)^2 \right] = 1
\]

(2.26)

where \( \sigma_{ij} \) is the macroscopic stress component, \( X_T \) and \( X_C \) is the ply tensile and compressive strength in the longitudinal direction respectively, \( Y_T \) and \( Y_C \) is the ply tensile and compressive strength in the transverse direction respectively and \( X_S \) is the ply in-plane shear strength.

To account for the rate dependence in their failure criterion, Thiruppukuzhi et al.\(^9\) modified the transverse strength with a rate-sensitive parameter for the matrix dominated failure modes. A similar approach is used here to modify Equation 2.25 and 2.26 so that the matrix failure modes are rate dependent.

\[
\left( \frac{\sigma_{22}}{Y_T} \right)^2 + \left( \frac{\sigma_{12}}{X_S} \right)^2 = \left( \frac{\dot{\epsilon}_{22}}{\dot{\epsilon}_0} \right)^{2\beta}
\]

(2.27)
where \( \dot{\epsilon}_0 \) is the reference strain rate at which the values of the strengths are determined. \( \dot{\epsilon}_0 \) is again assumed to be 1/s as in Section 2.2.1. \( \dot{\epsilon}_{22} \) is the applied strain rate at the transverse direction and \( \dot{\epsilon}_{22} \geq \dot{\epsilon}_0 \). \( \beta \) is the parameter that determines the rate sensitivity of strengths and can be approximated with the transverse strength values obtained at different strain rates, as being stated in Equation 2.29:

\[
\frac{Y_{T,C}}{Y_{T,C0}} = \left( \frac{\dot{\epsilon}_{22}}{\dot{\epsilon}_0} \right)^\beta
\]

where \( Y_{T,C0} \) is the transverse tensile or compressive strength value at the reference strain rate, and \( Y_{T,C} \) is the value at the current strain rate corresponding to \( \dot{\epsilon}_{22} \). An example will be given in Chapter V for the failure-envelop prediction carried out for unidirectional composites of various fiber orientations.

2.3.5 Composite progressive post failure model

While a variety of models exist for the prediction of failure initiation in composite material, they usually assume that in the post failure regime a lamina behaves in an ideally brittle manner with the stresses reduced to zero instantaneously. This assumption is not realistic due to the existence of the constraints that are imposed on the failed lamina by the adjacent laminas and undamaged elements in the neighborhood of the damage site. These constraints result in a gradual release of stresses instead of an abrupt one. Another issue associated with the instantaneous failure is the mesh sensitivity in modeling
of transverse impact with finite element methods. Recent studies\textsuperscript{19,55,56} have shown that post failure models with strain softening behavior are good remedies for these issues, which can significantly improve damage predictions and reduce mesh sensitivity.

Continuum damage mechanics (CDM) has become popular in modeling progressive damage in laminated composites. Williams and Vaziri\textsuperscript{57} gave a review on CDM based models for composite materials and first implemented the well-known CDM model developed by Matzenmiller, Lubliner and Taylor (MLT)\textsuperscript{58}.

As the characteristic of many CDM models, the MLT approach uses a single mathematical expression to describe the damage evolution as a function of strain over the entire loading range. Combining this with a linearly varying modulus reduction as a function of the damage results in a nonlinear stress–strain curve (corresponding to a damaging material) over the entire range of strains. The ascending and descending branches of the stress–strain curve are coupled in the MLT model thus restricting its versatility to represent different amounts of strain softening without affecting the elastic loading response. Being aware of the weakness of the MLT model, Williams and Vaziri\textsuperscript{59} proposed a more versatile approach which differentiates the undamaged elastic phase and the damaged phase. The new model is linear elastic before the damage initiation. After the onset of the damage a bilinear damage growth law is used to characterize the strain softening. Similarly in this study, the CDM model will be applied to the post failure stage only, while the nonlinear, rate dependent constitutive model will be kept unchanged to describe the composite behavior before failure.

In this study, two damage variables ($\omega_f$ and $\omega_m$) are used, each associated with failure in the fiber or matrix mode respectively. The damage thresholds are as
been described in Section 2.3.2. When the lamina fails in the fiber mode, the stresses related with the fiber direction \((\sigma_{ij}, j=1-3)\) are reduced to \((1-\omega_1)\sigma_{ij}\). When the lamina fails in the matrix mode, all stresses are reduced to \((1-\omega_2)\sigma_{ij}, (i, j=1-3)\). If failure occurs in both the fiber and matrix modes, the stresses related to the fiber direction \(\sigma_{ij}\) are reduced to \((1-\omega_1)(1-\omega_2)\sigma_{ij}, (j=1-3)\), and the remaining stresses are reduced to \((1-\omega_2)\sigma_{ij}, (i=2,3, j=1-3)\).

Before and at the onset of damage, the damage variable \(\omega_i\) is equal to zero. After the damage initiation, \(\omega_i\) evolves by the chosen damage growth law. It is important to note that damage is assumed to be an irreversible process such that the value of \(\omega_i\) at time \(t+\Delta t\) is always larger than that at time \(t\).

The MLT’s damage growth was based on a Weibull distribution of strengths, commonly associated with the strength of fiber bundles with initial defects. It takes the following form

\[
\omega = 1 - e^{-\frac{(\varepsilon - \varepsilon_f)^m}{m}}
\]

where \(\varepsilon\) is the current strain and \(\varepsilon_f\) is the failure strain, and the exponent \(m\) controls the shape of the stress-strain curve. The damage growth with the strain increase is plotted in Figure2.14 with various values of \(m\) exponent. It can be seen from the figure that with a small \(m\), the damage variable \(\omega\) increases fast at the beginning and keeps a slowly decreasing growth rate; while with a large \(m\), \(\omega\) keeps a small value at the beginning and
then increases abruptly to a large value. This implies that a high value of \( m \) will result in a brittle failure and a low one gives the material a more ductile behavior.

An alternative damage growth law was proposed by Yen\(^{19}\) in which the damage starts to grow after the failure initiation:

\[
\omega = 1 - e^{-\left| \frac{\epsilon}{\epsilon_f} \right|^m}, \quad \epsilon \leq \epsilon_f
\]

The effect of the damage exponent \( m \) on the damage growth is shown in Figure 2.15. It can be seen that the damage growth with different exponent \( m \) follows the same shape, while a larger \( m \) results in a faster damage growth. This damage growth law is more straightforward and consistent than the MLT model. Therefore it was chosen as the law.
for the composite post failure damage growth in this study and was implemented into the composite model. An example will be given in Chapter V to illustrate the effect of exponent $m$ on the damage progression.

![Figure 2.15 Effect of $m$ on the damage growth in Yen’s model](image)

Figure 2.15 Effect of $m$ on the damage growth in Yen’s model
CHAPTER III

EXPLICIT FINITE ELEMENT METHOD IN LS-DYNA

The finite element code that was chosen for implementation of the composite model is the commercially available software LS-DYNA. This chapter will first give an overview of LS-DYNA, including aspects such as the explicit finite element method, formulation of the shell elements, and time step calculation. Only the features that are relevant to this study are covered. Then a stand alone explicit finite element code that was developed in order to better understand the LS-DYNA procedure will be described.

3.1 LS-DYNA overview

The origin of LS-DYNA\textsuperscript{60} can be traced back to the public domain software, DYNA3D, which was developed by the Lawrence Livermore National Laboratory in 1976. The early applications were primarily for the stress analysis of structures subjected to a variety of impact loadings. In 1988, Livermore Software Technology Corporation was founded to continue the development of DYNA3D as a commercial
version called LS-DYNA3D, later shortened to LS-DYNA. Through the past decade’s development, considerable progress has been made with hundreds of new features added, including contact-impact algorithms, new element formulations, new material types etc. LS-DYNA became a general purpose finite element tool for large deformation dynamic response analysis of structures. Relevant to this study, LS-DYNA has gained and continues to gain wide usage in the automotive industry in the field of crashworthiness and occupant safety, and the aircraft industry in bird strike and blade containment analysis etc.

The main solution methodology in LS-DYNA is based on explicit time integration. An implicit solver is also available but with limited capabilities, including structural analysis and heat transfer. LS-DYNA has a large selection of element types which include four node tetrahedron and eight node solid elements, two node beam elements, three and four node shell elements, eight node solid shell elements, truss elements, membrane elements, discrete elements and rigid bodies. A variety of element formulations are available for each element type. Several types of contact interfaces can be defined in LS-DYNA. These include surface to surface, nodes to surface, nodes tied to surface, and surface tied to surface contacts and etc. LS-DYNA currently contains approximately one hundred material constitutive models and ten equations-of-state to cover material behavior ranging from the very simple elastic material to the elastic-plastic strain-rate-dependent material. The most advantageous capability of LS-DYNA over other finite element codes is its contact algorithm. Several types of contact interfaces can be defined in LS-DYNA including surface to surface, nodes to surface, nodes tied to surface, and surface tied to surface contacts and etc.
3.2 Explicit Finite Element Method

This section will describe the explicit finite element method used in LS-DYNA, including its Lagrangian formulation, governing equations, shell formulation and time step calculation.

3.2.1 Lagrangian formulation

Finite element equations can be formulated in Lagrangian or Eulerian spaces. In the Lagrangian formulation, the material is fixed to the finite element mesh and the mesh deforms and moves with the material (Figure 3.1a, 3.1b). In Eulerian space the finite element mesh is stationary and the material flows through this mesh (Figure 3.1a, 3.1c). This type of formulation is well suited for fluid dynamic problems. As most structural analysis problems are formulated in Lagrangian space, the finite element formulation used in LS-DYNA is derived in Lagrangian space. However, Arbitrary Lagrangian-Eulerian (ALE) formulation is also available for fluid-like material simulation. It consists of a Lagrangian time step followed by a “remap” or “advection” step. For example, after the mesh is deformed in Lagrangian space as shown in Figure 3.1b, node 2 is moved back to its original position in Figure 3.1c and the material is moved from element 1 to element 2. Element 2 is gaining material (Flux>0) while element 1 is loosing material (Flux<0). This study will focus on the Lagrangian formulation.

In order to solve a nonlinear three-dimensional dynamic problem, the deformed geometry of a body that is subjected to external forces needs to be calculated. To formulate this problem, each particle in the body is assigned three coordinates \( x_i \) \((i=1, 2, 3)\). These coordinates represent the location of the particle in space at the current time.
In the Lagrangian formulation, the current coordinates of each particle in the body are expressed as functions of the particle's initial coordinates, $X_\alpha (\alpha = 1, 2, 3)$, and the current time $t$:

$$x_i = x_i (X_\alpha, t) \tag{3.1}$$

At time $t=0$, the initial coordinates, velocities, and accelerations of the particle are:

$$x_i (X_\alpha, 0) = X_\alpha$$
$$\dot{x}_i (X_\alpha, 0) = V_\alpha$$
$$\ddot{x}_i (X_\alpha, 0) = A_\alpha \tag{3.2}$$

where $V_\alpha$ and $A_\alpha$ denote the initial velocities and initial accelerations of the particle respectively. $\dot{x}_i$ are the current velocities and $\ddot{x}_i$ are the current accelerations of the particle.

In the Lagrangian formulation, the material derivative of a function $f(X, t)$ with respect to time is equal to its partial derivative with respect to time:

$$\frac{df}{dt} = \frac{\partial f}{\partial t} + \frac{\partial f}{\partial X_\alpha} \frac{\partial X_\alpha}{\partial t} = \frac{\partial f}{\partial t} \tag{3.3}$$

In addition, in the Lagrangian formulation, the conservation of mass and energy equations are automatically satisfied. This simplifies the formulation since only the equations of conservation of momentum need to be solved.
3.2.2 Governing equations

The three-dimensional body is located in a Lagrangian space, as shown in Figure 3.2. The body is subjected to external body force $b_i(t)$ (force per unit volume) over its entire volume $V$, traction forces $t_i(t)$ (forces per unit area) over a portion of its outer surface $S_t$, and prescribed displacements $d_i(t)$ over the surface $S_d$. 
To solve the problem, one must seek a solution to the momentum equation

\[ \sigma_{ij,j} + \rho \rho \ddot{x}_i = 0 \]  \hspace{1cm} (3.4)

satisfying the traction boundary conditions over the traction surface \( S_t \)

\[ \sigma_{ij} n_j = t_i (t) \]  \hspace{1cm} (3.5)

and the displacement boundary conditions over the displacement boundary \( S_d \)

\[ x_i (X_a, t) = d_i (t) \]  \hspace{1cm} (3.6)

where \( \sigma_{ij} \) is Cauchy's stress tensor, \( \rho \) is the material density, \( \ddot{x}_i \) is the acceleration, the comma denotes covariant differentiation, and \( n_j \) is the outward normal unit vector to the traction surface \( S_t \).

These equations are stating the problem in the strong form which means they need to be satisfied at every point in the body or in the surface. To solve the problem
numerically using the finite element method, the problem has to be defined in the weak form, in which the conditions have to be met only on an average or integral sense.

In the weak form equation, an arbitrary virtual displacement $\delta x_i$, that satisfies the displacement boundary condition in $S_d$, is introduced. Multiplying Equation 3.4 by the virtual displacement and integrating over the volume of the body yields:

$$\int_V \left( \sigma_{y,j} + \rho b_i - \rho \ddot{x}_i \right) \delta x_i \, dV = 0 \quad (3.7)$$

Noting that

$$(\sigma_{y} \delta x_i)_{,j} = \sigma_{ij,j} \delta x_i + \sigma_{ij} \delta x_{i,j} \quad (3.8)$$

Substituting for the first term in Equation 3.7 leads to:

$$\int_V \left( (\sigma_{y} \delta x_i)_{,j} - \sigma_{y} \delta x_{i,j} + \rho b_i \delta x_i - \rho \ddot{x}_i \delta x_i \right) \, dV = 0 \quad (3.9)$$

The first term in Equation 3.9 can be expressed as:

$$\int_V (\sigma_{y} \delta x_i)_{,j} \, dV = \int_{S_t} (\sigma_{y} \delta x_i) n_j \, dS \quad (3.10)$$

With the traction boundary in Equation 3.5, Equation 3.10 can be written as:

$$\int_V (\sigma_{y} \delta x_i)_{,j} \, dV = \int_{S_t} t_i \delta x_i \, dS \quad (3.11)$$

Substituting Equation 3.11 into Equation 3.9 leads to the weak form of the equilibrium equations:

$$\int_V \rho \ddot{x}_i \delta x_i \, dV + \int_V \sigma_{y} \delta x_{i,j} \, dV - \int_V \rho b_i \delta x_i \, dV - \int_{S_t} t_i \delta x_i \, dS = 0 \quad (3.12)$$

It is a statement of the principle of virtual work for the general three-dimensional problem defined in Figure 3.2.
3.2.3 Spatial discretization

The next step in deriving the finite element equations is spatial discretization. This is achieved by superimposing a mesh of finite elements interconnected at nodal points. Then shape functions are introduced to establish a relationship between the displacements at inner points in the elements and the displacements at the nodal points:

\[ \delta x_i = \sum_{\alpha=1}^{n} N_{\alpha} \delta x_{\alpha i} \]  

(3.13)

where \( \delta x_i \) are the displacements at a point inside the element, \( n \) is the number of nodal points defining the element, \( N_{\alpha} \) is the shape function at node \( \alpha \), and \( \delta x_{\alpha i} \) are the displacements at node \( \alpha \). Similar expressions can be written for the coordinates, velocities, and acceleration of a point inside the element.

\[ x_i = \sum_{\alpha=1}^{n} N_{\alpha} x_{\alpha i} \]
\[ \dot{x}_i = \sum_{\alpha=1}^{n} N_{\alpha} \dot{x}_{\alpha i} \]  

(3.14)
\[ \ddot{x}_i = \sum_{\alpha=1}^{n} N_{\alpha} \ddot{x}_{\alpha i} \]

where \( x_{\alpha i} \), \( \dot{x}_{\alpha i} \) and \( \ddot{x}_{\alpha i} \) are the displacements, velocities, and accelerations at nodal point \( \alpha \) respectively.

The shape functions used in LS-DYNA are the isoparametric shape functions. To give an example, a 4-node shell element is shown in Figure 3.3. The surface of the shell element is approximated by calculating the coordinates of any point in the shell as a bilinear function of the coordinates at the 4 nodal points as expressed in Equation 3.15:
\[
\begin{pmatrix}
  x \\
  y \\
  z
\end{pmatrix} = N_1 \begin{pmatrix} x_1 \\
  y_1 \\
  z_1
\end{pmatrix} + N_2 \begin{pmatrix} x_2 \\
  y_2 \\
  z_2
\end{pmatrix} + N_3 \begin{pmatrix} x_3 \\
  y_3 \\
  z_3
\end{pmatrix} + N_4 \begin{pmatrix} x_4 \\
  y_4 \\
  z_4
\end{pmatrix}
\] (3.15)

A local coordinate system \((\xi, \eta)\) is created at the center of element as shown in Figure 3.4. Any point inside the element has local coordinates ranging between \(-1\) and \(+1\) in both directions. The nodal points have local coordinates equal to \(-1\) or \(+1\). The shape functions for the 4-node shell element can be expressed as:

\[
N_1 = \frac{1}{4}(1 - \xi)(1 - \eta)
\]
\[
N_2 = \frac{1}{4}(1 + \xi)(1 - \eta)
\]
\[
N_3 = \frac{1}{4}(1 + \xi)(1 + \eta)
\]
\[
N_4 = \frac{1}{4}(1 - \xi)(1 + \eta)
\] (3.16)

User Input

4 Nodal Points

N1 \((x_1, y_1, z_1)\)
N2 \((x_2, y_2, z_2)\)
N3 \((x_3, y_3, z_3)\)
N4 \((x_4, y_4, z_4)\)

Figure 3.3 4-node shell element surface
The finite element equations are derived by discretizing the virtual work Equation 3.12 in space as:

\[
\sum_{m=1}^{M} \left\{ \int_{V_m} \rho \ddot{x}_i \delta \chi_{i} dV_m + \int_{V_m} \sigma_{ij} \delta \chi_{i,j} dV_m - \int_{S_m} \rho \dot{b}_i \delta \chi_{i} dS_m - \int_{V_m} t_i \delta \chi_{i} dV_m \right\} = 0
\]  

(3.17)

where \( M \) is the total number of elements in the system and \( V_m \) is the volume of the element. Replacing the virtual displacements \( \delta x_i \), and the accelerations \( \ddot{x}_i \) with the interpolations from Equations 3.13 and 3.14 gives:

\[
\sum_{m=1}^{M} \left\{ \int_{V_m} \rho (N_{\beta} \ddot{x}_{\beta,j})(N_{\alpha} \delta x_{\alpha}) dV_m + \int_{V_m} \sigma_{ij} (N_{\alpha,j} \delta x_{\alpha}) dV_m - \int_{S_m} \rho b_i (N_{\alpha} \delta x_{\alpha}) dS_m - \int_{V_m} t_i (N_{\alpha} \delta x_{\alpha}) dV_m \right\} = 0
\]  

(3.18)

where \( \delta x_{\alpha} \) and \( \ddot{x}_{\alpha} \) are the virtual displacement and the accelerations at the nodal points respectively. Equation 3.18 can be simplified to:
Equation 3.19 can be rewritten as:

\[
\sum_{m=1}^{M} \left\{ \int_{V_m} \rho N_{\alpha} N_{\beta} \dddot{x}_{\beta_i} dV_m + \int_{V_m} N_{\alpha,i} \sigma_{ij} dV_m \right. \\
- \int_{V_m} N_{\alpha} \rho b_i dV_m - \int_{S_i} N_{\alpha} t_i dS_m \right\} = 0
\] (3.19)

In matrix form, Equation 3.20 becomes:

\[
[M] \ddot{\{x\}} = \{F\}
\] (3.21)

where \([M]\) is the mass matrix, \(\{\dddot{x}\}\) is the acceleration vector, and \(\{F\}\) is the vector summation of all the internal and external forces. Equation 3.21 is the finite element equation that needs to be solved at each time step. The solution procedure consists of first computing all internal and external forces and summing them for each degree of freedom of the system. The accelerations are then determined by dividing these forces at the corresponding degree of freedom by the mass.

3.2.4 Lumped mass matrix

The mass matrix \([M]\) in Equation 3.21 is expressed as:

\[
[M] = \sum_{m=1}^{M} \int_{V_m} \rho N^i N \ dV_m
\] (3.22)
The mass matrix obtained from the above equation is not diagonal (some of the off-diagonal terms are nonzero) and is called the consistent mass matrix. In order to avoid solving any system equations when updating the nodal accelerations, the formulation in LS-DYNA uses lumped masses to diagonalize the mass matrix. This is achieved by adding all the components of a row to the diagonal component of that row and then setting all but the diagonal term to zero. A diagonal mass matrix renders the solution to accelerations in Equation 3.21 trivial since no matrix inversion is needed. This results in the reduction of the computational time and storage needed for the solution. In addition, it has been found that a lumped mass matrix provides a more accurate solution in many cases, especially for high rate transient and wave propagation problems.

3.2.5 Central difference method

The governing finite element Equation 3.21 needs to be solved at each time step. In order to do so, it is written in discrete form as:

\[
[M]\{\ddot{x}\}_n = \{F\}_n
\]  

(3.23)

where \(\{\ddot{x}\}_n\) is the acceleration vector at time \(t_n\), and \(\{F\}_n\) is the sum of all external and internal force vectors at time \(t_n\). The time interval between two successive points in time, \(t_{n-1}\) and \(t_n\), is the time step \(\Delta t_n\) (\(\Delta t_n = t_n - t_{n-1}\)). LS-DYNA uses time steps varying with time. This is necessary in most practical calculations since the stable time step will change as the mesh deforms.
In numerical analysis, integration methods are classified according to the structure of the time difference equation. The difference formula is called explicit if the equation for the function at time step $n$ only involves the derivatives at previous time steps; otherwise it is called implicit. Explicit integration methods generally lead to solution schemes which do not require the solution of a coupled system of equations, provided that a lumped mass matrix is used instead of a consistent mass matrix.

In computational mechanics and physics, the central difference method is a popular explicit method. The formulation in LS-DYNA uses the central difference explicit method to discretize the finite element equation in time. The displacement and acceleration vectors are computed at times $t_1, \ldots, t_n, t_{n+1}, \ldots t_f$ (where $t_f$ is the final problem time) and the velocity vector is computed at times $t_{1/2}, \ldots, t_{n-1/2}, t_{n+1/2}, \ldots t_{f-1/2}$. The calculation is started with the initial values for the displacements and accelerations at time $t_0$ and an approximation of the velocities at time $t_{-1/2}$ (half a time step before time $t_0$). The solution is incremented using the central difference equations as.

\[
\dot{x}_{n+\frac{1}{2}} = \dot{x}_{n-\frac{1}{2}} + \ddot{x}_n \Delta t_n \tag{3.24}
\]

\[
x_{n+1} = x_n + \dot{x}_{n+\frac{1}{2}} \Delta t_{n+\frac{1}{2}} \tag{3.25}
\]

\[
\ddot{x}_{n+1} = M^{-1} f_{n+1} \tag{3.26}
\]

\[
\Delta t_{n+\frac{1}{2}} = \frac{1}{2}(\Delta t_n + \Delta t_{n+1}) \tag{3.27}
\]

From Equation 3.24, the velocity vector at $t_{n+1/2}$ is computed. This velocity vector is then used in Equation 3.25 to compute the displacement vector at $t_{n+1}$. With strain-displacement relationship and constitutive equation, stresses can be calculated to
obtain the internal forces on element nodes. The internal and external forces are summed at each nodal degree of freedom to assemble the force vector. Then the acceleration vector at time $t_{n+1}$ can be determined with Equation 3.26. The final step is to update the time step $\Delta t_{n+1/2}$ using Equation 3.27. The calculation of time step $\Delta t_{n+1}$ will be explained in Section 3.4.

### 3.3 Implicit method vs. explicit method

Most popular finite element software, such as ANSYS$^{62}$ and ABAQUS/Standard$^{63}$, are developed with implicit integration methods. In implicit methods, all of the unknown variables are computed simultaneously at the same point in time. The advantage of this method over the explicit method is that it is unconditionally stable for linear, transient problems. Time steps that can be used with implicit integrators are much larger than those required for explicit methods. However, implicit methods require much more computer resources and computation time per cycle than explicit methods. Large matrices need to be stored and a large system of algebraic equations needs to be solved at each cycle. Consequently, implicit methods are mostly used for static and low rate dynamic analyses.

The explicit method is conditionally stable; i.e. for the solution to be stable, the time step has to be small enough such that information does not propagate across more than one element per time step. A typical time step for explicit solutions is in the order of $10^{-6}$ seconds. This restriction makes the explicit method inadequate for long duration dynamic problems. The advantages of the explicit method are that the time integration is
easy to implement, the material non-linearity can be cheaply and accurately treated, and the computer resources required are small even for large problems. These advantages make the explicit method ideal for short-duration nonlinear dynamic problems, such as impact and penetration. Based on the explicit finite element method, LS-DYNA excels in dynamic analyses such as contact-impact problems, and it is increasingly gaining favor in the automobile and aircraft industries.

3.4 Time step control

The time step of an explicit analysis is determined as the minimum stable time step in any deformable finite element in the mesh. The choice of the time step is critical since a large time step can result in an unstable solution, while a small time step can make the computation inefficient. Therefore, an accurate estimation of the critical time step has to be determined. The critical time step has to be small enough such that the stress wave does not travel across more than one element at each time step. This is achieved by using the Courant criteria:

$$\Delta t_e = \frac{l}{c}$$  \hspace{1cm} (3.28)

where $\Delta t_e$ is the critical time step of an element in the model, $l$ is the characteristic length, and $c$ is the wave speed.

For shell elements, the characteristic length is given by:

$$l = \frac{A}{\max(L_1, L_2, L_3, L_4)} \quad (4 \text{- node})$$  \hspace{1cm} (3.29)

$$l = \frac{2A}{\max(L_1, L_2, L_3)} \quad (3 \text{- node})$$
where $A$ is the element area, and $L_i (i=1,\ldots,4)$ is the length of the sides defining the shell elements.

The wave speed $c$ can be expressed as:

$$c = \sqrt{\frac{E}{\rho(1-\nu^2)}}$$  \hspace{1cm} (3.30)

where $E$, $\rho$ and $\nu$ are the Young's modulus, density and Poisson's ratio of the material respectively.

The time step of the system is determined by taking the minimum value over all elements:

$$\Delta t_{n+1} = \alpha \cdot \min \{ \Delta t_1, \Delta t_2, \Delta t_3, \ldots, \Delta t_M \}$$  \hspace{1cm} (3.31)

where $M$ is the number of elements. For stability reasons, the scale factor $\alpha$ is typically set to a value of 0.9 (the default in LS-DYNA) or some smaller value.

3.5 Belytschko-Tsay shell formulation

The fiber reinforced polymer matrix composites in this study are normally thin structures. Therefore, shell elements will be utilized in the material model development in this study instead of solid elements to enhance the computational time efficiency.

LS-DYNA has over 20 types of shell elements. Most shell element formulations are based on the collapsing of an eight-noded solid element to form a four-noded shell or plate element. The Hughes-Liu element family\textsuperscript{64,65,75} was the first set of shell elements to be implemented in LS-DYNA. Next, the Belytschko-Tsay shell element\textsuperscript{66,67} was implemented in LS-DYNA as a computationally efficient alternative to the Hughes-Liu
shell element. It became the default shell element formulation in LS-DYNA. In this section, the formulation for the Belytschko-Tsay shell element is presented.

The efficiency of the Belytschko-Tsay shell over other shell formulations is obtained from mathematical simplifications based on co-rotational and velocity-strain formulations.

3.5.1 Co-rotational coordinates

The co-rotational formulation avoids the complexities of nonlinear mechanics by embedding a coordinate system in the element. The coordinate system unit vectors ($\hat{e}_1$, $\hat{e}_2$, $\hat{e}_3$) are determined from the coordinates of the four nodes defining the element. The unit vectors are updated at each time increment cycle using the current coordinates of the nodes at that point in time, hence the coordinate system rotates with the element. The procedure for constructing the co-rotational coordinate system begins by first finding the unit vector normal to the two main diagonals of the element. This vector is named as $\hat{e}_3$, as shown in Figure 3.5. It can be expressed as:

$$\hat{e}_3 = \frac{\vec{r}_{13} \times \vec{r}_{24}}{||\vec{r}_{13} \times \vec{r}_{24}||} \quad (3.32)$$

where $\vec{r}_{13}$ is the vector between nodes 1 and 3, $\vec{r}_{24}$ is the vector between nodes 2 and 4, and “×” denotes a cross product. The unit normal vector $\hat{e}_3$ is chosen to be the closest vector to $\vec{r}_{12}$ (the vector between nodes 1 and 2) and normal to $\hat{e}_3$.

$$\hat{e}_1 = \frac{\vec{r}_{12} - (\vec{r}_{12} \cdot \hat{e}_3) \hat{e}_3}{||\vec{r}_{12} - (\vec{r}_{12} \cdot \hat{e}_3) \hat{e}_3||} \quad (3.33)$$
The unit vector $\hat{e}_2$ is taken in such a way that $(\hat{e}_1, \hat{e}_2, \hat{e}_3)$ forms a right-hand Cartesian coordinate system:

$$\hat{e}_2 = \hat{e}_3 \times \hat{e}_1$$ \hspace{1cm} (3.34)

These vectors are used to define a transformation matrix between the global and local element coordinate systems.

![Co-rotational coordinate system](image)

Figure 3.5 Co-rotational coordinate system

If the four nodes of the element are coplanar, then the unit vectors $\hat{e}_1$ and $\hat{e}_2$ lie in the plane of the element and $\hat{e}_3$ is identical to the thickness direction. As the element deforms, the unit normal vector $\hat{e}_3$ may deviate from the element thickness direction. The formulation assumes that this deviation has to remain small. In other words, it is assumed that the difference between the rotation of the material and the rotation of the local coordinates is small. This small rotation condition does not put restrictions on the rigid body rotation of the element. The restriction is rather put on the out-of-plane deformations in the element which are small for most engineering problems.
3.5.2 Velocity-strain displacement relations

Using the Mindlin theory of plates and shells\textsuperscript{68}, the displacements at any point in the shell element can be decomposed into two components: nodal translation and nodal rotation. The nodal translation components are attributed to the mid-surface displacements and the nodal rotation components are associated with rotations of the element fibers. The Belytschko-Tsay shell formulation utilizes the Mindlin theory to define the velocity of any point in the shell as.

\[
\nu = \nu^m + z \hat{e}_3 \times \theta
\]  

where \( \nu^m \) is the velocity vector at the mid-surface of the shell, \( \theta \) is the angular velocity vector, and \( z \) is the distance from the point to the element mid-surface. The corresponding co-rotational components of the velocity-strain (deformation rate) can be expressed in terms of velocities as follows:

\[
\dot{\epsilon}_v = \frac{1}{2} \left( \frac{\partial \nu_i}{\partial x_j} + \frac{\partial \nu_j}{\partial x_i} \right)
\]  

Substituting for the velocity from Equation 3.36 into Equation 3.37, the velocity-strains can be expressed as:
\[
\begin{align*}
\dot{e}_{11} &= \frac{\partial \nu_{1}^{m}}{\partial x_{1}} + z \frac{\partial \theta_{z}}{\partial x_{1}} \\
\dot{e}_{22} &= \frac{\partial \nu_{2}^{m}}{\partial x_{2}} - z \frac{\partial \theta_{z}}{\partial x_{2}} \\
\dot{e}_{12} &= \frac{1}{2} \left( \frac{\partial \nu_{1}^{m}}{\partial x_{2}} + \frac{\partial \nu_{2}^{m}}{\partial x_{1}} + z \frac{\partial \theta_{z}}{\partial x_{2}} - z \frac{\partial \theta_{z}}{\partial x_{1}} \right) \\
\dot{e}_{23} &= \frac{1}{2} \left( \frac{\partial \nu_{3}^{m}}{\partial x_{2}} - \theta_{1} \right) \\
\dot{e}_{13} &= \frac{1}{2} \left( \frac{\partial \nu_{3}^{m}}{\partial x_{1}} + \theta_{2} \right)
\end{align*}
\] (3.37)

Using Equation 3.14, the mid-surface velocities \( \nu^{m} \) and the angular velocities \( \theta \) can be expressed in terms of their values at the nodal point:

\[
\begin{align*}
\nu^{m} &= N_{\text{I}} \nu_{\text{I}} \\
\theta &= N_{\text{I}} \theta_{\text{I}}
\end{align*}
\] (3.38)

where \( N_{\text{I}} \) is the bilinear shape function as defined in Equation 3.16. The subscript \( \text{I} \) is summed over all the element’s nodes.

The Belytschko-Tsay shell element uses a single in-plane integration point at the center of the element \( (\xi = \eta = 0) \), hence it is called an underintegrated shell element. By substituting Equations 3.38 in Equations 3.37 the velocity-strain nodal-velocity relationship is obtained:

\[
\begin{align*}
\dot{e}_{11} &= B_{11}\nu_{1} + z B_{11}\theta_{21} \\
\dot{e}_{22} &= B_{22}\nu_{2} - z B_{22}\theta_{11} \\
\dot{e}_{12} &= \frac{1}{2} \left( B_{11}\nu_{1} + B_{11}\nu_{2} + z B_{22}\theta_{21} - z B_{22}\theta_{11} \right) \\
\dot{e}_{23} &= \frac{1}{2} \left( B_{23}\nu_{3} - N_{1}\theta_{11} \right) \\
\dot{e}_{13} &= \frac{1}{2} \left( B_{13}\nu_{3} + N_{1}\theta_{21} \right)
\end{align*}
\] (3.39)
where

\[ B_{11} = \frac{\partial N_I}{\partial x_1} \]  (3.40)

\[ B_{21} = \frac{\partial N_I}{\partial x_2} \]  (3.41)

Even though one integration point is used in the 1-2 plane of the shell element, two or more integration points can be used along the thickness of the element.

3.5.3 Stress resultants and nodal forces

After the velocity strains are obtained, with suitable constitutive equations, stresses \( \sigma \) at the in-plane integration point can be calculated. The resulting stresses \( f^R \) are integrated through the thickness of the shell to obtain local resultant forces and moments:

\[ f^R = \int \sigma dz \]  (3.42)

\[ m^R = -\int z\sigma dz \]  (3.43)

The above element-centered resultant force and moment are then distributed to the nodal points by the following equations:

\[ f_{11} = A\left(B_{11}f^R_{11} + B_{21}f^R_{12}\right) \]  (3.44a)

\[ f_{21} = A\left(B_{21}f^R_{22} + B_{11}f^R_{12}\right) \]  (3.44b)

\[ f_{31} = A\kappa\left(B_{11}f^R_{13} + B_{21}f^R_{23}\right) \]  (3.44c)

\[ m_{11} = A\left(B_{22}m^R_{22} + B_{12}m^R_{12} - \frac{\kappa}{4}f^R_{23}\right) \]  (3.44d)
\[ m_{2j} = -A \left( B_{11,1} m_{11}^R + B_{21,1} m_{12}^R - \frac{\kappa f_{13}^R}{4} \right) \]  \hspace{1cm} (3.44e)

\[ m_{3j} = 0 \] \hspace{1cm} (3.44f)

where \( A \) is the element area and \( \kappa \) is the shear deduction factor from the Mindlin theory\(^6\).

The above local forces and moments are then transformed to the global coordinate system and summed over all the nodes. Together with the external forces and boundary conditions, nodal accelerations are solved with the global equations of motion.

### 3.6 Stand-alone explicit FEM code

The objective of this study is to implement a nonlinear rate-dependent composite model into the explicit finite element code, LS-DYNA. There is an option in LS-DYNA that users can substitute their own material constitutive model in the code as a user defined material (UMAT). To examine and trouble shoot the material model, a stand-alone explicit finite element code was developed using the procedures as described above.

A simple example problem is used to illustrate the procedure of the explicit FEM code as shown in Figure 3.6. It consists of 5 by 5 shell elements with one edge fixed and the opposite edge pulled at a constant speed. Each shell element has one integration point at the center.
The major stages of the stand-alone explicit FEM code can be explained as follows:

1. **displacement increment on each node**
   \[
   \left( \Delta d_x, \Delta d_y, \Delta d_z, \Delta \theta_x, \Delta \theta_y \right)
   \]

   - **Strain-displacement relationship**
   - **B Matrix**
   \[
   \begin{align*}
   B_{ix} &= \frac{\partial N_i}{\partial x} \Delta d_x + z B_{11} \Delta \theta_y \\
   \Delta \varepsilon_x &= B_{11} \Delta d_y + B_{21} \Delta \theta_x \\
   \Delta \varepsilon_y &= B_{21} \Delta d_y - B_{11} \Delta \theta_x \\
   \Delta \varepsilon_z &= B_{31} \Delta d_y + B_{13} \Delta \theta_x + z \left( B_{23} \Delta d_y - B_{21} \Delta \theta_x \right) \\
   \Delta \varepsilon_{xy} &= B_{12} \Delta d_y + B_{22} \Delta \theta_x + z \left( B_{32} \Delta d_y - B_{31} \Delta \theta_x \right) \\
   \Delta \varepsilon_{xz} &= B_{13} \Delta d_y + B_{33} \Delta \theta_x + z \left( B_{23} \Delta d_y - B_{21} \Delta \theta_x \right) \\
   \Delta \varepsilon_{yz} &= B_{23} \Delta d_y + B_{32} \Delta \theta_x + z \left( B_{13} \Delta d_y - B_{12} \Delta \theta_x \right)
   \end{align*}
   \]

2. **strain increment at integration point**
   \[
   \left( \Delta \varepsilon_x, \Delta \varepsilon_y, \Delta \varepsilon_{xy}, \Delta \varepsilon_{xz}, \Delta \varepsilon_{yz} \right)
   \]

   - **Material constitutive equations**
   - **UMAT**
3. Stresses calculated at each integration point
\[
\left(\sigma_x, \sigma_y, \sigma_{xy}, \sigma_{xz}, \sigma_{yz}\right)
\]
Resultant forces and moments
\[
f_{\alpha}^R = \int \sigma_{\alpha \beta} dz \\
m_{\alpha}^R = -\int z \sigma_{\alpha \beta} dz
\]
\[\downarrow\] Summation over all elements

4. Internal forces on each node
\[
\left(f_{xI}, f_{yI}, f_{zI}, m_{xI}, m_{yI}\right) = f_{\text{int}}
\]
\[
f_{xI} = A \left(B_{11} f_{x}^R + B_{21} f_{y}^R \right)
\]
\[
f_{yI} = A \left(B_{21} f_{y}^R + B_{11} f_{x}^R \right)
\]
\[
f_{zI} = A \kappa \left(B_{11} f_{z}^R + B_{21} f_{y}^R \right)
\]
\[
m_{xI} = A \left(B_{21} m_{y}^R + B_{11} m_{y}^R - \frac{\kappa}{4} f_{y}^R \right)
\]
\[
m_{yI} = -A \left(B_{11} m_{x}^R + B_{21} m_{x}^R - \frac{\kappa}{4} f_{x}^R \right)
\]
\[\downarrow\]

5. Governing equation:
\[
Ma_n = f_{n}^{\text{ext}} - f_{n}^{\text{int}}
\]
B.C. \[\Rightarrow a_n = M^{-1} \left(f_{n}^{\text{ext}} - f_{n}^{\text{int}} \right)
\]
\[\downarrow\]

6. Update nodal displacement
\[
\nu_{n+1/2} = \nu_{n-1/2} + a_n \Delta t
\]
\[
d_{n+1} = d_n + \nu_{n+1/2} \Delta t
\]
Central difference method
The material constitutive model is called for to proceed from stage 2 to stage 3. This is the place where users insert their UMAT. The procedure on how to develop a UMAT in LS-DYNA will be explained in the next Chapter. The stand-alone explicit FEM code is attached in Appendix.
CHAPTER IV

MATERIAL MODEL IMPLEMENTATION IN LS-DYNA

With LS-DYNA continuously expanding and developing its capabilities, new material types have been added every year, from a few at the beginning to approximately a hundred to date. Many of the new material models were contributed by the users through the user defined material (UMAT) option in LS-DYNA. In this Chapter, the procedure on how to develop UMAT in LS-DYNA will be explained in detail. Before that, the currently available composite material models in LS-DYNA will be reviewed.

4.1 Current composite models in LS-DYNA

As an important category of material types, 15 composite models can be found in the LS-DYNA material library. These include:

Material Model 2: Orthotropic Elastic

Material Model 22: Composite Damage

Material Model 23: Temperature Dependent Orthotropic
Material Model 26: Honeycomb
Material Model 32: Laminated Glass Model
Material Model 40: Nonlinear Orthotropic
Material Model 54, 55: Enhanced Composite Damage
Material Model 58: Laminated Composite Fabric
Material Model 59: Composite Failure Shell (Solid) Model
Material Model 114: Layered Linear Plasticity
Material Model 116: Composite Layup
Material Model 117: Composite Matrix
Material Model 118: Composite Direct
Material Model 161: Composite MSC

All of the above models deal specifically with orthotropic composite materials. Mechanical material properties are needed in the input file, such as Young’s modulus, Poisson’s ratio and shear modulus. For failure analysis, material strengths must be provided, such as longitudinal tensile strength, longitudinal compressive strength, transverse tensile strength, transverse compressive strength and shear strength. Each model has a material axes option to determine the locally orthotropic material axes. For those material models, the stress-strain curves are linear before damage occurs. To compare their differences, several most often used models will be described in the following sections with parametric studies.
4.1.1 Material Model 22

This Chang-Chang composite failure model is the first composite material model with failure implemented in LS-DYNA. It can be used in both shell and solid elements. Suggested by Chang and Chang\textsuperscript{54,69}, three failure modes are available:

Fiber breakage failure, \((\sigma_{11} > 0)\)

\[
\left( \frac{\sigma_{11}}{X_T} \right)^2 + \bar{\tau} = 1
\]  

(4.1)

Matrix cracking failure, \((\sigma_{22} > 0)\)

\[
\left( \frac{\sigma_{22}}{Y_T} \right)^2 + \bar{\tau} = 1
\]

(4.2)

Compression failure, \((\sigma_{22} < 0)\)

\[
\left( \frac{\sigma_{22}}{2X_S} \right)^2 + \left[ \left( \frac{Y_C}{2X_S} \right)^2 - 1 \right] \frac{\sigma_{22}}{Y_C} + \bar{\tau} = 1
\]

(4.3)

where

\[
\bar{\tau} = \frac{\sigma_{12}^2}{2G_{12}} + \frac{3}{4} \alpha \sigma_{12}^4
\]

\[
\frac{X_S^2}{2G_{12}} + \frac{3}{4} \alpha X_S^4
\]

(4.4)

In the above equations, \(\sigma_{ij}\) is the macroscopic stress component, \(X_T\) is the longitudinal tensile strength, \(Y_T\) and \(Y_C\) are the transverse tensile and compressive strength respectively and \(X_S\) is the in-plan shear strength. \(G_{12}\) is the in-plane shear modulus. \(\alpha\) is the nonlinear shear stress parameter, which is defined by the shear stress-shear strain
relationship in plane stress: The stress-strain relationship in normal directions is assumed to be linear.

\[ 2\varepsilon_{12} = \frac{1}{G_{12}}\sigma_{12} + \alpha\sigma_{12}^3 \]  \hspace{4cm} (4.5)

4.1.2 Material Model 54, 55

These are the enhanced versions of Material Model 22, but are only valid for thin shell elements. The material is assumed to be orthotropic and linear elastic when it is undamaged. Nonlinearity is introduced into the material model when damage occurs. Chang-Chang criteria\textsuperscript{54,69} as in Material 22 is used in Material 54 with added fiber compressive mode. It includes tensile and compressive fiber failure as well as tensile and compressive matrix failure as follows:

Fiber tensile mode, \((\sigma_{11} > 0)\)

\[ \left( \frac{\sigma_{11}}{X_T} \right)^2 + \bar{\tau} = 1 \]  \hspace{4cm} (4.6)

Fiber compressive mode \((\sigma_{11} < 0)\)

\[ \left( \frac{\sigma_{11}}{X_C} \right)^2 = 1 \]  \hspace{4cm} (4.7)

Matrix tensile mode \((\sigma_{22} > 0)\)

\[ \left( \frac{\sigma_{22}}{Y_T} \right)^2 + \bar{\tau} = 1 \]  \hspace{4cm} (4.8)
Matrix compressive mode \((\sigma_{22} < 0)\)

\[
\left( \frac{\sigma_{22}}{2X_s} \right)^2 + \left[ \frac{Y_c}{2X_s} \right]^2 - 1 \frac{\sigma_{22}}{Y_c} + \bar{r} = 1
\]  

(4.9)

where \(\bar{r}\) is as defined in Equation 4.4. \(X_c\) is the longitudinal compressive strength. The other parameters and variables have been explained in Section 4.1.1. When the nonlinear shear stress parameter \(\alpha\) is set to 0.0, the above four failure modes represent the original criteria of Hashin53.

In material model 55, the Tsay-Wu criteria70 is used, which replaces the tensile and compressive matrix modes in the Chang-Chang criteria with the following single expression:

\[
\frac{\sigma_{22}^2}{Y_c Y_T} + \left( \frac{\sigma_{12}}{X_S} \right)^2 + \left( \frac{Y_c - Y_T}{Y_c Y_T} \right) \sigma_{22}^2 = 1
\]  

(4.10)

When fiber tension criterion is met, all moduli and Poisson’s ratios are set to zero, that is \(E_1 = E_2 = G_{12} = \nu_{12} = \nu_{21} = 0\), where \(E_1\) is the longitudinal elastic modulus, \(E_2\) is the transverse elastic modulus, \(G_{12}\) is the shear modulus, \(\nu_{12}\) and \(\nu_{21}\) are the in-plane Poisson’s ratios. All the stresses are reduced to zero and the layer in the element is failed. For fiber compression mode, \(E_1 = \nu_{12} = \nu_{21} = 0\). For matrix tension mode, \(E_2 = G_{12} = \nu_{21} = 0\). For matrix compression mode, \(E_2 = G_{12} = \nu_{12} = \nu_{21} = 0\). For a brittle material, after matrix compression criterion is met, a softening factor is applied to reduce fiber tensile strength, and a reduction factor is used to reduce the compressive fiber strength.
Besides strengths, maximum strains for fiber tension and effective failure strain are required in the input parameters. The layer in the element is removed instantaneously when either of the strains is reached.

When failure occurs in all the composite layers (through thickness integration points), the element is deleted. Elements which share nodes with the deleted element become “crashfront” elements and their elastic moduli and strengths can be reduced by using a softening reduction factor. In this way, the sudden release of stress concentration caused by the element deletion is compensated so that a crushing process with a damage front can be simulated.

To better understand this composite model, some simple computations with a 4-noded shell element test problem are carried out, as seen in Figure 4.1. The material modulus and Poisson’s ratio are given as: $E_1=80\text{GPa}$, $E_2=15\text{GPa}$, $\nu_{12}=0.3$. The strengths in fiber direction are $X_f=800\text{MPa}$, $X_c=500\text{MPa}$, in transverse direction are $Y_f=45\text{MPa}$, $Y_c=80\text{MPa}$, and the strength in the shear direction is $X_s=100\text{MPa}$.

![Figure 4.1 Single element under tension](image-url)
First the element is loaded in the fiber direction with a constant strain rate of 1/s. As shown in Figure 4.2a, the element stress in the fiber direction increases linearly until it reaches the maximum value of 800MPa. Then all the elastic constants and consequently the stresses are reduced to zero in 100 time steps. The predetermined 100 time steps before the ultimate failure are used to ensure the dynamic stability of the model. The material exhibits a linear brittle behavior.

In the above example, the maximum strain for fiber tension was set to zero and not considered in the computation. The curve in Figure 4.2b has the same loading and material constants except that the maximum strain for fiber tension is set to 0.03. After reaching the maximum stress in the fiber direction, the stress remains constant until the maximum longitudinal strain of 0.03 is reached, then the element is deleted at this point instantaneously. Similar behavior can be found by applying the effective strain limit.
Figure 4.2 Tensile stress-strain curves in fiber direction
Next, compression is applied in the matrix transverse direction simultaneously with tension in the fiber direction, as shown in Figure 4.3. The applied compressive strain rate in the transverse direction is three times of the tensile strain rate in the fiber direction so that compressive failure occurs first in the transverse direction. As stated previously, in Material 54-55, the strengths in the fiber direction can be reduced after the matrix compressive failure. This is achieved by applying the deduction factor FBRT in the model for tensile strength and YCFAC for compressive strength reduction. With the FBRT parameter, the fiber strength is reduced to \( X_T^* = FBRT \times X_T \). In the example shown in Figure 4.4a, FBRT is set to 0.5, so that the strength in fiber direction after matrix compressive failure is reduced to \( X_T^* = 400 \text{ MPa} \). With YCFAC, the fiber compressive strength after matrix compressive failure becomes \( X_C^* = YCFAC \times Y_C \). As depicted in Figure 4.4b, by applying compression in both the fiber and matrix directions, and with YCFAC = 3.0, \( X_C \) is reduced to \( X_C^* = 240 \text{ MPa} \) after the matrix failure at -80MPa. It can be concluded from Figure 4.4 that, with the strength reduction factors, the yielding in fiber tension and fiber compression is coupled with the yielding in matrix compression.

![Figure 4.3 Single element under tension and compression](image-url)
Figure 4.4 Strength reduction in fiber direction after matrix compressive failure
Another point to note in Figure 4.4 is that, only after the fiber tensile failure, all the stresses are reduced to zero. When matrix compression and fiber compression modes being met, the stresses are kept constant. In Material 54-55, ultimate failure (stresses reduce to zero) can occur in only four different ways:

1) Chang-Chang failure criterion is satisfied in tensile fiber mode
2) Maximum fiber tensile strain is met
3) Maximum effective failure strain is met
4) Minimum time step is met

4.1.3 Material Model 58

The Laminated Composite Fabric model is only applicable for shell elements. This model is based on a plane-stress continuum damage mechanics model developed by Matzenmiller, Lubliner and Taylor (MLT)\(^{58}\). More detail on this LS-DYNA implemented MLT composite model can be found in Schweizerhof et al.\(^{71}\).

The MLT model assumes that the deformation introduces micro cracks and cavities into the material, which causes the stiffness degradation. The moduli are modified so that only the undamaged part of a cross section is accounted for. Three damage variables (\(\omega_1, \omega_2, \omega_3\)) are introduced to account for the effect of damage on the longitudinal, transverse and in-plane shear response respectively. The compliance relationship for the damaged material is defined by:

\[
\{\varepsilon\} = [H]\{\sigma\}
\]

where \(\{\varepsilon\} = \{\varepsilon_{11}, \varepsilon_{22}, 2\varepsilon_{12}\}^T\) are the strain components, and \(\{\sigma\} = \{\sigma_{11}, \sigma_{22}, \sigma_{12}\}^T\) are the
stress components, both in orthotropic material directions. \([H]\) is the compliance matrix for the damaged lamina defined by

\[
[H] = \begin{bmatrix}
\frac{1}{(1-\omega_{11})E_1} & -\frac{\nu_{12}}{E_1} & 0 \\
-\frac{\nu_{12}}{E_1} & \frac{1}{(1-\omega_{22})E_2} & 0 \\
0 & 0 & \frac{1}{(1-\omega_{12})G_{12}}
\end{bmatrix}
\] (4.12)

where \(E_1, E_2, G_{12}\) are the original elasticity constants of an undamaged lamina. Then \([H]\) is reversed to obtain the material stiffness matrix. The material constitutive equation becomes:

\[
\{\sigma\} = [H]^{-1}\{\varepsilon\}
\] (4.13)

Note that, the two damage variables \(\omega_{11}\) and \(\omega_{22}\) assume different values for tension (\(\omega_{11t}\) and \(\omega_{22t}\)) and compression (\(\omega_{11c}\) and \(\omega_{22c}\)) in order to account for the one-sidedness phenomenon observed in many materials; whereas the damage variable \(\omega_{12}\) is independent of the sign of the shear stress \(\sigma_{12}\). The evolution of the damage variable \(\omega\) is defined as:

\[
\omega = 1 - e^{-\frac{\varepsilon}{m\varepsilon_f}m}
\] (4.14)

where \(\varepsilon\) is the current strain in the corresponding damage direction, \(\varepsilon_f\) is the failure elastic strain that is calculated by dividing the strength \(\sigma_f\) with the corresponding initial modulus (\(E_1, E_2\) or \(G_{12}\)). \(m\) is the damage exponent corresponding to the failure mode which is calculated in Material 58 as:
where $\varepsilon_q$ is the strain at which the strength is reached. The relationship between $\varepsilon_f, \varepsilon_q$ and $\sigma_f$ is shown in Figure 4.5, where a typical stress-strain curve simulated by Material 58 for a uniaxial loading is demonstrated. Each of the five damage variables, $\omega_{1w}$, $\omega_{2w}$, $\omega_{1c}$, $\omega_{2c}$ and $\omega_2$, is associated with one exponent $m$, and the related $\sigma_f$ is assumed to be identical to $X_T$, $X_C$, $Y_T$, $Y_C$ and $X_S$ respectively. By defining the strains $\varepsilon_q$ at strengths for Material 58, users indirectly determine the values for $m$.

\[
m = \frac{1}{\ln(\varepsilon_q/\varepsilon_f)} \quad (4.15)
\]

Figure 4.5 Schematic demonstration of a typical stress-strain curve simulated by Material 58

It is assumed in the MLT model that the state of damage in the material will not change within a certain region in the stress space (or strain space). Therefore the damage variables are monotonically increasing values ranging from 0.0 to 1.0. It can be seen
from Equation 4.11 that, with the application of the damage variables, the nonlinearity in the composite deformation is captured by the MLT model.

To better illustrate the effect of the damage exponent $m$, simple tension in fiber direction is applied to a single shell element as shown in Figure 4.1. Same material elastic properties and strengths were used here as in the example for Material 54-55. The elastic failure strain $\varepsilon_f$ in the longitudinal fiber direction can be calculated as $\varepsilon_f = X_F / E_1 = 0.01$. Three tests were demonstrated in this example with the strain at the longitudinal tensile strength $\varepsilon_q$ set to $0.01\varepsilon$, $0.01\varepsilon^{0.5}$ and $0.01\varepsilon^{0.25}$ respectively. Therefore, according to Equation 4.14, $m$ had the corresponding value of 1, 2, 4 respectively. The evolution of damage variable $\omega$ with different exponent $m$ is shown in Figure 4.6. It can be seen from the figure that a larger $m$ results in a faster damage growth. The effect of damage exponent $m$ on the tensile stress-strain curves is shown in Figure 4.7. The value of $m$ controls the shape of the stress-strain curve over the entire strain range as shown. As $m$ decreases, not only does the strain softening behavior become more gradual but also the nonlinearity increases before yielding occurs. It should be noted that in Material 58 all nonlinearity is assumed to be due to damage.
Figure 4.6 Evolution of damage variable $\omega$ with different exponent $m$

Figure 4.7 Effect of exponent $m$ on the stress-strain curve in Material 58
The Hashin failure criteria\textsuperscript{53} are used in Material 58 with variations for different types of composites. For the unidirectional layered composites, the failure criteria are given as:

Fiber tensile mode ($\hat{\sigma}_{11}$ > 0)

$$\left(\frac{\hat{\sigma}_{11}}{X_T}\right)^2 = 1$$  \hspace{1cm} (4.16)

Fiber compressive mode ($\hat{\sigma}_{11}$ < 0)

$$\left(\frac{\hat{\sigma}_{11}}{X_C}\right)^2 = 1$$  \hspace{1cm} (4.17)

Matrix tensile mode ($\hat{\sigma}_{22}$ > 0)

$$\left(\frac{\hat{\sigma}_{22}}{Y_T}\right)^2 + \left(\frac{\hat{\sigma}_{12}}{X_S}\right)^2 = 1$$  \hspace{1cm} (4.18)

Matrix compressive mode ($\hat{\sigma}_{22}$ < 0)

$$\left(\frac{\hat{\sigma}_{22}}{Y_C}\right)^2 + \left(\frac{\hat{\sigma}_{12}}{X_S}\right)^2 = 1$$  \hspace{1cm} (4.19)

where $\hat{\sigma}_{ij}$ are the effective stress components defined by

$$\begin{pmatrix}
\hat{\sigma}_{11} \\
\hat{\sigma}_{22} \\
\hat{\sigma}_{12}
\end{pmatrix} = \begin{bmatrix}
1 & 0 & 0 \\
1-\omega_{11} & 1 & 0 \\
0 & 1-\omega_{22} & 1 \\
0 & 0 & 1-\omega_{12}
\end{bmatrix}\begin{pmatrix}
\sigma_{11} \\
\sigma_{22} \\
\sigma_{12}
\end{pmatrix}$$  \hspace{1cm} (4.20)

For complete laminates and fabrics, quadratic failure criteria are also used for fiber modes as for matrix modes, which result in a smooth failure surface. The failure
criteria in matrix modes remain the same and the criteria in fiber modes are changed to:

Fiber tensile mode ($\sigma_{11} > 0$)

$$\left( \frac{\sigma_{11}}{X_T} \right)^2 + \left( \frac{\sigma_{12}}{X_S} \right)^2 = 1$$ (4.21)

Fiber compressive mode ($\sigma_{11} < 0$)

$$\left( \frac{\sigma_{11}}{X_C} \right)^2 + \left( \frac{\sigma_{12}}{X_S} \right)^2 = 1$$ (4.22)

In order to allow for an almost uncoupled failure of an arbitrary composite, failure criteria with non-smooth failure surfaces are available, in which all failure criteria are taken to be independent of each other. The failure criteria in fiber modes are as Equation 4.16 and 4.17, but in matrix modes the criteria change to:

Matrix tensile mode ($\sigma_{22} > 0$)

$$\left( \frac{\sigma_{22}}{Y_T} \right)^2 = 1$$ (4.23)

Matrix compressive mode ($\sigma_{22} < 0$)

$$\left( \frac{\sigma_{22}}{Y_C} \right)^2 = 1$$ (4.24)

And failure in shear is added as:

$$\left( \frac{\sigma_{12}}{X_S} \right)^2 = 1$$ (4.25)

The maximum effective strain is also applicable for failure in the element layer for all types of composites.
To avoid localization, stress limits are used in Material 58 to modify the damage evolution law after failure criteria have been met, such that the stress does not fall below a threshold value. The damage variable is modified as:

$$\omega = 1 - \frac{\alpha \sigma_f}{E}\quad (4.26)$$

where \(\alpha\) is the stress limit parameter associated with each strength value \(\sigma_f\) which is identical to \(X_T, X_C, Y_T, Y_C\) and \(X_S\) respectively for each damage mode. \(E\) is the elastic modulus equal to \(E_1, E_2,\) or \(G_{12}\) accordingly. The stress limits \(\alpha\) determine the minimum stress values after the stress hits the maximum. The values of \(\alpha\) should be between 0.0 and 1.0. With a value of 0.0, the damage evolution law is unchanged and the stress will be reduced to zero after the maximum value is reached. With a value of 1.0, the stress remains at a maximum value identical to the strength. To give an example, the effect of the stress limit for fiber tension on the stress-strain curve is shown in Figure 4.8. \(m=4\) is used in this example. It can be observed that the element stress remains constant at \(\alpha \times X_T\) after passing the limit value \(X_T\). For instance, with \(\alpha = 1.0\), the stress keeps constant at the maximum stress, while with \(\alpha = 0.0\) the stress finally reduces to zero. The corresponding evolution of damage variable \(\omega\) with different stress limit parameter \(\alpha\) is plotted in Figure 4.9, showing that a larger \(\alpha\) results in an earlier change in the path of the damage evolution. With \(\alpha = 0.0\), the evolution of \(\omega\) keeps its original path.
Figure 4.8 Effect of stress limit parameter $\alpha$

Figure 4.9 Evolution of damage variable $\omega$ with different stress limit parameter $\alpha$
4.1.4 Material Model 161, 162

The Composite MSC models are recently developed by Material Sciences Corporation\textsuperscript{19}. They are rate dependent models applicable only for solid elements. The material deformation behavior is linear before the failure criterion is met. There are two sets of failure criteria, one for a unidirectional layer model and the other for a fabric layer model. Delamination is considered in both models.

In the unidirectional layer model, the Hashin failure criteria\textsuperscript{53} in fiber modes are generalized to characterize the fiber damage in terms of strain components. Three damage functions in fiber mode are used, including fiber tension/shear, fiber compression and fiber crush under pressure. Matrix mode failures must occur without fiber failure, hence they will be on planes parallel to fibers. The matrix failure criteria include tension/shear failure (or transverse matrix failure) and compression/shear failure (or delamination).

For the fabric layer model, the fill and warp fiber tensile/shear failure criteria are given by the quadratic interaction between the associated axial and shear stresses. The in-plane compressive failure in both directions is given by the maximum stress criterion. In addition, there is crush failure under compressive pressure. The matrix failure modes include in-plane shear and delamination. A plain weave layer can fail under in-plane shear stress without fiber breakage. The delamination mode is associated with the through-thickness tensile strength and transverse shear strengths.

The difference between Material 161 and 162 lies in the post failure behavior. In Material 161, the failure is instantaneous. When fiber failure in tension/shear mode occurs in a layer, the load carrying capability of that layer is completely eliminated, and
all the stress components are reduced to zero instantaneously (with 100 time steps to avoid numerical instability). For compressive fiber failure, the layer is assumed to carry a residual axial load, while the load carrying capability in the transverse direction is reduced to zero. When the fiber crush failure occurs, the material is assumed to behave elastically for compressive pressure and to carry no load for tensile pressure. When the matrix fails, the associated shear strengths are reduced to zero.

Similar to Material 58, Material 162 adopts the MLT\textsuperscript{58} damage mechanics approach for characterizing the softening behavior after damage initiation. A set of damage variables $\omega_i$ ($i=1,\ldots,6$) are introduced to relate the onset and growth of damage to stiffness losses in the material. The initial elastic moduli $E_{i0}$ and shear moduli $G_{i0}$ are modified by the damage variables as

\begin{align}
E_i &= (1 - \omega_i)E_{i0} \\
G_i &= (1 - \omega_i)G_{i0}
\end{align}

(4.27)

The damage variables $\omega_i$ are calculated as

$$\omega_j = 1 - e^{-m(l-r_j^n)}, \quad r_j \geq 1$$

(4.28)

where $r_j$ is the damage threshold related to the damage mode $j$, and $m$ is the material damage parameter.

Strain rate effects are accounted for in the strengths and elastic moduli by the following equations, respectively, as:

$$\{S_{\text{rate}}\} = \{S_0\} \left(1 + C_1 \ln \frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right)$$

(4.29)
where
\[
\{S_{rate}\} = \begin{bmatrix}
X_T \\
X_C \\
Y_T \\
Y_C \\
X_{FC} \\
X_{FS}
\end{bmatrix}, \quad \text{and} \quad \|\dot{\varepsilon}\| = \begin{bmatrix}
|\dot{\varepsilon}_{11}| \\
|\dot{\varepsilon}_{12}| \\
|\dot{\varepsilon}_{22}| \\
|\dot{\varepsilon}_{33}|
\end{bmatrix} \sqrt{\dot{\varepsilon}_{11}^2 + \dot{\varepsilon}_{22}^2}
\]

\[
\{E_{rate}\} = \{E_0\} \left(1 + \{C\} \ln \frac{\|\dot{\varepsilon}\|}{\dot{\varepsilon}_0} \right)
\] (4.30)

where
\[
\{E_{rate}\} = \begin{bmatrix}
E_1 \\
E_2 \\
E_3 \\
G_{12} \\
G_{23} \\
G_{13}
\end{bmatrix}, \quad \{C\} = \begin{bmatrix}
C_1 \\
C_2 \\
C_4 \\
C_3 \\
C_3 \\
C_3
\end{bmatrix}, \quad \text{and} \quad \|\dot{\varepsilon}\| = \begin{bmatrix}
|\dot{\varepsilon}_{11}| \\
|\dot{\varepsilon}_{22}| \\
|\dot{\varepsilon}_{33}|
\end{bmatrix} \begin{bmatrix}
|\dot{\varepsilon}_{12}| \\
|\dot{\varepsilon}_{23}| \\
|\dot{\varepsilon}_{13}|
\end{bmatrix}
\]

In the above two equations, \(C_1, C_2, C_3\) and \(C_4\) are the strain rate constants for strengths, axial modulus, shear moduli and transverse moduli, respectively. \(X_{FC}\) is the fiber crush strength, \(X_{FS}\) is the fiber shear strength. \(\{S_0\}\) and \(\{E_0\}\) are the strength and modulus values of \(\{S_{rate}\}\) and \(\{E_{rate}\}\), respectively at the reference strain rate \(\dot{\varepsilon}_0 = 1/s\).

To better understand the strain rate dependent model, parametric studies were carried out on a single 8-node solid element representing a unidirectional composite layer. As shown in Figure 4.10, the solid element is fixed on one side, and pulled on the other along the fiber direction at a constant strain rate. The material modulus and Poisson’s ratio were given as: \(E_1 = 35\text{GPa}, E_2 = 15\text{GPa}, \nu_{12} = 0.3\). The longitudinal tensile strength \(X_T = 850\text{MPa}\), longitudinal compressive strength \(X_C = 500\text{MPa}\), transverse tensile strength \(Y_T = 45\text{MPa}\), transverse compressive strength \(Y_C = 80\text{MPa}\), and in-plane
shear strength \( Y_s = 100 \text{MPa} \). The following tests were done with MAT162 since it is a more advanced model than MAT161.

To study the strain rate effect on the material strength, the strain rate constant for strength \( C_1 \) was set to 0.11, and the strain rate constants for the moduli were set to zero. Single element tests were carried out with five strain rates, 1.0/s, 2.0/s, 4.0/s, 8.0/s and 12.0/s. The results are shown in Figure 4.11. At a strain rate of 1.0/s, which is equal to the reference strain rate, the stress reaches the maximum at 850 MPa which is the material longitudinal tensile strength, and then the stress curve goes down to zero. With increasing strain rate, the maximum stress increases, giving the material higher strength. Material 161 will have the same behavior in the strain rate effect on material strengths.

To study the strain rate effect on the material modulus, \( C_1 \) was set to zero to keep a constant material strength under various strain rates; the strain rate constants for shear and transverse moduli \( C_3 \) and \( C_4 \) were set to zero to avoid their influence and the strain rate constant for axial modulus \( C_2 \) was set to 0.06. The same set of five strain rates was tested. As shown in Figure 4.12, the strength remains constant, while the axial modulus

Figure 4.10 Single 8-node solid element under tension

To study the strain rate effect on the material strength, the strain rate constant for strength \( C_1 \) was set to 0.11, and the strain rate constants for the moduli were set to zero. Single element tests were carried out with five strain rates, 1.0/s, 2.0/s, 4.0/s, 8.0/s and 12.0/s. The results are shown in Figure 4.11. At a strain rate of 1.0/s, which is equal to the reference strain rate, the stress reaches the maximum at 850 MPa which is the material longitudinal tensile strength, and then the stress curve goes down to zero. With increasing strain rate, the maximum stress increases, giving the material higher strength. Material 161 will have the same behavior in the strain rate effect on material strengths.

To study the strain rate effect on the material modulus, \( C_1 \) was set to zero to keep a constant material strength under various strain rates; the strain rate constants for shear and transverse moduli \( C_3 \) and \( C_4 \) were set to zero to avoid their influence and the strain rate constant for axial modulus \( C_2 \) was set to 0.06. The same set of five strain rates was tested. As shown in Figure 4.12, the strength remains constant, while the axial modulus
increases with increasing strain rate. In Material 161, this option is not available and the material moduli will not change with strain rates.

Figure 4.11 Strain rate effect on material strength

Figure 4.12 Strain rate effect on material modulus
Unlike in Material 58 where the damage variables influence the stress-strain curves over the entire strain range, in Material 162 the damage variables are only applicable to the post-failure part. Figure 4.13 shows the effect of $m$ with the single solid element test under tension in fiber direction. It can be seen that the deformation behavior in the loading path is unchanged with various damage exponent $m$. After the maximum stress is reached, the damage progresses at a faster speed with a larger $m$, representing a more brittle material.

![Figure 4.13 Influence of damage parameter $m$](image)

As in Material 58, in Material 162 there is a limit damage parameter for elastic modulus reduction. With this parameter, the material modulus will not be reduced to zero after damage occurs, but limited to a percentage of its original value. Consequently, the material has the capability to carry a small amount of stresses. Figure 4.14 shows the effect of the limit damage parameter. Note that the value of the parameter has to be very close to 1.0 to ensure stability.
4.1.5 Summary on the current composite models

Several representative composite models currently available in LS-DYNA have been studied in the previous sections. The common features of these models include:

(1) The deformation behavior before failure is linear elastic, except for Material 58 in which the reductions in the elastic moduli introduce nonlinearity in the deformation behavior. But the nonlinearity in the loading path in Material 58 is coupled with its post-failure behavior, which restricted its versatility in composite modeling. For Material 22, 54 and 55, nonlinearity is considered in shear with a nonlinear term added to the shear strain, while the deformation in the normal directions is linear.

(2) Strain rate dependence is not considered. One exception is Material 161-162 which added strain rate dependence in strengths and initial moduli by scale factors. But this material model is only applicable for solid elements and the deformation behavior
before failure is linear elastic.

The damage criteria in these composite models take the forms of Chang-Chang\textsuperscript{54,69} or Hashin\textsuperscript{53} criteria with variations. These criteria are failure-mode-based which have gained favor in the composite research community.

The post-failure behavior of the composite models takes several forms, as shown in Figure 4.15. After reaching the strength value $\sigma_f$, instantaneous failure is applied by immediately reducing the stress to zero, as path (1) in the figure. Such brittle failure behavior is often used for the fiber breakage mode. If the stress remains constant after hitting the maximum until maximum strain criterion is met, a elastic-plastic deformation behavior is simulated, as path (2) in the figure. This strategy is usually used for damage in matrix modes and fiber compression mode. The third method is to add strain softening part after the onset of failure. The stress gradually diminishes along path (3) by degrading the elastic properties through certain damage growth law. This third approach has been adopted by Material 58 and Material 162. The damage progression in Material 162 is applicable only to the post-failure part; whereas in Material 58 the damage is progressing through the entire strain range.

As has been discussed in Chapter II, under circumstances where the strain rate is high, such as in an impact problem, the nonlinear, strain rate dependent deformation behavior of composites becomes important for a proper simulation. For such cases, the currently available elastic composite models in LS-DYNA can not adequately capture the complicated deformation behavior of composites. Though Material 162 has rate-dependence in elastic moduli and strengths, it is basically a linear elastic model and only
applicable to solid elements. For typically thin-structured composite material, the use of shell elements is more physical and computational efficient.

Based on the above discussions, a more versatile composite model is desired to capture the nonlinear, strain rate dependent deformation behavior of composites. Progressive damage model is also preferred to provide users with better control over the post-failure behavior of composites. The new composite model would be applicable to shell elements due to the reasons discussed in the last paragraph.

The following section will explain the procedure on how to develop a user defined material model in LS-DYNA.

4.1.6 Modeling multi-layer composites with shell elements

The above tests were performed on one single layer of composite with a single shell element. This approach was reasonable since, in a macroscopic manner, composite lay-ups can be simulated with one layer of an anisotropic material. All the through-
thickness integration points are assigned the same material definition. This approach is sometimes convenient, since in many practical applications the material properties are measured using the whole lay-up. The whole composite is considered to be failed if the criteria are fulfilled in one integration point throughout the thickness.

The other, more physical, way to simulate multi-layer composites is to assign different material definitions to each through-thickness integration point. The simplest way to do this is to define the material angle for each through-thickness integration point in the composite laminate. Then the stresses and strain increments will be transformed from global coordinate system into the material coordinate system defined by this angle. A user defined integration rule should also be used to control the layer thickness. Each layer can have more than one integration point. By assigning different angles to the through-thickness integration points in one layer in a systematic way, braided or woven composite laminates can be simulated. This braiding/weaving with through-thickness integration points methodology will be further explained with examples in Chapter V.

4.2 Procedure of UMAT development

As has been stated in Section 3.6, material models are called for at stage 2 when the strain increment has been calculated at the through-thickness integration point. The function of the material models is to output the stresses at this integration point.

LS-DYNA has an option that allows users to define their own material model, which is called a UMAT. This section will explain in detail the procedure on how to develop a UMAT in LS-DYNA.
4.2.1 Structure of the UMAT source code

The UMAT source code in FORTRAN provides the entry to the LS-DYNA main code by which the UMAT subroutine is linked to and thus can be called by the main program. It is composed of subroutines for each element type including shell element, beam element, discrete beam element and solid element.

The subroutine starts with several common blocks to get access to the parameters and variables in the main program. Then the Young’s modulus and Poisson’s ratio of the material are calculated from the bulk modulus and shear modulus obtained from the input deck. These values are used to calculate the time step sizes.

For orthotropic/anisotropic layered composite model, the accumulated stresses and incremental strains are transformed from the global to local coordinates by the material angle defined for each integration point. Then the subroutine loops over each element and calls for the user developed subroutines. Note that, in LS-DYNA, elements are divided into blocks during processing. The number of elements in a block is optimized to obtain a most time efficient computation. Each element in the block is assigned an internal element ID. The current optimum block size in LS-DYNA is 89. To better illustrate the calculation sequence, a simple example is given for 100 elements and each with 3 through-thickness integration points. LS-DYNA will process the calculation in the sequence as shown in Figure 4.16 during one time step. The physical element ID is called external element ID. In the example in Figure 4.16, the external element ID ranges from 1 to 100, and the internal ID repeats from 1 to 89. UMAT subroutine is called for on each integration point of each element. Here in this case it has been called for 300 times in one time step calculation.
Before entering the UMAT subroutine, the strain increments, stresses and history variables from the previous time step are input to the UMAT subroutines. All these data are in the local coordinate system. The function of the UMAT subroutine is to calculate the new stresses at time $t + \Delta t$ through the material constitutive laws, to check failures with failure criteria, then to output stresses and update the history variables. For shell element, the strain increment in the thickness direction needs to be calculated and delivered to the next time step. The strain increments and stresses will be then transformed back to the global coordinate system after exiting the subroutine.
Figure 4.16 Element processing sequence
4.2.2 Modification of the UMAT subroutine

In the UMAT subroutine, all the material constants, such as the elastic moduli, Poisson’s ratios, strengths etc., are saved in one array. They are read from the input deck. Users can currently define up to 40 material constants. Among these 40 constants, there are two required constants, the bulk modulus and the shear modulus. These two moduli are mainly used to calculate the time step size as has been stated in Section 3.2.2. They are also required for transmitting boundaries, contact interfaces, rigid body constraints. Besides these 40 material constants, LS-DYNA keeps 8 more positions to save the values for defining the coordinate system.

The local engineering strain increments and local engineering stresses are input to the subroutine from the source code. The strain increments can be converted to strain rates by being divided by the current time step. For shell elements, the strain increment in thickness direction must be determined at the end of the subroutine to account for the change in the thickness. The input stresses are essentially stresses at time $t$, and stresses at time $t + \Delta t$ need to be calculated before return.

History variables are stored in another array. The history variables are basically arrays of numbers that are stored between time steps, so upon the next re-entry to the routine the information can be retrieved. They can be used to save and transmit variables such as inelastic strain increments, damage variables etc..

In summary, the input to the UMAT subroutine include: 1) the material constants obtained from the input deck, 2) the strain increments, stresses and history variables at time $t$ from the source code. Users need to construct their own material constitutive equations based on these input. The expected outputs from the equations are the updated
stresses and history variables at time $t + \Delta t$. For shell element, the through-thickness strain increment also needs to be calculated.

In addition, users can define failure criteria and flag elements which have failed so that they can be deleted from the simulation. In order to do so, some common blocks need to be added to the UMAT subroutine.

Following the above described procedure, the polymer and composite models described in Chapter II were implemented into LS-DYNA as two independent UMATs. The verification studies on these UMATs will be carried out in Chapter V.
CHAPTER V

MATERIAL MODEL VERIFICATION IN LS-DYNA

5.1 Simulation of strain rate dependent polymer deformation

To verify the implemented polymer constitutive UMAT, the deformation behavior of several types of polymers under a wide range of strain rate loading is examined and compared with experimental results. The studied polymers include PR520 and 977-2 toughened epoxies, a standard brittle epoxy E-862, and the thermoplastic Poly ether ether ketone (PEEK). Then the hydrostatic stress effect and improved unloading behavior were examined with PEEK. The characterized 977-2 and PEEK will be used later in the composite UMAT simulations.

5.1.1 Single element model for UMAT testing

In the finite element analyses, a four-node single Belytschko-Tsay shell element\textsuperscript{18} was used for the UMAT examination. The load and boundary conditions applied to the model are shown in Figure 5.1. The applied constant strain rates were defined by linear
displacement-time curves in the LS-DYNA input file. Table 1 lists the material constants used in the input for each polymer.

![Boundary and loading conditions for single element testing](image)

**Figure 5.1** Boundary and loading conditions for single element testing

**Table 1** Material constants for polymer modeling

<table>
<thead>
<tr>
<th>Material</th>
<th>$E_0$ (GPa)</th>
<th>$C$</th>
<th>$V_m$</th>
<th>$D_0$ (1/s)</th>
<th>$n$</th>
<th>$Z_0$ (MPa)</th>
<th>$Z_1$ (MPa)</th>
<th>$q$</th>
<th>$\alpha_0$</th>
<th>$\alpha_1$</th>
</tr>
</thead>
<tbody>
<tr>
<td>PR520</td>
<td>3.24</td>
<td>0.195</td>
<td>0.38</td>
<td>$1\times10^6$</td>
<td>0.93</td>
<td>396.1</td>
<td>753.8</td>
<td>279.3</td>
<td>0.568</td>
<td>0.13</td>
</tr>
<tr>
<td>977-2</td>
<td>3.20</td>
<td>0.166</td>
<td>0.40</td>
<td>$1\times10^6$</td>
<td>0.85</td>
<td>259.5</td>
<td>1131.4</td>
<td>150.5</td>
<td>0.129</td>
<td>0.15</td>
</tr>
<tr>
<td>E-862</td>
<td>2.93</td>
<td>0.278</td>
<td>0.38</td>
<td>$1\times10^6$</td>
<td>0.80</td>
<td>420.0</td>
<td>820.0</td>
<td>120.0</td>
<td>0.15</td>
<td>0.06</td>
</tr>
<tr>
<td>PEEK</td>
<td>4.50</td>
<td>0.0</td>
<td>0.40</td>
<td>$1\times10^6$</td>
<td>1.25</td>
<td>250.0</td>
<td>550.0</td>
<td>190.0</td>
<td>0.04</td>
<td>0.087</td>
</tr>
</tbody>
</table>

5.1.2 PR520 and 977-2 toughened epoxies

Two representative toughened epoxies, PR520 and 977-2, were analyzed using the polymer UMAT with LS-DYNA, and the computed stress-strain curves were compared to experimental results obtained by Goldberg et al.\textsuperscript{72}. In the experimental tests,
longitudinal tensile tests and pure shear tests were conducted at room temperature on the materials at strain rates of about $5.0 \times 10^{-4}$/s, 1/s and 400/s. The low and moderate strain rate tests were conducted using an Instron hydraulic testing machine. The high strain rate tests were conducted using a split Hopkinson bar. Engineering stress and engineering strain were measured until failure.

Predicted and experimental shear stress-shear strain curves for PR520 are shown in Figure 5.2 for the three strain rates examined, while tensile stress-strain curves are shown in Figure 5.3. Predicted and experimental shear stress-shear strain curves for 977-2 are shown in Figure 5.4, and tensile stress-strain curves are shown in Figure 5.5. Both materials exhibit a strain rate dependent, nonlinear deformation response under both shear and tensile loading. As noted by Goldberg et al., for the experimental data, at high strain rates, the sharp increase in stress at the beginning of the loading with negligible increase in strain observed for PR520 under shear and tensile loading was most likely not representative of the actual material behavior, but rather an artifact of the experimental tests. Also note that the oscillations observed in the high strain rate tensile tests for 977-2 were a result of the specimen geometry that was utilized for these tests. One more important point to note is that the LS-DYNA simulated stress-strain curves have been artificially truncated at the experimental failure points since failure is not considered in the polymer UMAT.

From Figures 5.2-5.5, it can be seen that, though there is some underestimation of the tensile stresses, particularly in the case of PR520 under high strain rate tensile loading, the finite element results correlate well with the experimental data. The nonlinearity and rate dependence observed in the deformation of the two polymers are
captured qualitatively, and the quantitative match between the experimental and computed results is reasonably good.

The effects of strain rate on the polymer deformation behavior are examined further in Figure 5.6, where the shear deformation of PR520 is computed at various strain rates. In the results presented in Figure 5.6(a) a fixed initial elastic modulus \( C=0 \) is used, while in Figure 5.6(b), the initial elastic modulus is allowed to vary with strain rate as described in Equation 2.15. Both plots show that with increasing strain rate, the overall material stiffness increases and the stress level at which the stress-strain curve flattens out increases. However, by allowing the elastic modulus to vary with strain rate, the initial stiffness of the material response increases with increasing strain rate, which Figures 5.2-5.5 indicate is more representative of the actual polymer behavior.

Figure 5.2 Experimental and LS-DYNA predicted shear stress-shear strain curves for PR520 resin at strain rates of 420/s, 1.76/s and \( 7.0 \times 10^{-5} /s \)
Figure 5.3 Experimental and LS-DYNA predicted tensile stress-strain curves for PR520 resin at strain rates of 510/s, 1.4/s and $5.0 \times 10^{-5}$/s
Figure 5.4 Experimental and LS-DYNA predicted shear stress-shear strain curves for 977-2 resin at strain rates of 518/s, 1.91/s and $9 \times 10^{-5}$/s
Figure 5.5 Experimental and LS-DYNA predicted tensile stress-strain curves for 977-2 resin at strain rates of 365/s, 1.31/s and $5.7 \times 10^{-5}$/s
Figure 5.6 Strain rate effect on polymer deformation behavior

(a) Without initial elastic modification

(b) With initial elastic modification
5.1.3 E-862 standard brittle epoxy

A standard brittle epoxy, E-862, was examined with the polymer UMAT and compared with the experimental results\(^ {73}\). The experimental tests were carried out with similar test methods and conditions as those for PR520 and 977-2. Predicted and experimental\(^ {73}\) shear stress-shear strain curves for E-862 are shown in Figure 5.7 and the tensile stress-strain curves are shown in Figure 5.8. It can be seen from the figures that the predicted results correlate well with the experimental values for all strain rates for both shear and tensile tests.

![Figure 5.7 Experimental and LS-DYNA predicted shear stress-strain curves for E-862 resin at strain rates of 404/s, 1.5/s and 7.5×10⁻⁵/s](image-url)

Figure 5.7 Experimental and LS-DYNA predicted shear stress-strain curves for E-862 resin at strain rates of 404/s, 1.5/s and 7.5×10⁻⁵/s
After having demonstrated satisfactory predicting capability with several thermoset epoxies, the polymer UMAT was further examined with the thermoplastic PEEK and compared with the experimental results obtained by Hsu et al.\textsuperscript{16} and Bordonaro\textsuperscript{33}. A slightly larger elastic modulus in compression than that in tension was observed in the experimental results. The difference in the modulus may be due to the different sample and experimental setup, since the tensile and compressive tests data were from two independent works\textsuperscript{16,33}. Another reason may lies in the hydrostatic stress effect on the initial modulus as have been argued by Spitzig and Richmond\textsuperscript{25} with
Polyethylene and Polycarbonate. This hydrostatic stress effect on the initial elastic modulus is not considered in the polymer model since this effect is not significant, unlike its effect on the maximum stresses. In the material characterization for PEEK in the simulation, elastic modulus obtained from the compression tests is used in the prediction for both tension and compression.

Experimental\textsuperscript{16} and predicted compressive stress-strain curves for PEEK under four low strain rates are shown in Figure 5.9. Experimental\textsuperscript{33} and predicted tensile stress-strain curves for PEEK under three low strain rates are shown in Figure 5.10. Both the predicted tensile and compressive stress-strain curves capture the nonlinearity and rate dependence of the experimental results, and the quantitative comparison between the experimental and predicted results is very good. Note that the initial elastic modulus remained constant for all the strain rates examined. This is due to the fact that the testing strain rates in this example were all under 1/s. As stated in Chapter II, the polymer initial elastic modulus is not rate sensitive at low strain rates. This example also justified the choice of 1/s as the reference strain rate for Equation 2.15.

The compressive and tensile stress-strain curves at strain rate of $1 \times 10^{-3}$/s are plotted in Figure 5.11. The curves of compression were inversed and superimposed onto those of tension. It can be seen from the figure that, with the application of $\alpha$, the stress is significantly larger in compressive loading than in tensile loading and the predicted results correlates well with the experimental results in both tension and compression. However, without $\alpha$ modification, the compressive and tensile response of the polymer can not be differentiated; indicating the hydrostatic stress effect which is typical in polymers is not accounted for.
The characterized PEEK will be used later for the prediction with the composite UMAT.

Figure 5.9 Experimental and LS-DYNA predicted compressive stress-strain curves for PEEK resin at strain rates of 0.1/s, 0.01/s, $1 \times 10^{-3}$/s and $1 \times 10^{-5}$/s
Figure 5.10 Experimental and LS-DYNA predicted tensile stress-strain curves for PEEK resin at strain rates of $1 \times 10^{-3}$/s, $1 \times 10^{-4}$/s and $1 \times 10^{-6}$/s
5.1.5 Improved unloading behavior with polymer damage model

The loading-unloading behavior of thermoplastic PEEK was examined with the polymer damage model and compared with the experimental results obtained by Bordonaro. A constant tensile strain rate of $1 \times 10^{-3}$/s was applied to the test sample up to a longitudinal strain of 4.4%, and then the sample was unloaded at the same rate. The damage parameter $C_1$ as in Equation 2.20 was assumed to be 32.0 in this example. Figure 5.12 shows the comparison of the loading-unloading stress-strain curves obtained by the original Goldberg’s polymer model ($C_1 = 0.0$) and the damage model ($C_1 = 32.0$) with the experimental results. It can be seen from the figure that the original Goldberg’s model predicts the unloading too rapid and leaves a large inelastic strain. The elastic
modulus at unloading remains the same as the initial elastic modulus. With damage model applied to the elastic modulus, the unloading elastic modulus is obviously smaller than the initial elastic modulus and the remaining inelastic strain is greatly reduced. With a damage parameter $C_1 = 32.0$ the stress-strain curve in the unloading path correlates well with the experimental results. This demonstrates that it is possible to improve the unloading prediction with the polymer damage model. However, the damage evolution law needs to be further explored with loading-unloading experiments.

Figure 5.12 Comparison of the unloading behavior between the original Goldberg’s polymer model and the modified model with damage

One important point to note is that, with the reduced elastic modulus, there is a slight decrease in the predicted stress in the loading path. Therefore, in the characterization of the polymer deformation, such effect resulted from the application of
damage needs to be watched to make sure the decrease amount is in an acceptable small range. Due to the incremental form of the stress calculation, the amount in the stress decrease is generally kept small, especially for the small strain.

5.2 Simulation of strain rate dependent composite deformation

Predictions of the composite UMAT were carried out with two types of fiber reinforced polymer matrix composites, namely IM7/977-2 and AS4/PEEK. Both off-axis and balanced ply-angle configurations were examined with various strain rates in LS-DYNA and compared with the experimental results. Failure envelopes at two strain rates were also predicted and compared with experimental observations for unidirectional IM7/977-2 composites of various fiber orientations. The material properties for the two types of carbon fibers, IM7 and AS4, are listed in Table 2.

Table 2 Mechanical properties for fibers

<table>
<thead>
<tr>
<th></th>
<th>$E_{11}$(GPa)</th>
<th>$E_{22}$(GPa)</th>
<th>$G_{12}$(GPa)</th>
<th>$\nu_{12}$</th>
<th>$\nu_{23}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>IM7</td>
<td>276</td>
<td>13.8</td>
<td>20.0</td>
<td>0.25</td>
<td>0.25</td>
</tr>
<tr>
<td>AS4</td>
<td>214</td>
<td>14</td>
<td>28</td>
<td>0.20</td>
<td>0.25</td>
</tr>
</tbody>
</table>

5.2.1 Off-axis IM7/977-2 predictions

This composite material consists of carbon IM7 fibers with a fiber volume ratio of 0.60 in the 977-2 toughened epoxy matrix discussed earlier. The IM7 fiber properties can be found in Table 2. The same properties for the 977-2 material as defined in Table 1 are used for the matrix phase in the composite model. The loading and boundary conditions used for the composite UMAT verification studies are identical to those used for the
polymer UMAT. The fiber orientation of each lamina was defined by the through-thickness integration points of the shell element in the LS-DYNA input deck.

Experimental\textsuperscript{73} and predicted longitudinal tensile stress-strain curves for three laminate configurations ([10°], [45°] and [90°]) of the IM7/977-2 material at strain rates of approximately 1 /s and 400 /s are shown in Figures 5.13-5.15. In Figures 5.14 and 5.15, the experimental curve at the high strain rate shows a higher stress at the initial stage of loading than the numerical results. However, the high stress levels seen in the experimental results in the early portion of the stress-strain curve are most likely an artifact of the specimen geometry and the way the stress waves propagated through the specimen. At higher strain levels, the comparison is much improved, and the overall predictions may actually be fairly accurate. In Figure 5.13, at the high strain rate for higher strain levels the experimental curve flattens out where the computed curve does not, but this could be due to an early failure in the experimental specimen. The comparison between the experimental and numerical results is excellent for the lower strain rate in all three of the laminate configurations.
Figure 5.13 Experimental and LS-DYNA predicted shear stress-strain curves for IM7/977-2 [10°] at strain rates of 320/s and 0.56/s

Figure 5.14 Experimental and LS-DYNA predicted shear stress-strain curves for IM7/977-2 [45°] at strain rates of 405/s and 1.2/s
5.2.2 Failure envelope prediction for IM7/977-2

The strength values of carbon fiber reinforced polymer matrix composite IM7/977-2 at various off axis angles were examined with the rate-dependent failure criteria in the composite UMAT. The reference strain rate is 1/s and $\beta$ is equal to 0.065. The longitudinal tensile strength was defined as 2300MPa, the longitudinal compressive strength was 900MPa, the transverse tensile strength was 73MPa, the transverse compressive strength was 146MPa and the shear strength was 85MPa$^{34}$. Note that these strength values are quasi-static values.

The model prediction of the failure envelopes at strain rates of 1/s and 405/s are compared with the experiment results as shown in Figure 5.16. As can be seen from the
figure, the model predictions are in good agreement with the experiment, implying that the single rate-sensitive parameter $\beta$ captures the rate dependent failure strength reasonably accurate. Note that the experimental obtained strength value at the off-axis angle of 10 degree at 405/s is approximated with the value obtained at 320/s, therefore the actual strength value at 405/s is supposed to be slightly higher than that in the chart, which will fit the failure envelope even better.

![Graph showing model predictions and experimental results](image)

Figure 5.16 Model predictions of the failure envelopes for the unidirectional IM7/977-2 composite at various strain rates

5.2.3 Angle-ply IM7/977-2 predictions

As has been stated before, the composite UMAT is designed for a unidirectional single ply. By defining the orientations of each through-thickness integration points in the shell elements, the UMAT can also simulate the behavior of composite laminate of various fiber configurations. The deformation behavior of IM7/977-2 [$\pm 45^\circ$], laminate
at tensile strain rates of $9.0 \times 10^{-5}/s$, 2.1 /s and 604/s was predicted with the composite UMAT. Four through-thickness integration points were defined in the shell element and oriented at $+45^\circ$, $-45^\circ$, $-45^\circ$ and $+45^\circ$, respectively.

Initially the damage of the polymer was not considered (damage constant $C_1=0.0$) in the composite UMAT prediction. The computed results are compared with the experimental data\textsuperscript{74}, as shown in Figure 5.17(a). The model without polymer damage predicts the nonlinear, strain rate dependent deformation behavior of the $[\pm 45^\circ]_s$ laminate well up to a strain level of 1%. Beyond that strain, the model overpredicts the experimental results significantly. Similar situation has been found by Thiruppukuzhi and Sun\textsuperscript{9}. They claimed that at large strain, damage set in the laminate which was not considered in their model. It has been known that the angle-ply laminates are more matrix dominated than unidirectional lamina. The large in-plane shear induces damage in the polymer matrix which significantly reduces the load carrying capability of the laminate. Therefore, taking account for the damage in the polymer would capture the deformation behavior of the angle-ply laminate more realistically.

The UMAT simulation was then carried out with polymer damage considered by setting the damage constant $C_1=120$ in Equation 2.18. The simulation in comparison with the experimental results is shown in Figure 5.17(b). Good correlation between the numerical and experimental results over the whole strain range is observed in the figure for the low ($9 \times 10^{-5}/s$) and moderate (2.1/s) strain rates. Some degree of over prediction for the high strain rate (604/s) is still observed, but is in a significantly reduced amount than in Figure 5.18 (a). By setting $C_1=240$ for the high strain rate simulation, good agreement between the numerical and experiment results is achieved over the entire strain
range for the high strain rates, as shown in Figure 5.17(c). This may indicate that the polymer damage parameter $C_1$ might be strain rate dependent.

Figure 5.17 Experimental and LS-DYNA simulated stress-strain curves

(a) Without polymer damage ($C_1=0$)

(b) With polymer damage ($C_1=100$)

Figure 5.17 Experimental and LS-DYNA simulated stress-strain curves
(c) With polymer damage ($C_1=100$ for strain rates of 2.1/s and $9 \times 10^{-5}$/s,

$C_1=120$ for strain rate of 604/s)

Figure 5.17 (continued) Experimental and LS-DYNA simulated stress-strain curves

for IM7/977-2 [$\pm 45^\circ$], at strain rates of 604/s, 2.1/s and $9 \times 10^{-5}$/s

5.2.4 Off-axis AS4/PEEK predictions

The nonlinear, rate-dependent deformation behavior of AS4/PEEK composite is studied in this section for various off-axis angles. The AS4 fiber mechanical properties can be found in Table 2, and the material constants for thermoplastic PEEK have been characterized previously as listed in Table 1. Uniaxial tension tests were performed by Weeks and Sun\textsuperscript{13} with a servo-hydraulic testing machine. The LS-DYNA simulated and experimental\textsuperscript{13} results are compared in Figure 5.18 for off-axis angles of 14°, 30°, 45° and 90° at strain rate of $1 \times 10^{-5}$/s. Results for off-axis angles of 15°, 30°, 45° and 90° at strain
rate of 0.1/s is shown in Figure 5.19. From both of the figures it can be seen that the predicted and experimental results correlates well for all the fiber orientations, and the strain rate dependent behavior is captured for the two strain rates studied.

Figure 5.18 Experimental and LS-DYNA predicted stress-strain curves for off-axis AS4/PEEK at strain rate of $1 \times 10^{-5}$/s
Figure 5.19 Experimental and LS-DYNA predicted stress-strain curves for off-axis AS4/PEEK at strain rate of 0.1/s.

The unidirectional [0°] AS4/PEEK laminate was examined for strain rates of $1 \times 10^{-5} /s$, $1 \times 10^{-3}/s$ and 0.1/s. The experimental\textsuperscript{13} and predicted results are shown in Figure 5.20. It can be seen from the figure that the fiber dominated [0°] laminate exhibits neither nonlinearity nor strain rate effect. The LS-DYNA simulated stress-strain curves for the three strain rates overlaps with each other and appears to be one linear curve.

One important point to note is that, as mentioned by Weeks and Sun\textsuperscript{13}, the end points presented in their experimental results should not be taken as strains to failure or failure stresses. Therefore, failure predictions for these experimental tests were not carried out with the composite UMAT.
Figure 5.20 Experimental and LS-DYNA predicted stress-strain curves for AS4/PEEK $[0^\circ]$ at strain rate of $1 \times 10^{-5}$ /s, $1 \times 10^{-3}$ /s and 0.1/s
5.2.5 Angle-ply AS4/PEEK predictions

The deformation behavior of balanced angle-ply AS4/PEEK laminates of $[\pm 30^\circ]_{2s}$, $[\pm 47^\circ]_{2s}$ and $[\pm 60^\circ]_{2s}$ was examined for strain rate of 0.01/s. The LS-DYNA predicted and experimental results are compared in Figure 5.21. Though there is some over prediction for $[\pm 30^\circ]_{2s}$ laminate, the UMAT predicted results agrees reasonably well with the experimental data. Note that the damage model for the polymer did not need to be applied for this material.

![Figure 5.21 Experimental and LS-DYNA predicted stress-strain curves for AS4/PEEK $[\pm \theta^\circ]_{2s}$ at strain rate of 0.01/s](image)

Figure 5.21 Experimental and LS-DYNA predicted stress-strain curves for AS4/PEEK $[\pm \theta^\circ]_{2s}$ at strain rate of 0.01/s
The predicted and experimental stress-strain curve for $[\pm 15^\circ]_{2s}$ at strain rate of 0.1/s is shown in Figure 5.22. With the shallow ply angle, the linear deformation is observed in the figure, and the LS-DYNA predicted result correlates well with the experiment.

![Graph showing stress-strain relation for composite material](image)

Figure 5.22 Experimental and LS-DYNA predicted stress-strain curves for AS4/PEEK $[\pm 15^\circ]_{2s}$ at strain rate of 0.1/s

5.2.6 Example for progressive composite post failure model

To evaluate the composite post failure model, the longitudinal stress-strain behavior of IM7/977-2 under tension at the fiber direction is studied. The effect of damage exponent $m$ on the post failure prediction is shown in Figure 5.23. It can be seen from the figure that large values of $m$ represent brittle failure that is close to
instantaneous failure, whereas small values of $m$ result in a higher post failure strength and more ductile behavior.

![Stress vs. Strain graph](image.png)

**Figure 5.23** Effect of damage exponent $m$ on the post failure prediction

5.3 Braiding/weaving with through-thickness integration points method

The micromechanical based composite UMAT is designed for unidirectional lamina. By defining the fiber orientation angles for each layer at the through-thickness integration points, the UMAT has been verified in the above sections with composite laminates of off-axis and angle-plies.

Due to their low cost and high impact resistance, textile composites are gaining more favor in aerospace industry nowadays. To expand the application of the
unidirectional composite UMAT to the simulation of textile composites, braiding/weaving with through-thickness integration points method proposed by Cheng was modified and utilized with the composite UMAT.

The basic idea in the braiding methodology is first to select the unit cell representing the fiber architecture in the lamina. Unit cell here is defined as the smallest repeating fiber architecture in the textile composites, as the example shown in Figure 5.24 for a triaxial braided lamina. In the Figure, the axis 1 is the fiber direction, the axis 2 is the transverse direction and the axis 3 is the thickness direction. The unit cell is then divided into four sub-cells, namely sub-cell A, B, C and D. For one layer, each sub-cell is divided into 2 or 3 slices of equal thickness in the thickness direction. Each of the slices represents a fiber bundle in $0^\circ, +\theta^\circ$ or $-\theta^\circ$ direction respectively. In sub-cells A and C, there are three fiber bundles $(0^\circ \pm \theta^\circ)$ in one layer; while sub-cells B and D contain only two fiber bundles $(\pm \theta^\circ)$ in the bias directions in one layer. Equally spaced integration points are assigned to each of the slices in the sub-cells. Each of the integration points is associated with a material angle of $0^\circ, +\theta^\circ$ or $-\theta^\circ$ and assigned a material definition of the existing material types.
Figure 5.24 Unit cell for braided composites
In the modified braiding methodology, as will be applied to the following examples, the dimensions of sub-cells in the unit cell are recalculated in two aspects.

1) In order to simplify the mesh generation, the widths of the subcells are assumed to be equal since the difference was negligible, as seen in the braided fiber perform shown in Figure 5.25.

2) To ensure the fiber volume ratio does not exceed 100% in any of the slices, the thickness in the 0° slices in is assumed to be larger than that in the ±θ° slices in sub-cells A and C. The integration rules for the through-thickness integration points are accordingly modified.

![Figure 5.25 Braided fiber preform](image)

The composite UMAT is applied to the braided composite architecture to capture its nonlinear deformation behavior. Detailed explanation will be given with examples in the following sections.
5.3.1 Unit cell dimensions

The examples used in this study are the triaxial-braid T700/M36 toughened epoxy composites of two fiber architectures \((0^\circ \pm 60^\circ, 0^\circ \pm 45^\circ)\) tested by Bowman et al.\(^7\). In this study, these braided composites will be used as examples in explaining the braiding/weaving with through-thickness integration points methodology and their tensile behavior is simulated with the composite UMAT.

As described by Bowman et al.\(^7\), the carbon/epoxy composite systems had a fiber volume ratio of 56.5%. In the \(0^\circ \pm 60^\circ\) system, each direction contained 1/3 of the fibers. In the \(0^\circ \pm 45^\circ\) system, 51.5% of the fibers were in the \(0^\circ\) direction and 48.5% were in the \(\pm 45^\circ\) direction. The \(0^\circ \pm 60^\circ\) braided preform had 12k flat tow fibers in the \(\pm 60^\circ\) bias directions and 24k flat tow fibers in the \(0^\circ\) axial direction. The \(0^\circ \pm 45^\circ\) braided preform had 12k flat tow fibers in the \(\pm 45^\circ\) bias directions and 36k flat tow fibers in the \(0^\circ\) axial direction. The spacing between axial tows was 8.9mm in both systems. The spacing between the perpendicular repeating units was 5.2mm for the \(\pm 60^\circ\) layup and 8.9mm for the \(\pm 45^\circ\) layup. The cured composites contained six plies with a total nominal thickness of 3.2mm.

Based on the information given above, unit cells for both \(0^\circ \pm 60^\circ\) and \(0^\circ \pm 45^\circ\) systems can be determined. The dimensions of the unit cell for \(0^\circ \pm 60^\circ\) braided system are shown in Figure 5.26 and those for \(0^\circ \pm 45^\circ\) system are shown in Figure 5.27. Since the fibers in the \(0^\circ\) axial direction had more tows than those in the bias directions, the thickness assigned for the \(0^\circ\) fibers in sub-cells A and D is larger than that for the \(\pm 60^\circ\) or \(\pm 45^\circ\) directions. The fiber volume ratios can be calculated based on the
dimensions of the thickness slices as shown in Table 3 for $0^\circ \pm 60^\circ$ system and in Table 4 for $0^\circ \pm 45^\circ$ system.

Figure 5.26 Unit cell for $0^\circ \pm 60^\circ$ braided composite system

Figure 5.27 Unit cell for $0^\circ \pm 45^\circ$ braided composite system
Table 3 Fiber volume ratios for thickness slices in $0^\circ \pm 60^\circ$ system

<table>
<thead>
<tr>
<th></th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
</tr>
</thead>
<tbody>
<tr>
<td>$0^\circ V_f$</td>
<td>85.6%</td>
<td>N/A</td>
<td>85.6%</td>
<td>N/A</td>
</tr>
<tr>
<td>$60^\circ V_f$</td>
<td>67.3%</td>
<td>37.7%</td>
<td>67.3%</td>
<td>37.7%</td>
</tr>
<tr>
<td>$-60^\circ V_f$</td>
<td>67.3%</td>
<td>37.7%</td>
<td>67.3%</td>
<td>37.7%</td>
</tr>
</tbody>
</table>

Table 4 Fiber volume ratios for thickness slices in $0^\circ \pm 45^\circ$ system

<table>
<thead>
<tr>
<th></th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
</tr>
</thead>
<tbody>
<tr>
<td>$0^\circ V_f$</td>
<td>97.0%</td>
<td>N/A</td>
<td>97.0%</td>
<td>N/A</td>
</tr>
<tr>
<td>$45^\circ V_f$</td>
<td>68.5%</td>
<td>27.4%</td>
<td>68.5%</td>
<td>27.4%</td>
</tr>
<tr>
<td>$-45^\circ V_f$</td>
<td>68.5%</td>
<td>27.4%</td>
<td>68.5%</td>
<td>27.4%</td>
</tr>
</tbody>
</table>

5.3.2 Integration rules

Each of the slices in the thickness direction in the unit sub-cells can be simulated with one through-thickness integration point. More integration points can be used for one slice to improve the transverse shear calculations, but the price is the consequently increased computation time. In this study, only one integration point is used for each thickness slice. Therefore, for sub-cells A and C, three integration points are used for one layer and total 18 points for the 6 layers in the composites. For sub-cells B and D, two integration points are used for one layer and hence 12 points for the whole composites.

Integration rules need to be defined for each integration point, including its position and weight. The actual positions of the integration points are scaled to (-1,1) in the shell formulation and the weights are calculated by dividing the actual slice thickness by the total composite thickness, assuming one integration point is used for one slice.
The integration rule for each sub-cell in $0^\circ \pm 60^\circ$ and $0^\circ \pm 45^\circ$ systems are listed in Table 5.

Table 5  Integration rules for the $0^\circ \pm 60^\circ$ and $0^\circ \pm 45^\circ$ systems

<table>
<thead>
<tr>
<th>Number</th>
<th>Position</th>
<th>Weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-0.9533</td>
<td>0.0466</td>
</tr>
<tr>
<td>2</td>
<td>-0.8333</td>
<td>0.0733</td>
</tr>
<tr>
<td>3</td>
<td>-0.7133</td>
<td>0.0467</td>
</tr>
<tr>
<td>4</td>
<td>-0.6200</td>
<td>0.0467</td>
</tr>
<tr>
<td>5</td>
<td>-0.5000</td>
<td>0.0733</td>
</tr>
<tr>
<td>6</td>
<td>-0.3800</td>
<td>0.0467</td>
</tr>
<tr>
<td>7</td>
<td>-0.2867</td>
<td>0.0467</td>
</tr>
<tr>
<td>8</td>
<td>-0.1667</td>
<td>0.0733</td>
</tr>
<tr>
<td>9</td>
<td>-0.0467</td>
<td>0.0467</td>
</tr>
<tr>
<td>10</td>
<td>0.0467</td>
<td>0.0467</td>
</tr>
<tr>
<td>11</td>
<td>0.1667</td>
<td>0.0733</td>
</tr>
<tr>
<td>12</td>
<td>0.2867</td>
<td>0.0467</td>
</tr>
<tr>
<td>13</td>
<td>0.3800</td>
<td>0.0467</td>
</tr>
<tr>
<td>14</td>
<td>0.5000</td>
<td>0.0733</td>
</tr>
<tr>
<td>15</td>
<td>0.6200</td>
<td>0.0467</td>
</tr>
<tr>
<td>16</td>
<td>0.7133</td>
<td>0.0467</td>
</tr>
<tr>
<td>17</td>
<td>0.8333</td>
<td>0.0733</td>
</tr>
<tr>
<td>18</td>
<td>0.9533</td>
<td>0.0466</td>
</tr>
</tbody>
</table>
Table 5 (continued) Integration rules for the $0^\circ \pm 60^\circ$ and $0^\circ \pm 45^\circ$ systems

<table>
<thead>
<tr>
<th>Sub-cell A&amp;C in the $0^\circ \pm 45^\circ$ system</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Number</td>
<td>Position</td>
</tr>
<tr>
<td>1</td>
<td>-0.9667</td>
</tr>
<tr>
<td>2</td>
<td>-0.8333</td>
</tr>
<tr>
<td>3</td>
<td>-0.7000</td>
</tr>
<tr>
<td>4</td>
<td>-0.6333</td>
</tr>
<tr>
<td>5</td>
<td>-0.5000</td>
</tr>
<tr>
<td>6</td>
<td>-0.3667</td>
</tr>
<tr>
<td>7</td>
<td>-0.3000</td>
</tr>
<tr>
<td>8</td>
<td>-0.1667</td>
</tr>
<tr>
<td>9</td>
<td>-0.0333</td>
</tr>
<tr>
<td>10</td>
<td>0.0333</td>
</tr>
<tr>
<td>11</td>
<td>0.1667</td>
</tr>
<tr>
<td>12</td>
<td>0.3000</td>
</tr>
<tr>
<td>13</td>
<td>0.3667</td>
</tr>
<tr>
<td>14</td>
<td>0.5000</td>
</tr>
<tr>
<td>15</td>
<td>0.6333</td>
</tr>
<tr>
<td>16</td>
<td>0.7000</td>
</tr>
<tr>
<td>17</td>
<td>0.8333</td>
</tr>
<tr>
<td>18</td>
<td>0.9667</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Sub-cell B&amp;D in both systems</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Number</td>
<td>Position</td>
</tr>
<tr>
<td>1</td>
<td>-0.9167</td>
</tr>
<tr>
<td>2</td>
<td>-0.7500</td>
</tr>
<tr>
<td>3</td>
<td>-0.5833</td>
</tr>
<tr>
<td>4</td>
<td>-0.4167</td>
</tr>
<tr>
<td>5</td>
<td>-0.2500</td>
</tr>
<tr>
<td>6</td>
<td>-0.0833</td>
</tr>
<tr>
<td>7</td>
<td>0.0833</td>
</tr>
<tr>
<td>8</td>
<td>0.2500</td>
</tr>
<tr>
<td>9</td>
<td>0.4167</td>
</tr>
<tr>
<td>10</td>
<td>0.5833</td>
</tr>
<tr>
<td>11</td>
<td>0.7500</td>
</tr>
<tr>
<td>12</td>
<td>0.9167</td>
</tr>
</tbody>
</table>
5.3.3 Material type assignment

Each through-thickness integration point is assigned with a material angle and a material definition. In this example, the angles are $0^\circ$, $+\theta^\circ$ or $-\theta^\circ$ ($\theta^\circ = 60^\circ$ or $45^\circ$). For the material definition, from Table 3 and 4 it can be seen that 3 types of materials are needed:

1) The $0^\circ$ slices in sub-cells A and C
2) The $\pm \theta^\circ$ slices in sub-cells A and C
3) The $\pm \theta^\circ$ slices in sub-cells B and D

The composite UMAT will be used for the $\pm \theta^\circ$ slices in sub-cells B and D. Due to the high fiber volume ratio in the $0^\circ$ slices in sub-cells A and C, the composite UMAT is not suitable since the maximum applicable fiber volume ratio for the developed composite model is 78.54%, as has been discussed in Chapter 2. Therefore, an existing orthotropic composite model Material 54 is used for the $0^\circ$ slices in sub-cells A and C. Note that in LS-DYNA 970, only one material type can be used in one element, but the material constants can be different for different through-thickness integration points. Consequently, Material 54 is also used for the $\pm \theta^\circ$ slices in sub-cells A and C, but a different set of material constants will be used from that for the $0^\circ$ slices.

5.3.4 Material constants determination

The mechanical properties for the carbon fiber T700 is listed in Table 6. The longitudinal elastic modulus $E_{11}$ shown in the table was obtained from the official
website\textsuperscript{77} of Toray Industries Inc. The transverse properties were provided by Gundberg\textsuperscript{78}.

Table 6 Material properties for T700 carbon fiber

<table>
<thead>
<tr>
<th>E\textsubscript{11f}(GPa)</th>
<th>E\textsubscript{22f}(GPa)</th>
<th>G\textsubscript{12f}(GPa)</th>
<th>V\textsubscript{12f}</th>
</tr>
</thead>
<tbody>
<tr>
<td>230</td>
<td>10</td>
<td>17</td>
<td>0.27</td>
</tr>
</tbody>
</table>

The material constants for Hexply M36 toughened epoxy is listed in Table 7. Among these constants, the elastic modulus and Poisson’s ratio were obtained from the official website of Hexcel Corporation\textsuperscript{79}. The strain rate effects were not considered in this study due to two reasons: first, the strain rate tests data were not available; second, the coupon tests going to be studies were carried out with approximately equal quasi static strain rates at around 7.5\times10^{-5}/s. In such case, the material constant C\textsubscript{1} associated with the rate dependent elastic modulus was set to zero, and the rate parameter n was assumed to be 0.93 (though other values were also possible). Then the remaining material constants were characterized at the strain rate of 7.5\times10^{-5}/s. To determine the constants \(\alpha\_0\) and \(\alpha\_1\) for the hydrostatic stress effects, the values of tensile strength (81MPa) and compressive strength (146MPa) \textsuperscript{79} were assumed to equal the stresses at saturation. Then \(\alpha\_1\) was calculated by \(146(1-\sqrt{3}\alpha\_1) = 81(1+\sqrt{3}\alpha\_1)\), which yielded \(\alpha\_1 = 0.165\). \(\alpha\_0\) was assumed to be equal to \(\alpha\_1\). The value of \(Z\_1\) was determined by carrying out tensile and compressive single element tests in such a way that the maximum tensile stress was equal to the tensile strength (81MPa) and the maximum compressive stress was equal to the compressive strength (146MPa). The value of \(Z\_0\) was
approximated so that the nonlinearity started approximately at strain of 1%, which is similar as the previously studied epoxies. The simulated stress-strain curves under tension and compression using the above assumed material constants are shown in Figure 5.28.

Table 7 Material properties for Hexply M36 epoxy

<table>
<thead>
<tr>
<th>$E_m$ (GPa)</th>
<th>C</th>
<th>$\nu_m$</th>
<th>$D_o$ (1/s)</th>
<th>n</th>
<th>$Z_o$ (MPa)</th>
<th>$Z_1$ (MPa)</th>
<th>q</th>
<th>$\alpha_o$</th>
<th>$\alpha_1$</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.5</td>
<td>0.0</td>
<td>0.42</td>
<td>1x10^6</td>
<td>0.93</td>
<td>500</td>
<td>1220</td>
<td>100.0</td>
<td>0.165</td>
<td>0.165</td>
</tr>
</tbody>
</table>

Figure 5.28 Simulated tensile and compressive stress-strain curves of M36 epoxy
The fiber and the matrix material constants listed in Tables 6 and 7 were used in the composite UMAT for the slices in sub-cells B and D.

To obtain the material constants in Material 54 for slices in sub-cells A and C, rule of mixture\(^80\) was used as following:

\[
E_{11} = V_f E_{11f} + (1 - V_f)E_m \tag{5.1}
\]

\[
1/E_{22} = V_f/E_{22f} + (1 - V_f)/E_m \tag{5.2}
\]

\[
\nu_{12} = V_f\nu_f + (1 - V_f)\nu_m \tag{5.3}
\]

\[
1/G_{12} = V_f/G_{12f} + (1 - V_f)/G_m \tag{5.4}
\]

\[
\nu_{21} = \nu_{21} \cdot E_{22} / E_{11} \tag{5.5}
\]

where \(E_{11}\) and \(E_{22}\) are the lamina longitudinal and transverse elastic modulus, \(\nu_{12}\) and \(\nu_{21}\) are the in-plane Poisson’s ratios and \(G_{12}\) is the lamina shear modulus. \(G_m\) is the matrix shear modulus which is calculated as

\[
G_m = \frac{E_m}{2(1 + \nu_m)} \tag{5.6}
\]

The calculated material constants used in Material 54 for sub-cells A and C are listed in Table 8.

Table 8 Material constants in Material 54 for Sub-cells A & C

<table>
<thead>
<tr>
<th>System</th>
<th>Slice</th>
<th>(E_{11}) (GPa)</th>
<th>(E_{22}) (GPa)</th>
<th>(\nu_{21})</th>
<th>(G_{12}) (GPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>(0^\circ \pm 60^\circ)</td>
<td>(0^\circ)</td>
<td>197.4</td>
<td>7.9</td>
<td>0.0117</td>
<td>6.0</td>
</tr>
<tr>
<td></td>
<td>(\pm 60^\circ)</td>
<td>155.9</td>
<td>6.2</td>
<td>0.0122</td>
<td>3.3</td>
</tr>
<tr>
<td>(0^\circ \pm 45^\circ)</td>
<td>(0^\circ)</td>
<td>223.2</td>
<td>8.2</td>
<td>0.0100</td>
<td>12.3</td>
</tr>
<tr>
<td></td>
<td>(\pm 45^\circ)</td>
<td>158.7</td>
<td>6.3</td>
<td>0.0121</td>
<td>3.4</td>
</tr>
</tbody>
</table>
5.3.5 Coupon tests simulation

Coupon tensile tests on the axial and transverse direction were carried out by Bowman et al.\textsuperscript{76} with the two systems of the triaxial braided composites. The straight-sided tensile specimens were 305mm long in the loading direction and had a width of approximately 4 repeating unit. The loading rate was about $7.5 \times 10^{-5}$/s.

Belytschko-Tsay shell elements were used in the coupon simulation. The dimensions of the shell elements, the integration rules and material definitions were as described above. The finite element models for the axial and transverse coupon tensile tests are shown in Figure 5.29 and Figure 5.30 respectively. The coupons were fixed at one end and pulled at the other at a constant strain rate of $7.5 \times 10^{-5}$/s. The resultant forces in the loading direction at the nodes of the pulling end were output and summed up. Then the summed force was divided by the cross section of the coupon to obtain the tensile stress. The strain was obtained by dividing the displacement of the node on the pulling end by the specimen length. Then the stress-strain curves can be plotted as shown in Figure 5.31 for the $0^\circ \pm 60^\circ$ and $0^\circ \pm 45^\circ$ systems in both axial and transverse directions. It can be seen from the figure that the prediction of the deformation behavior of the braided composites correlates well with the experimental data. Linear deformation behavior is observed in the axial loading of the $0^\circ \pm 45^\circ$ system since in this loading condition the system is extensively fiber dominant. The $0^\circ \pm 60^\circ$ system exhibits the quasi-isotropic behavior with is captured by the LS-DYNA prediction with the braiding methodology and the material definitions. Though there is slight nonlinearity shown at the end of the curves, the linear deformation behavior of the $0^\circ \pm 60^\circ$ system due to the
fiber domination is predicted reasonably well by the LS-DYNA simulation. The deformation behavior is matrix dominant in the transverse direction of the $0^\circ \pm 45^\circ$ system and obvious nonlinearity is shown in the experimental tests. This nonlinearity is successfully captured by the composite UMAT.

Figure 5.29 Coupon tensile tests in axial direction
Figure 5.30 Coupon tensile tests in transverse direction

![Graph showing coupon tensile tests with different orientations and stress-strain relationships.]

Figure 5.31 Experimental and predicted coupon tensile tests
CHAPTER VI

CONCLUSIONS

In this study, the nonlinear, rate dependent constitutive polymer model developed by Goldberg has been modified in three aspects:

1) incorporated the rate dependence of the elastic modulus,

2) modified the evolution law for the variable of the hydrostatic stress effect to eliminate the influence of the shear components,

3) proposed a simple damage model to improve the unloading prediction.

The modified polymer model was implemented into the explicit finite element code, LS-DYNA, as a user defined material model (UMAT). The polymer UMAT was verified with two types of toughened epoxies (PR520, 977-2), one standard epoxy (E862) and a thermoplastic (PEEK), showing good prediction capabilities in the polymer’s nonlinear, rate-dependent deformation behavior. The hydrostatic stress effect and the improved unloading behavior were also studied with PEEK.
The modified polymer constitutive equations were then implemented into a micromechanical composite model to predict the nonlinear, strain rate dependent deformation of the composite. Strain rate dependent failure criteria and post failure progressive damage model were incorporated into the composite model.

The composite constitutive and damage model was implemented into LS-DYNA as UMAT. The deformation behaviors of two carbon fiber reinforced polymer matrix composites (IM7/977-2 and AS4/PEEK) of various fiber configurations were simulated in LS-DYNA with the composite UMAT for a wide range of strain rates, and the numerical results agreed well with the experimental data. The strain rate dependent failure criteria were also used to obtain failure envelopes which successfully predicted the strength values of off-axis composite laminas at two strain rates.

In order to expand the application of the unidirectional composite UMAT, braiding/weaving with through-thickness integration points method was proposed with examples to simulate the deformation behavior of textile composites. The triaxial braided T700/M36 carbon fiber/epoxy composites of two fiber architectures (0° ± 60° and 0° ± 45°) were examined with this method and the use of the composite UMAT. The deformation behavior of the coupon tensile specimen were successfully predicted, especially the nonlinear deformation behavior in the matrix dominated transverse 0° ± 45° system.

Possibilities for future work of this study include:

1) examine the UMATs with multi-axial loading and varying strain rates

2) the unloading behavior of polymers needs to be further investigated especially
for the high strain rates. Based on the unloading experiments, the damage evolution law can then be properly determined.

3) apply the composite UMAT to the large scale impact simulations.
REFERENCES


151
12 Goldberg, R.K., Roberts, G.D. and Gilat, A., "Implementation of an associative flow rule including hydrostatic stress effects into the high strain rate deformation analysis of polymer matrix composites", Journal of Aerospace Engineering, Vol.18, pp.18


33 Bordonaro, C.M., “Rate dependent mechanical behavior of high strength plastics: experiment and modeling”, PhD dissertation, Rensselaer Polytechnic Institute, Troy, New York, 1995


153


55 Murray, Y. and Schwer, L., "Implementation and verification of fiber composite damage models", Failure Criteria and Analysis in Dynamic Response; ASME AMD, Vol.107, pp.21-30, 1990


78 Correspondence with Trevor A. Gundberg, Assistant technical marketing manager, Toray Carbon Fibers America, Inc., 2006

79 Hexply M36 data sheet, Hexcel company, http://www.hexcel.com/Products/Matrix+Products/Other+FRM/RFI/, 2002


81 Cheng, J. Material Modeling of Strain Rate Dependent Polymer and 2D Tri-axially Braided Composites, PhD dissertation, University of Akron, Akron, Ohio, 2006
APPENDIX

c******************************************************************
c FEM code for the nonlinear, strain rate dependent material testing
c using the same method as in LS-DYNA
c******************************************************************

program fempolymer

dimension Xcod(3,100),Xcod0(3,100),ELENOD(100,4),X(4),Y(4)

dimension a(5,100),ve(5,100),d(5,100),dtot(5,100)

dimension b1(4),b2(4),bM(100,100),bLM(100)

dimension f(5,100),fr(6),fm(3),IMbc(100),IMload(100)

dimension u(4),v(4),w(4)

dimension cm(40),eps(6),hsv(100),sig(6),hisv(100,100)

dimension ET(100,6),sigma(100,6)

OPEN (UNIT=1,FILE='input11.txt',STATUS='UNKNOWN',
   ACCESS='SEQUENTIAL')

OPEN (UNIT=2,FILE='input2.txt',STATUS='UNKNOWN',
   ACCESS='SEQUENTIAL')

OPEN (UNIT=3,FILE='3description.txt',STATUS='UNKNOWN',
   ACCESS='SEQUENTIAL')
OPEN (UNIT=4,FILE='4NodalDisplacement.txt',STATUS='UNKNOWN',
1 ACCESS='SEQUENTIAL')
OPEN (UNIT=5,FILE='5ElementVelocity.txt',STATUS='UNKNOWN',
1 ACCESS='SEQUENTIAL')
OPEN (UNIT=6,FILE='6StrainIncrement.txt',STATUS='UNKNOWN',
1 ACCESS='SEQUENTIAL')
OPEN (UNIT=7,FILE='7Stress.txt',STATUS='UNKNOWN',
1 ACCESS='SEQUENTIAL')
OPEN (UNIT=8,FILE='8NodalForce.txt',STATUS='UNKNOWN',
1 ACCESS='SEQUENTIAL')
OPEN (UNIT=9,FILE='9Matrix.txt',STATUS='UNKNOWN',
1 ACCESS='SEQUENTIAL')

C***************************************************************
C Construct the geometry: read in node and element, BC and load
C***************************************************************
READ(1,*) nNode, nEle
c Read node global coordinate
write(3,*) "* Node"
   DO 100 I=1,nNode
      READ(1,*) IM, Xcod0(1,I),Xcod0(2,I),Xcod0(3,I)
      write(3,*) IM, Xcod0(1,I),Xcod0(2,I),Xcod0(3,I)
   100 CONTINUE
c Read the 4 node numbers for each shell element
write(3,*) "* Element"

DO 110 J=1,nEle
    READ(1,*) JM,(ELENOD(J,I),I=1,4)
    write(3,*) J,(ELENOD(J,I),I=1,4)
110    CONTINUE

c Read boundary and loading conditions

write(3,*) "* Boundary Node"
READ(1,*) nBC,nLOAD
DO 120 I=1,nBC
    READ(1,*) IMbc(i)
    write(3,*) IMbc(i)
120 CONTINUE

c Write(3,*) "* Loading Node"
DO 130 I=1,nLOAD
    READ(1,*) IMload(i)
    write(3,*) IMload(i)
130 CONTINUE

c Read the number of time steps, loading direction, nodal displacement rate (mm/s),
cshell thickness

READ(1,*) nstep
READ(1,*) idirection
READ(1,*) velo
READ(1,*) tz
READ(1,*) np
READ(1,*) outele
write(3,*) "* Number of Time Step: 
write(3,*) nstep
write(3,*) "* Loading Direction: 
write(3,*) idirection
write(3,*) "* Displacement per second: 
write(3,*) velo
write(3,*) "* Shell thickness: 
write(3,*) tz
write(3,*) "* Output interval: 
write(3,*) np
write(3,*) "* Output element: 
write(3,*) outele

c*************************************************************************

c Read material properties

c*************************************************************************

READ(2,*) ro
READ(2,*) Ncm
READ(2,*) Nhv
write(3,*) "* Material Density:"
write(3,*) ro
write(3,*) "* Number of Material Constants"
write(3,*) Ncm
write(3,*) "* Number of History Variable"
write(3,*) Nhv
write(3,*) "* Material Constants:"
DO 150 i=1,Ncm
    READ(2,*) cm(i)
    write(3,*) i,cm(i)
150    CONTINUE

c Calculate dt (simplified the calculation of dl)
    Bulk=cm(1)
    G=cm(2)

c the minimum element size is assumed to be constant here for simplicity
    dl=abs(Xcod0(1,1)-Xcod0(1,2))
    dt=dl/sqrt((Bulk+4.0*G/3.0)/ro)
    dd=velo*dt
    time=0.0
write(4,*) "dt=",dt
do 155 i=1,3
    do 156 j=1,nNode
        Xcod(i,j)=Xcod0(i,j)
    156    continue
155    continue
c*****************************************************************************
c Apply initial condition
c*****************************************************************************
do 159 idof=1,5
   do 158 j=1,nNode
      a(idof,j)=0.0
      ve(idof,j)=0.0
      d(idof,j)=0.0
   158 continue
   continue
159 continue
   do 160 i=1,nLoad
      ve(idirection,IMload(i))=velo
   160 continue

*****************************************************************************
c Build Mass Matrix
*****************************************************************************
do 310 i=1,nEle
   do 320 j=1,4
      IM=ELENOD(i,j)
      X(j)=Xcod(1,IM)
      Y(j)=Xcod(2,IM)
   320 continue
\[ A_0 = 0.5 \times ((X(3)-X(1)) \times (Y(4)-Y(2)) + (X(2)-X(4)) \times (Y(3)-Y(1))) \]

do 321 \( j = 1,4 \)

do 322 \( jj = 1,4 \)

\[ bM(ELENOD(i,j),ELENOD(i,jj)) = bM(ELENOD(i,j),ELENOD(i,jj)) + \frac{\text{ro} \times A_0 \times tz}{16.0} \]

322 \( \text{continue} \)

321 \( \text{continue} \)

310 \( \text{continue} \)

\( \text{write}(9,* \) "* consistent Mass matrix"

DO 311 \( I = 1, n\text{Node} \)

\( \text{write}(9,2 \) \( (bM(I,J),J = 1, n\text{Node}) \)

311 \( \text{CONTINUE} \)

c \( \text{Calculate the lumped mass} \)

do 312 \( i = 1, n\text{Node} \)

\( \text{bsum} = 0.0 \)

do 313 \( j = 1, n\text{Node} \)

\( \text{bsum} = \text{bsum} + bM(i,j) \)

313 \( \text{continue} \)

\( \text{bLM}(i) = \text{bsum} \)

312 \( \text{continue} \)

\( \text{write}(9,* \) "* Lumped Mass matrix"

\( \text{write}(9,2 \) \( (\text{bLM}(J),J = 1, n\text{Node}) \)
C#*********************************************************************************************
C    START TIME LOOP
C#*********************************************************************************************
do 1000 kk=1,nstep
    time=time+dt
    do 166 i=1,5
        do 167 j=1,nNode
            f(i,j)=0.0
        167    continue
    166    continue
C***************************************************
C Update velocity,displacement and node coordinates
C Apply B.C. and displacement loading
C***************************************************
do 170 j=1,nNode
    do 171 idof=1,5
        ve(idof,j)=ve(idof,j)+a(idof,j)*dt
d(idof,j)=ve(idof,j)*dt
dtot(idof,j)=dtot(idof,j)+d(idof,j)
171    continue
    do 172 jj=1,3
        Xcod(jj,j)=Xcod(jj,j)+dtot(jj,j)
165
continue

C$\ldots$

c LOOP OVER ELEMENTS

C$\ldots$

c displacement increment-->strain increment

C$\ldots$

c Jacobian matrix

    do 200 i=1,nEle
        do 210 j=1,4
            IM=ELENOD(i,j)
            X(j)=Xcod(1,IM)
            Y(j)=Xcod(2,IM)
            u(j)=d(1,IM)
            v(j)=d(2,IM)
            w(j)=d(3,IM)
        210  continue
        X31=X(3)-X(1)
        Y42=Y(4)-Y(2)
        X24=X(2)-X(4)
        Y31=Y(3)-Y(1)
        Area=0.5*(X31*Y42+X24*Y31)

    200  continue
Y24 = -Y42
Y13 = -Y31
B1(1) = Y24/(2.0*Area)
B1(2) = Y31/(2.0*Area)
B1(3) = Y42/(2.0*Area)
B1(4) = Y13/(2.0*Area)
X42 = -X24
X13 = -X31
B2(1) = X42/(2.0*Area)
B2(2) = X13/(2.0*Area)
B2(3) = X24/(2.0*Area)
B2(4) = X31/(2.0*Area)

c Calculate Strain increment from displacement increment

    do 215 j=1,6
        eps(j) = 0.0
    sig(j) = sigma(i,j)
    215 continue

do 220 j=1,4
    eps(1) = eps(1) + B1(j)*u(j)
    eps(2) = eps(2) + B2(j)*v(j)
    eps(4) = eps(4) + (B2(j)*u(j) + B1(j)*v(j))/2.0
    eps(5) = eps(5) + B2(j)*w(j)/2.0
    eps(6) = eps(6) + B1(j)*w(j)/2.0

167
220 continue
do 224 j=1,Nhv
    hsv(j)=hisv(i,j)
224 continue

  c******************************************************************************
c call user developed subroutines here ---- eps --> sigma
c******************************************************************************
call umat41 (cm,eps,sig,hsv,dt,capa,etype,time,temp)
do 225 j=1,Nhv
    hisv(i,j)=hsv(j)
225 continue
do 226 j=1,6
    ET(i,j)=ET(i,j)+eps(j)
sigma(i,j)=sig(j)
226 continue

  c******************************************************************************
c Calculate stress resultants and sum up nodal forces
c******************************************************************************
do 230 j=1,6
    fr(j)=sig(j)*tz
230 continue

Area=0.50*(X(1)*Y(2)-X(2)*Y(1)+X(2)*Y(3)-X(3)*Y(2)+X(3)*Y(4)
+    -X(4)*Y(3)+X(4)*Y(1)-X(1)*Y(4))

168
do 240 j=1,4
    im=ELENOD(i,j)
    f(1,im)=f(1,im)-Area*(B1(j)*fr(1)+B2(j)*fr(4))
    f(2,im)=f(2,im)-Area*(B2(j)*fr(2)+B1(j)*fr(4))
    f(3,im)=f(3,im)-Area*(B1(j)*fr(6)+B2(j)*fr(5))
    f(4,im)=0.0
    f(5,im)=0.0
240  continue

c ccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccc

c   END ELEMENT LOOP

c ccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccc
200  continue

c$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$\n
c   Global equations of motion ----- Nodal force --> Nodal displacement

c$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
1   FORMAT(10f12.3)
2   FORMAT(10e12.3)
do 400 idof=1,5
    do 410 i=1,nNode
        a(idof,i)=f(idof,i)/bLM(i)
    410  continue

do 415 i=1,nBC
    a(idof,IMbc(i))=0.0
169
continue

do 420 j=1,nLOAD
    a(idirection,IMload(j))=0.0
continue

c#........................................................................
c  END TIME LOOP
c#........................................................................
    if(kk.eq.(np*(kk/np))) then
        write(7,2) time,et(outele,1),sigma(outele,1)
    end if
continue

END

subroutine umat41 (cm,eps,sig,hisv,dt,capa,etype,time,temp)
c  the material subroutine is not disclosed due to copyright issue