



NAVAL POSTGRADUATE SCHOOL

MONTEREY, CALIFORNIA

THESIS

TESTING AND MODELING HIGH STRAIN RATE FAILURE IN COMPOSITE MATERIALS

by

Connor J. Panick

June 2019

Thesis Advisor:
Second Reader:

Young W. Kwon
Jarema M. Didoszak

Approved for public release. Distribution is unlimited.

THIS PAGE INTENTIONALLY LEFT BLANK

REPORT DOCUMENTATION PAGE			<i>Form Approved OMB No. 0704-0188</i>	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instruction, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188) Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)	2. REPORT DATE June 2019	3. REPORT TYPE AND DATES COVERED Master's thesis		
4. TITLE AND SUBTITLE TESTING AND MODELING HIGH STRAIN RATE FAILURE IN COMPOSITE MATERIALS			5. FUNDING NUMBERS	
6. AUTHOR(S) Connor J. Panick				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Naval Postgraduate School Monterey, CA 93943-5000			8. PERFORMING ORGANIZATION REPORT NUMBER	
9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES) Office of Naval Research, Arlington, VA 22217			10. SPONSORING / MONITORING AGENCY REPORT NUMBER	
11. SUPPLEMENTARY NOTES The views expressed in this thesis are those of the author and do not reflect the official policy or position of the Department of Defense or the U.S. Government.				
12a. DISTRIBUTION / AVAILABILITY STATEMENT Approved for public release. Distribution is unlimited.			12b. DISTRIBUTION CODE A	
13. ABSTRACT (maximum 200 words) The use of fiber-matrix composite materials in military ordnance casings is of increasing interest to the U.S. Navy due to the potential for such casings to reduce collateral structure damage. In order to use fiber-matrix composite materials in such an application, the high strain rate failure characteristics of said materials must be well understood and capable of finite element (FE) implementation. This study evaluated the high strain rate behavior in relation to quasi-static behavior of a supplied carbon fiber material. Material samples were provided in three distinct composite layup orientations. High strain rate testing utilized a Charpy impact tester, and slow strain rate testing used a standard tensile tester. Experimental data revealed a rate-dependent failure strain in the composite matrix material and a rate-invariant failure strain in the carbon fiber. Material property data (including high strain rate effects) were passed into an impact specimen and tensile specimen ABAQUS FE model that used a pre-defined user subroutine capable of performing damage calculations on fiber-matrix composite materials. Additional impact tests used a load cell in order to create a time-dependent input force for the FE model (unique to each composite layup). The ABAQUS FE model produced results that were in agreement with the experimentally observed changes in composite layup material properties as functions of the applied strain rate.				
14. SUBJECT TERMS composite, dynamic failure modes, dynamic failure characteristics, dynamic failure modeling, high strain rate, anisotropic, ABAQUS, impact testing, fiber-matrix material, carbon fiber, finite element modeling			15. NUMBER OF PAGES 129	
			16. PRICE CODE	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT UU	

THIS PAGE INTENTIONALLY LEFT BLANK

Approved for public release. Distribution is unlimited.

**TESTING AND MODELING HIGH STRAIN RATE FAILURE IN COMPOSITE
MATERIALS**

Connor J. Panick
Ensign, United States Navy
BSME, U.S. Naval Academy, 2018

Submitted in partial fulfillment of the
requirements for the degree of

MASTER OF SCIENCE IN MECHANICAL ENGINEERING

from the

**NAVAL POSTGRADUATE SCHOOL
June 2019**

Approved by: Young W. Kwon
Advisor

Jarema M. Didoszak
Second Reader

Garth V. Hobson
Chair, Department of Mechanical and Aerospace Engineering

THIS PAGE INTENTIONALLY LEFT BLANK

ABSTRACT

The use of fiber-matrix composite materials in military ordnance casings is of increasing interest to the U.S. Navy due to the potential for such casings to reduce collateral structure damage. In order to use fiber-matrix composite materials in such an application, the high strain rate failure characteristics of said materials must be well understood and capable of finite element (FE) implementation. This study evaluated the high strain rate behavior in relation to quasi-static behavior of a supplied carbon fiber material. Material samples were provided in three distinct composite layup orientations. High strain rate testing utilized a Charpy impact tester, and slow strain rate testing used a standard tensile tester. Experimental data revealed a rate-dependent failure strain in the composite matrix material and a rate-invariant failure strain in the carbon fiber. Material property data (including high strain rate effects) were passed into an impact specimen and tensile specimen ABAQUS FE model that used a pre-defined user subroutine capable of performing damage calculations on fiber-matrix composite materials. Additional impact tests used a load cell in order to create a time-dependent input force for the FE model (unique to each composite layup). The ABAQUS FE model produced results that were in agreement with the experimentally observed changes in composite layup material properties as functions of the applied strain rate.

THIS PAGE INTENTIONALLY LEFT BLANK

TABLE OF CONTENTS

I.	INTRODUCTION.....	1
A.	INTRODUCTION TO FIBER-MATRIX COMPOSITE MATERIALS	1
B.	GENERAL FIBER-MATRIX COMPOSITE FAILURE THEORIES.....	3
C.	PREVIOUS RESEARCH CONCERNING STRAIN RATE EFFECTS IN FIBER-MATRIX COMPOSITES	4
D.	CURRENT RESEARCH OBJECTIVES AND GOALS.....	5
1.	Research Overview and Description	6
2.	Finite Element Model Subroutine Description.....	8
II.	SPECIMEN AND EXPERIMENT PREPARATION	11
A.	SUPPLIED MATERIAL OVERVIEW	11
B.	TEST SPECIMEN PREPARATION.....	13
C.	HIGH AND LOW STRAIN RATE TEST SETUP	17
1.	High Strain Rate (Impact) Test Setup	17
2.	Quasi-Static (Tensile) Test Setup	20
D.	ADDITIONAL TEST SETUP FOR IMPACT LOAD READINGS	21
III.	HIGH STRAIN RATE TEST RESULTS.....	25
A.	UNIDIRECTIONAL SPECIMEN HIGH STRAIN RATE RESULTS	25
1.	Impact Specimen Subset U1.....	25
2.	Impact Specimen Subset U2.....	26
3.	Interpretation of High Strain Rate Results – Subsets U1 and U2	28
B.	BIDIRECTIONAL SPECIMEN HIGH STRAIN RATE RESULTS	29
1.	Impact Specimen Subset B1	30
2.	Impact Specimen Subset B2	31
3.	Interpretation of High Strain Rate Results – Subsets B1 and B2	32
C.	QUASI-ISOTROPIC SPECIMEN HIGH STRAIN RATE RESULTS	33
1.	Impact Specimen Subset QI1	34
2.	Impact Specimen Subset QI2	35

3.	Interpretation of High Strain Rate Results – Subsets QI1 and QI2	36
D.	SUMMARY OF HIGH STRAIN RATE (IMPACT) RESULTS	36
IV.	QUASI-STATIC TEST RESULTS	39
A.	UNIDIRECTIONAL SPECIMEN QUASI-STATIC RESULTS	39
B.	BIDIRECTIONAL SPECIMEN QUASI-STATIC RESULTS	42
C.	QUASI-ISOTROPIC SPECIMEN QUASI-STATIC RESULTS.....	44
D.	SUMMARY OF QUASI-STATIC (TENSILE) RESULTS	46
V.	STRAIN RATE CORRECTION FUNCTION IMPLEMENTATION	49
A.	STRAIN RATE CORRECTION FUNCTION DERIVATION	49
B.	CALCULATION OF CORRECTION CONSTANTS	50
C.	STRAIN RATE CORRECTION CRITERIA FOR FINITE ELEMENT MODELING.....	53
VI.	IMPACT LOAD DATA FOR FINITE ELEMENT MODELING.....	55
A.	IMPACT LOAD MEASUREMENT CHALLENGES	55
B.	LOAD DATA CURVE FITTING AND MODIFICATIONS	57
VII.	FINITE ELEMENT MODEL DEVELOPMENT	61
A.	IMPACT SPECIMEN FINITE ELEMENT MODEL DEVELOPMENT	61
1.	Impact Specimen FE Geometry	61
2.	Impact Specimen Material Property Inputs.....	63
3.	Impact Specimen FE Boundary Conditions	66
4.	Impact Specimen FE Meshing	67
5.	Impact Specimen FE Loading Conditions.....	68
B.	QUASI-STATIC SPECIMEN FINITE ELEMENT MODEL DEVELOPMENT	69
1.	Quasi-Static Specimen FE Geometry	69
2.	Quasi-Static Specimen Material Property Inputs.....	70
3.	Quasi-Static Specimen FE Boundary Conditions, Meshing, and Loading	70
VIII.	COMPARISON OF FINITE ELEMENT AND EXPERIMENTAL RESULTS	73
A.	IMPACT SPECIMEN FINITE ELEMENT STRESS RESULTS	73
B.	IMPACT SPECIMEN FINITE ELEMENT STRAIN RESULTS	76
1.	Unidirectional FE Impact Strain Results	77

2.	Bidirectional FE Impact Strain Results	79
3.	Quasi-Isotropic FE Impact Strain Results	81
4.	Comparison of FE and Experimental Impact Failure Loads	83
5.	Summary of FE Impact Strain Results.....	84
C.	QUASI-STATIC SPECIMEN FINITE ELEMENT RESULTS.....	86
IX.	CONCLUSIONS AND FUTURE WORK.....	91
A.	CONCLUSIONS	91
B.	FUTURE WORK.....	93
	APPENDIX A. EXAMPLE OF IMPACT DATA PROCESSING	95
	APPENDIX B. EXAMPLE OF QUASI-STATIC DATA PROCESSING	99
	APPENDIX C. INTERACTING WITH THE USER SUBROUTINE	103
	LIST OF REFERENCES	105
	INITIAL DISTRIBUTION LIST	107

THIS PAGE INTENTIONALLY LEFT BLANK

LIST OF FIGURES

Figure 1.	Graphic Displaying Basic Unidirectional Fiber-Matrix Composite Structure. Source: [1].	2
Figure 2.	Charpy Impact Tester Used to Produce High Strain Rates	7
Figure 3.	Example of FE Result Correlation with Experimental Data for E-Glass/LY556 Composite. Source: [5].	9
Figure 4.	Unidirectional Carbon Fiber Plate (Manufacturer-Supplied)	11
Figure 5.	Specimen Composite Layup Structure – Overview	12
Figure 6.	Aluminum Tab Schematic for Reduction of Stress Concentrations in Tensile Testing	15
Figure 7.	Strain Gauge Mounting for High Strain Rate (left) and Low Strain Rate (right) Specimens	16
Figure 8.	Image of Charpy Impact Tester (left) and Impact Hammer Angle Measurement Setup (right)	17
Figure 9.	Wiring Configuration for Impact Test Strain Gauges	18
Figure 10.	Example of Test-Ready Impact Specimen and Strain Gauge Wiring	19
Figure 11.	Image of Quasi-Static DAQ System	20
Figure 12.	Images of Load Cell Apparatus and Impact Hammer Integration	22
Figure 13.	Strain versus Time Curves for Specimen Subset U1 – High Strain Rate	26
Figure 14.	Strain versus Time Curves for Specimen Subset U2– High Strain Rate	27
Figure 15.	Failure Strain as a Function of Supplied High Strain Rate – U1 and U2	28
Figure 16.	Strain versus Time Curves for Specimen Subset B1– High Strain Rate	30
Figure 17.	Strain versus Time Curves for Specimen Subset B2 – High Strain Rate	31

Figure 18.	Failure Strain as a Function of Supplied High Strain Rate – B1 and B2	33
Figure 19.	Strain versus Time Curves for Specimen Subset QI1 – High Strain Rate	34
Figure 20.	Strain versus Time Curves for Specimen Subset QI2 – High Strain Rate	35
Figure 21.	Failure Strain as a Function of Supplied High Strain Rate – QI1 and QI2	36
Figure 22.	Strain versus Time Curves for Specimen Subsets U1 and U2 – Quasi-Static	40
Figure 23.	Engineering Stress-Strain Curves – Specimen Subsets U1 and U2	41
Figure 24.	Image of Intermittent Failure of Outer, Matrix-Dominated Layer in Tensile Subset B2	42
Figure 25.	Strain versus Time Curves for Specimen Subsets B1 and B2 – Quasi-Static	43
Figure 26.	Engineering Stress-Strain Curves – Specimen Subsets B1 and B2	44
Figure 27.	Strain versus Time Curves for Specimen Subsets QI1 and QI2 – Quasi-Static	45
Figure 28.	Engineering Stress-Strain Curves – Specimen Subsets QI1 and QI2	46
Figure 29.	Strain Rate Correction Function Visualization– Subsets U1 and U2	51
Figure 30.	Strain Rate Correction Function Visualization– Subsets B1 and B2	51
Figure 31.	Strain Rate Correction Function Visualization– Subsets QI1 and QI2	52
Figure 32.	Impact Subset B1 Strain Readings – Load Cell and Impact Hammer Contact Surfaces	55
Figure 33.	Image (top) and Graphic (bottom) Displaying Load Cell Casing Contact	56
Figure 34.	Time-Dependent Impact Force Data and Curve Fits – Subsets U1 and U2	58
Figure 35.	Time-Dependent Impact Force Data and Curve Fits – Subsets B1 and B2	58

Figure 36.	Time-Dependent Impact Force Data and Curve Fits – Subsets QI1 and QI2.....	59
Figure 37.	Basic Impact Specimen FE Model Geometry.....	62
Figure 38.	Graphic Displaying Fiber Orientation of Single FE Impact Specimen Composite Layer.....	63
Figure 39.	Impact Specimen FE Boundary Conditions.....	66
Figure 40.	Impact Specimen FE Mesh.....	67
Figure 41.	Impact Specimen Applied FE Loading Condition.....	68
Figure 42.	Basic Quasi-Static Specimen FE Model Geometry.....	69
Figure 43.	Quasi-Static Specimen FE Load and Boundary Conditions.....	71
Figure 44.	Quasi-Static Specimen FE Mesh.....	72
Figure 45.	Equivalent Von-Mises Stress Distribution (MPa) – FE Impact Specimen B2.....	74
Figure 46.	ABAQUS Image of Simulated Strain Data Nodal Location.....	76
Figure 47.	FE Impact Strain Results (with Experimental Data Shown) – Subset U1.....	77
Figure 48.	FE Impact Strain Results (with Experimental Data Shown) – Subset U2.....	78
Figure 49.	FE Impact Strain Results (with Experimental Data Shown) – Subset B1.....	79
Figure 50.	FE Impact Strain Results (with Experimental Data Shown) – Subset B2.....	80
Figure 51.	FE Impact Strain Results (with Experimental Data Shown) – Subset QI1.....	81
Figure 52.	FE Impact Strain Results (with Experimental Data Shown) – Subset QI2.....	82
Figure 53.	FE Stress Distribution (σ_{11}) in Tensile Subset U1.....	86
Figure 54.	FE Strain Distribution (ϵ_{11}) in Tensile Subset U1.....	88
Figure 55.	Example of User Subroutine Material Property Input GUI.....	104

THIS PAGE INTENTIONALLY LEFT BLANK

LIST OF TABLES

Table 1.	Carbon Fiber Specimen Weave Layups and Orientations	6
Table 2.	Fiber and Matrix Material Property Information. Adapted from [8] and [9].	13
Table 3.	High and Low Strain Rate Sample Geometries	13
Table 4.	Carbon Fiber Layer Orientations for Each Specimen Subset	14
Table 5.	Strain Gauge Specifications	15
Table 6.	Impact Test Specimens, Impact Hammer Angles, and Data Acquisition Rates	20
Table 7.	Load Cell Specifications. Adapted from [10].	21
Table 8.	Mean High Strain Rate and Associated Failure Strain Values	37
Table 9.	Mean Low Strain Rate and Associated Failure Stress and Strain Values	47
Table 10.	Strain Rate Correction Constants for Each Composite Subset	50
Table 11.	Failure Strain Multiplication Factors for FE Modeling	53
Table 12.	High Strain Rate FEM Material Input List	65
Table 13.	Quasi-Static FEM Material Input List	70
Table 14.	Axial Failure Stresses – Impact FE and Quasi-Static Experimental.....	75
Table 15.	FE and Experimental Impact Failure Loads	83
Table 16.	Summary of FE Impact Strain Results.....	84
Table 17.	Quasi-Static FE and Experimental Stress Results	87
Table 18.	Quasi-Static FE and Experimental Strain Results	89

THIS PAGE INTENTIONALLY LEFT BLANK

LIST OF ACRONYMS AND ABBREVIATIONS

B1	bidirectional subset one
B2	bidirectional subset two
DAQ	data acquisition
FE	finite element
GUI	graphical user interface
QI1	quasi-isotropic subset one
QI2	quasi-isotropic subset two
UMAT	user material
UTS	ultimate tensile strength
U1	unidirectional subset one
U2	unidirectional subset two

THIS PAGE INTENTIONALLY LEFT BLANK

ACKNOWLEDGMENTS

I would first like to thank my advisor, Distinguished Professor Young Kwon. Dr. Kwon provided a profound level of knowledge throughout the entire thesis process.

Additionally, I would like to thank my wife, Megan, for providing me with the motivation and support that I needed in order to complete this milestone.

THIS PAGE INTENTIONALLY LEFT BLANK

I. INTRODUCTION

The use of fiber-matrix composites in industrial and defense applications has grown significantly since the conception and popularization of such materials. Many fiber-matrix composites are beneficial to a wide spectrum of engineering applications due to their superior strength to weight ratios when compared to traditional metal alloys.

The U.S. Navy is currently researching fiber-matrix composite ordnance casings that are capable of expelling significant heat energy with low concussive effects upon detonation. Carbon fiber is a favorable candidate for such an application since it allows for a high-temperature deflagration without the destructive concussive blast of a traditional metal ordnance casing. An ordnance with such properties would allow for higher precision strikes accompanied by reduced non-combatant casualties due to the suppressed concussive blast effects.

Due to the U.S. Navy's interest in fiber-matrix composites as ordnance casings, the behavior of this category of materials under high rates of strain must be well understood, and the ability to accurately use these materials in finite element applications are paramount to the successful development of casing designs. Therefore, this research focused on the experimental testing and comparison of the high and low strain rate failure behaviors of manufacturer-supplied carbon fiber. The results of said empirical data were then imported into a robust finite element model in order to test the validity of modeling fiber-matrix composites subject to high dynamic loading conditions (i.e. high strain rate).

A. INTRODUCTION TO FIBER-MATRIX COMPOSITE MATERIALS

Fiber-matrix composites express drastically different material properties and failure characteristics when compared to “traditional” engineering materials like simple, isotropic metal alloys. The anisotropic behavior of many fiber-matrix composites can be attributed to their nature and manufacturing processes. In a simple sense, fiber-matrix composites (to include carbon fiber) are composed of fibers that typically have extremely high tensile and compressive strength. These fibers are then aligned in a particular ‘weave’ pattern that allows for high and low strength and stiffness in certain directions depending on the

engineering application of the component. Although fibers are typically very strong (in tension), they require a matrix material to hold their shape and stability. Matrix materials are typically polymer-based, and can exhibit high compressive strengths. A graphic of a simple, unidirectional fiber-matrix composite is shown in Figure 1.

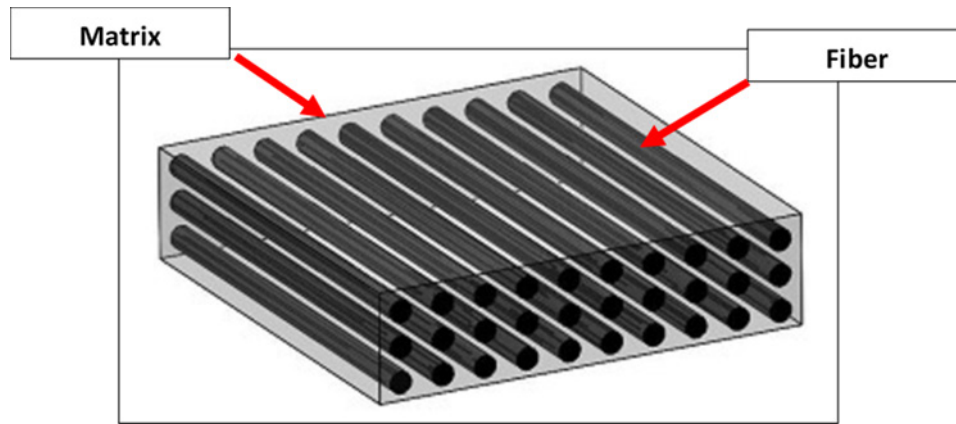


Figure 1. Graphic Displaying Basic Unidirectional Fiber-Matrix Composite Structure. Source: [1].

As previously mentioned, fiber-matrix composites can exhibit quite amazing tensile and compressive strengths relative to their material weight when compared to traditional metal alloys. To put this in perspective, a manufactured 12,000 filament carbon fiber (HexTow IM7) has a density of only 0.446 g/cm^3 , but an ultimate tensile strength of approximately 5.654 GPa (820 ksi) [2]. The manufacturer's accompanying matrix material has a density of 1.272 g/cm^3 and an ultimate tensile strength of approximately 99.28 MPa [3].

Although many fiber-matrix composite materials exhibit favorable material strengths and densities, such composites can be highly anisotropic in nature. Because the particular weave pattern of fibers in a fiber-matrix composite is determined by the manufacturer, the material may exhibit high strength, stiffness, etc. in certain loading directions; while under other loading conditions, the material may be significantly weaker and less stiff than an isotropic metal counterpart. Due to this anisotropic behavior, it is necessary to experimentally determine material properties in several orientations when

evaluating fiber-matrix composites. In Equation (1), the generalized Hooke's Law for a completely anisotropic material is shown [4].

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{21} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{31} & C_{32} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{41} & C_{42} & C_{43} & C_{44} & C_{45} & C_{46} \\ C_{51} & C_{52} & C_{53} & C_{54} & C_{55} & C_{56} \\ C_{61} & C_{62} & C_{63} & C_{64} & C_{65} & C_{66} \end{bmatrix} \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{bmatrix} \quad (1)$$

For an isotropic material, only two material constants (Young's modulus and shear modulus) are required to populate the stiffness matrix in Equation (1). However, even a simple, unidirectional fiber-matrix composite (such as that in Figure 1) requires five material constants as a result of anisotropy. It is clear that the complexity of fiber-matrix composite materials can result in variable material behaviors and failure mechanisms simply based on the direction of loading. The anisotropy of fiber-matrix materials translates into seemingly complex and unpredictable behaviors under certain loading conditions. These material behaviors have been studied and documented; however, there is still much engineering knowledge to be gained in the effort to fully characterize these materials. One such material property field of fiber-matrix composites that requires continued research is composite behavior under high strain rate conditions.

B. GENERAL FIBER-MATRIX COMPOSITE FAILURE THEORIES

As expected, the anisotropic nature and the physical composite structure of fiber-matrix composites results in failure mechanisms that are not seen in isotropic materials. Typically, the fiber material and the matrix material can be quite different with respect to strength and stiffness. This results in variable failure mechanisms that can be highly dependent on the loading conditions that a sample is subject to.

In a simplified sense, the failure mechanisms of fiber-matrix composites can be broken down into three major failure modes. The first simplified failure mode centers on fiber failure, whether it be in tension or in compression (buckling) [5]. A second failure mode involves a failure in the matrix material due to maximum principle strain [5]. The

experimental results of this thesis will prove that there is a definite and drastic difference in the failure characteristics of fiber-based and matrix-based failure mechanisms. A third, more complex failure mode exists in which the interface between the fiber and matrix fails (debonding), thus contributing to the overall specimen failure [5]. It is also possible that multiple failure modes can be expressed during any given experiment involving fiber-matrix composites—especially in complex fiber layup orientations.

The different failure modes and mechanisms of the fiber-matrix composite studied in this thesis provided results that enabled a better understanding of the high strain rate behaviors of fiber-matrix composites for naval ordnance design.

C. PREVIOUS RESEARCH CONCERNING STRAIN RATE EFFECTS IN FIBER-MATRIX COMPOSITES

Under static loading conditions, the anisotropic nature of fiber-matrix composites adds a high degree of complexity with respect to material characterization. However, an additional degree of complexity must be considered under dynamic (i.e. high strain rate) loading conditions.

Under high rates of strain and dynamic loading conditions, fiber-matrix composites can behave differently (sometimes in a drastic fashion) when compared to identical materials under static loading conditions. The aforementioned phenomenon is not particularly well-understood or documented in a manner that can be considered standard for this category of composite materials. Additionally, similar high strain rate experiments and research have produced contradictory results when evaluating nearly identical materials. As stated in Jacob, et al., certain high strain rate tests on glass-polyester fiber-matrix composites have produced up to a 55% increase in ultimate tensile strength (UTS) over static loading tests [6]. However, different experiments involving pendulum impact tests on glass-polyester composites produced rate-insensitive elastic moduli and ultimate tensile strengths at strain rates up to 4.9 m/s; lower UTS values were observed at higher strain rates [6]. To add to this variability in material behavior, experiments involving Hopkinson pressure bar tests performed on a graphite-epoxy fiber-matrix composite at

strain rates from 49 to 1430 s⁻¹ produced an increase in yield strength, but a rate-insensitive UTS and Young's modulus [6].

More relevant to the impact tests performed in this research, is the work by Hsiao and Daniel, where an impact drop tower was used to perform high strain rate tests on bi-directional carbon fiber samples. In this case, the fiber orientations were layered orthogonal to each other in order to allow high strength in orthogonal loading directions. This drop tower test produced strain rates of approximately 10 s⁻¹ [7]. Interestingly, the transverse strength results were dominated by the matrix material, but saw a nearly 37% increase in Young's modulus at higher strain rates coupled with a rate-invariant UTS [7]. Another interesting point highlighted by Hsiao and Daniel is the effect in which intermittent failure modes and damage have more time to occur under slow strain rates; whereas, high strain rates produce total material failure so quickly that the effects of these intermittent failure modes are not as pronounced [7]. The high strain rate results in Hsiao and Daniel are most comparable to those produced by the research described in this thesis—most likely due to the similarities between the materials used and the nature of the high strain rate tests.

Previous studies of high strain rate effects on fiber-matrix composites (particularly carbon fiber) allow for a better understanding of the expected results of concurrent high strain rate experiments. The results of previous studies allowed for more reasonable predictions and expectations with respect to the failure behavior of the manufacturer-supplied carbon fiber evaluated and modeled in the current research.

D. CURRENT RESEARCH OBJECTIVES AND GOALS

The results of previous research in high strain rate failure behavior of fiber-matrix composites have been relatively inconclusive, thus highlighting the necessity for further experimental data. The continued research in this thesis will be used to further refine and characterize the behavior of manufacturer-supplied carbon fiber under high strain rates, while concurrently providing valuable empirical data to improve upon existing composite finite element models.

1. Research Overview and Description

The research carried out in this thesis was guided by the necessity to characterize and model the high strain rate behavior of a specific carbon fiber material for use in potential naval ordnance casings. Materials for this research were provided by the composite manufacturing company, Dragon Plate™. Manufacturer-provided material properties for the aforementioned carbon fiber-matrix material are provided in Chapter II of this paper. The manufacturer provided three distinct layups (i.e. layering patterns) of carbon fiber material, exhibiting various degrees of anisotropy. Table 1 (below) indicates the average thickness of each carbon fiber sample (based on 8 separate measurements per sample), the number of carbon fiber weave layers, and the orientation of the fibers in each layer. The three carbon fiber layering patterns are referred to as ‘unidirectional,’ ‘bidirectional,’ and ‘quasi-isotropic.’ The quasi-isotropic specimens were layered in a manner that attempted to reduce the anisotropic properties of the material as much as possible, while maintaining material integrity.

Table 1. Carbon Fiber Specimen Weave Layups and Orientations

Specimen Type	Average Specimen Thickness (mm)	Number of Layers	Orientation Pattern of Layers (deg)
Unidirectional	1.915 mm	6	0° (all layers)
Bidirectional	1.883 mm	6	90°/0°/90°/90°/0°/90°
Quasi-isotropic	1.846 mm	12	90°/0°/45°/-45°/90°/0°/0°/90°/-45°/45°/0°/90°

In order to properly evaluate the directional behavior of each sample type, the samples were cut orthogonal to one another. For example, one set of specimens from the unidirectional sample was oriented in such a way that the carbon fibers ran parallel to an applied axial load, while orthogonal unidirectional specimens were oriented so that the fibers ran perpendicular to an applied axial load. Two sets of specimen types were cut from each carbon fiber sample. The first specimens were rectangular (approx. 76.20 mm x 12.70 mm), and were designed for high strain rate testing in a Charpy impact tester; the second

set of specimens was also rectangular (approx. 152.4 mm x 25.4 mm), and were designed for quasi-static rate testing in a tensile tester.

The primary goal of the physical experiments carried out in this research was to develop a correlation between the high and low strain rate behaviors of each of the three carbon fiber specimens in their various orthogonal orientations. In order to test the high strain rate behavior of the carbon fiber samples (and their corresponding orthogonal orientations), a standard Charpy impact tester was used in conjunction with strain gauges placed on each impact sample that provided strain readouts with respect to time. A picture of the particular Charpy impact tester used in this research is shown in Figure 2.



Figure 2. Charpy Impact Tester Used to Produce High Strain Rates

Quasi-static tests were performed using a standard 100 kN Instron tensile tester coupled with strain gauges mounted parallel to the tensile load direction. For both the high and low strain rate tests, strain readings with respect to time were collected in order to calculate strain rate. The strain at failure for each specimen was recorded. Using the aforementioned slow and high strain rate test data, a logarithmic correlation function was used to predict the failure strain of each carbon fiber sample as a function of applied strain rate.

The overall goal of the aforementioned physical experimentation was to develop a viable strain rate to failure strain correlation function for each of the three manufacturer-supplied carbon fiber layups. These correlation functions were then passed into a finite element (FE) model created using ABAQUS finite element software. The final research process involved creating a finite element model capable of accurately modeling the low and high strain rate behavior of the fiber-matrix composite material used in this research.

The finite element model relied on the versatility of ABAQUS finite element software. One of the advantages of using this software is the ability to operate FE simulations in conjunction with user-defined subroutines that can be tailored to model complex materials. The model developed in this research relied on a user subroutine (described in the following section) which was developed in [5]. The results of this FE model provided valid proof that complex fiber-matrix composite materials—with differing fiber-matrix layup orientations—can be accurately modeled under high strain rate conditions for future use in naval ordnance designs.

2. Finite Element Model Subroutine Description

The FE model subroutine that was used in this research was created in a previous effort to simulate the static loading behavior of fiber-matrix composites of multiple orientations. As stated in [5], the user subroutine takes in data from the ABAQUS finite element model at each time, and performs a multitude of material and damage calculations tailored specifically for fiber-matrix composite materials. The subroutine results at each time step are then passed back into the ABAQUS model [5]. The subroutine uses representative volume elements that assign either fiber or matrix material properties to subcells of the volume element based on fiber-matrix volume fractions (element divisions are referred to as “subcells”) [5]. Each representative volume element combination is referred to as a ‘unit cell,’ and it is important to note that the subroutine works by establishing continuity between fiber-matrix subcells within each unit cell [5]. Additionally, the total strain of each unit cell is defined by the sum of each subcell’s calculated strain multiplied the relevant volume fraction [5].

The user subroutine relies on intermittent failure calculations that lead up to final specimen failure. As the FE model iterates, the subroutine calculates stiffness reductions that mimic intermittent damage prior to final failure. To accomplish this, a 3x3 matrix is used, in which the rows represent failed subcells and the columns represent the failure direction (a zero value in the matrix indicates no damage) [5]. The overarching principle of the user subroutine involves scaling. The damage calculations for each unit cell are scaled up to the full model using simple linear algebra techniques; conversely, overall stresses and strains are downscaled to each unit cell per FE timestep [5].

The validity of the subroutine under static loading conditions was proved in Ref. [5] through the modeling of E-Glass fibers with LY556 resin. The user subroutine model results—under static loading conditions—correlated with experimental data as shown in the example in Figure 3.

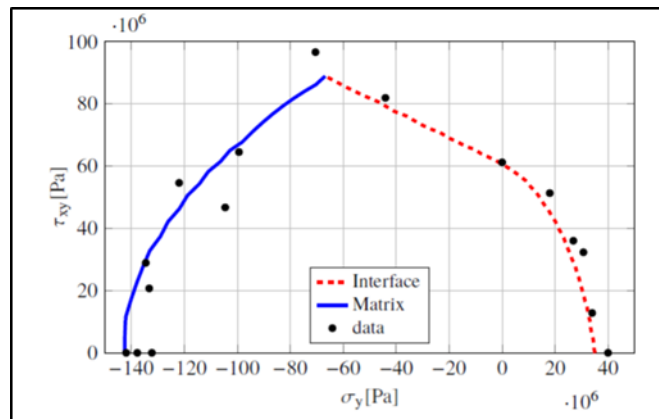


Figure 3. Example of FE Result Correlation with Experimental Data for E-Glass/LY556 Composite. Source: [5].

Based on the results in Ref. [5], this user subroutine was determined to be the most useful and accurate method for modeling the high strain rate behavior of the carbon fiber samples used in the current research. The high and low strain rate experimental data, coupled with the versatility of the user subroutine allowed for the complex and anisotropic behavior of carbon fiber (and potentially other fiber-matrix composites) to be accurately modeled for future use in naval ordnance casings.

THIS PAGE INTENTIONALLY LEFT BLANK

II. SPECIMEN AND EXPERIMENT PREPARATION

Manufacturer-supplied carbon fiber samples were provided in three separate composite layups. Material samples were provided in the form of 304.8 mm (12 in) by 304.8 mm plates with a typical thickness of approximately 1.90 mm. High and low strain rate test specimens were cut from each plate, while strain gauges were mounted along each specimen's long axis. High strain rate tests were conducted using a Charpy impact tester coupled with a National Instruments, PXIe-6358 data acquisition system. Quasi-static tests were performed through a standard, 100 kN tensile testing apparatus coupled with a low data rate National Instruments data acquisition system.

A. SUPPLIED MATERIAL OVERVIEW

The carbon fiber manufacturer, Dragon Plate, supplied several carbon fiber plates of three distinct composite layup structures. A picture of one of the unidirectional carbon fiber plates used in this research is shown in Figure 4.



Figure 4. Unidirectional Carbon Fiber Plate (Manufacturer-Supplied)

The three provided composite layup structures; unidirectional, bidirectional, and quasi-isotropic, expressed a decreasing degree of material anisotropy, respectively. Figure 5 displays an overview of the composite layup structure for each of the three sample types.

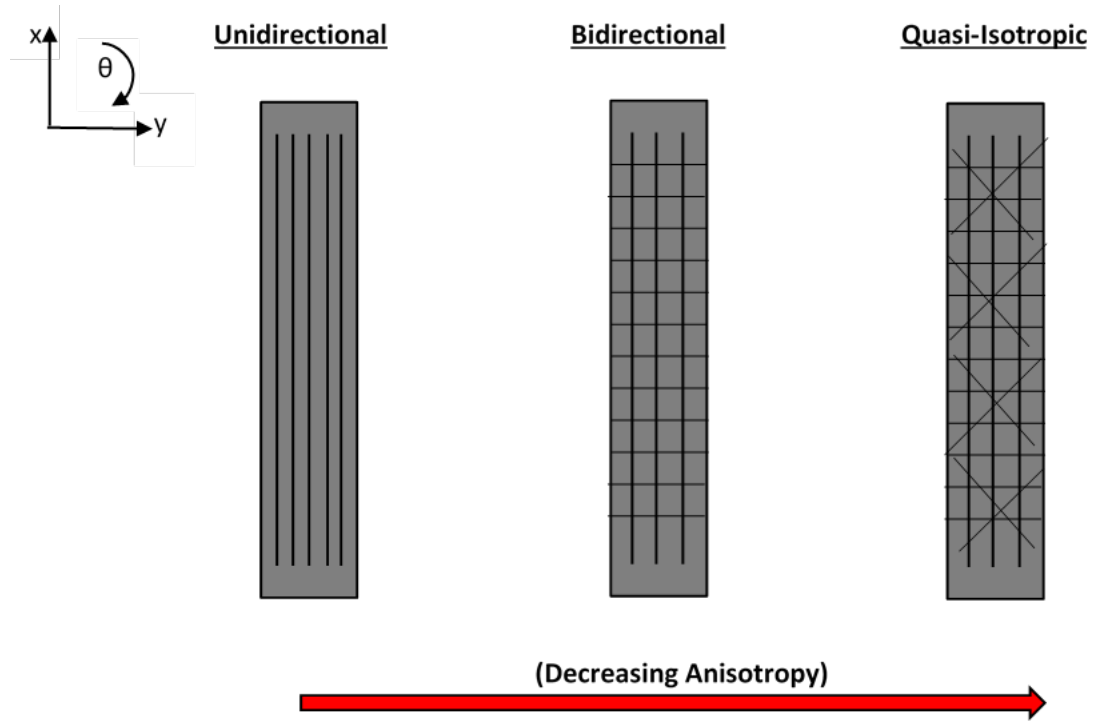


Figure 5. Specimen Composite Layup Structure – Overview

The composite layup examples shown above display a two-dimensional representation of the full structure of each provided material. In actuality, differing fiber orientations were layered on top of one another to create each composite layup. As shown in Table 1, the unidirectional and bidirectional samples were six-layered, while the quasi-isotropic samples were 12-layered. The material thickness remained approximately consistent across the three sample types, thus, each individual layer of the quasi-isotropic samples were half of the thickness of the unidirectional and bidirectional layers. For the purpose of consistency, the various fiber orientation angles were measured in degrees taken clockwise from the y-axis shown in Figure 5.

In addition to the orientation angles and layering schematics, the manufacturer supplied specifications for the fiber and matrix materials. Table 2 (below) displays the material names and various properties of the relevant fiber and matrix material.

Table 2. Fiber and Matrix Material Property Information. Adapted from [8] and [9].

	Fiber	Matrix
Material	Torray T700s	Newport 301
Tensile Strength, Ultimate	4,900 MPa	57.23 MPa
Young's Modulus	230.0 GPa	3.172 GPa
Density	1.80 g/cm ³	(not provided)
Strain at Failure	2.1%	(not provided)
Fiber Diameter	7 μ m	N/A

B. TEST SPECIMEN PREPARATION

As stated, two separate specimen types were created from the carbon fiber composite layup samples described in the preceding section. The first set of specimens were machined for use in high strain rate impact testing; the second set was machined for tensile testing at a low strain rate. All specimens were cut in accordance with the geometric specifications in Table 3 (thickness determined by manufacturer).

Table 3. High and Low Strain Rate Sample Geometries

	Impact Test Samples	Tensile Test Samples
Length (long axis)	76.20 mm (3 in)	152.4 mm (6 in)
Width (short axis)	12.70 mm (0.5 in)	25.4 mm (1 in)
Thickness (manufactured)	~1.900 mm	~1.900 mm

Every impact test (high strain rate) and tensile test (low strain rate) specimen was machined from the same carbon fiber plate for each of the three composite layups. In order to appropriately evaluate and categorize the anisotropic behavior of the three carbon fiber layups, impact samples and tensile samples were divided into two subsets. The specimen subsets were cut orthogonal to one another, thus allowing different loading conditions to be tested without having to modify the direction of the applied force in the high and low strain rate tests. For consistency, a specimen nomenclature was developed to keep track of the various orthogonal specimens and composite layups. Orthogonal Unidirectional specimen subsets were named ‘U1’ and ‘U2,’ respectively. The bidirectional—‘B1’ and ‘B2’—and quasi-isotropic—‘QI1’ and ‘QI2’—specimen subsets followed the same nomenclature. Table 4 (below) provides layer orientation information for each of the six specimen subsets. Note: layer orientation degrees were taken clockwise from the positive y-axis shown in Figure 5.

Table 4. Carbon Fiber Layer Orientations for Each Specimen Subset

Specimen	Number of Layers	Layer Orientation
U1	6	0°/0°/0°/0°/0°/0°
U2	6	90°/90°/90°/90°/90°/90°
B1	6	0°/90°/0°/0°/90°/0°
B2	6	90°/0°/90°/90°/0°/90°
QI1	12	0°/90°/-45°/45°/0°/90°/90°/0°/45°/-45°/90°/0°
QI2	12	90°/0°/45°/-45°/90°/0°/0°/90°/-45°/45°/0°/90°

All test specimens were machined in a rectangular shape. The standard ‘dog bone’ tensile specimen shape was avoided for the low strain rate specimens due to the difficulty inherent in the machining of carbon fiber. During the machining process, laminate separation occurred on several samples, thus rendering them useless for testing; therefore, only simple rectangular test specimens were used in order to avoid specimen damage prior to testing. The impact test samples were tested as-machined, but the tensile test samples required additional modifications in order to comply with the tensile tester clamping mechanism. Due to the relatively brittle nature of carbon fiber composite, it was necessary to create tabs out of a soft metal that could allow the tensile tester clamps to hold the test

specimens in place without any significant stress concentrations. Said tabs were machined out of aluminum in a 25.4 mm by 25.4 mm geometry with a 45 degree taper at the leading edge. The tabs were secured to each tensile sample with high-strength epoxy. A diagram of a tensile specimen with the aforementioned tabs is shown in Figure 6.

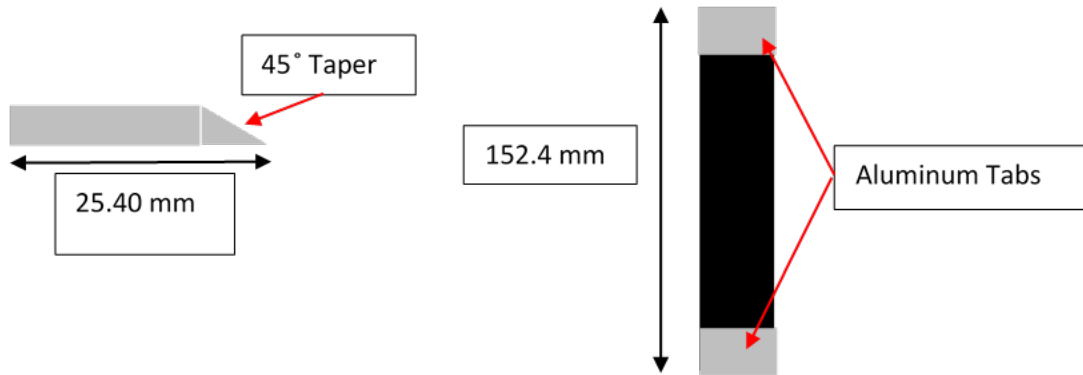


Figure 6. Aluminum Tab Schematic for Reduction of Stress Concentrations in Tensile Testing

Strain measurements were the primary form of data collected in both the tensile and impact tests. Therefore, it was necessary to apply strain gauges to each sample. The relevant strain gauge specifications are shown in Table 5.

Table 5. Strain Gauge Specifications

Gauge Type	Linear
Gauge Factor	2.13
Resistance	350 Ohms

Strain gauges were mounted to each test specimen on the outer most layer exposed to tensile bending or uniaxial stress. Charpy impact testing produces a tensile bending stress on the outer layer of an impact sample relative to the point of contact with the impact hammer; therefore, strain gauges were placed on said outer layer running parallel to the

specimen long axis. This particular strain gauge placement allowed for the ϵ_{11} strain tensor component (relative to the axis shown in Figure 5) to be measured on the outermost composite layer of each impact sample. Strain gauge placement on the quasi-static specimens was less restrictive since the tensile tests produced uniaxial stresses on both of the outer composite layers of each specimen. To remain consistent with the impact test strain measurements, strain gauges were again mounted along the long axis of each quasi-static specimen, parallel to the ϵ_{11} strain tensor component. Additional strain gauges were mounted orthogonal to the long axis gauges on each quasi-static specimen; however, the strain readings from said gauges were unreliable and inconsistent. Figure 7 displays images of strain gauges mounted on the high strain rate (left) and quasi-static (right) specimens.

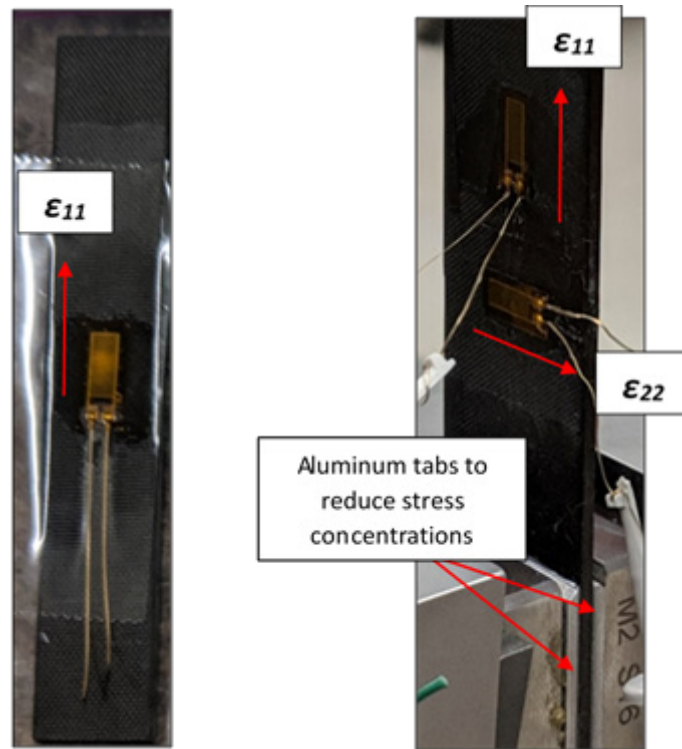


Figure 7. Strain Gauge Mounting for High Strain Rate (left) and Low Strain Rate (right) Specimens

C. HIGH AND LOW STRAIN RATE TEST SETUP

Setting up the high and low strain rate testing equipment required careful attention to detail in order to ensure appropriate and relevant data were acquired—especially in the high strain rate case since relevant data sets were on a time scale of 3 ms at a maximum.

1. High Strain Rate (Impact) Test Setup

High strain rate tests were performed using a Tinius Olsen Charpy impact apparatus, coupled with strain gauge data obtained via a high rate data acquisition system. It was initially determined that the high strain rate tests were to be divided into three separate strain rates of increasing magnitude for each specimen layup and orientation. To accomplish this, the angle of the impact test hammer was varied. To achieve consistent impact hammer angles, a laser indicator was secured to a specified location on the impact hammer arm. The laser indicator was then projected onto a black grid placed directly behind the impact hammer arm. Simple geometric calculations were made in order to correlate the laser indicator height on the aforementioned grid with the angle of the impact hammer. Images of the aforementioned impact tester and laser indicator setup are shown in Figure 8.

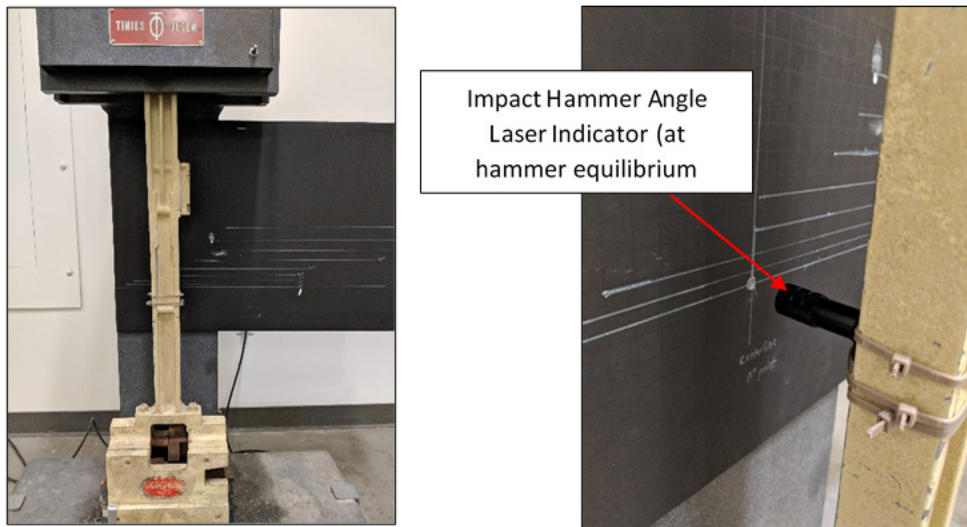


Figure 8. Image of Charpy Impact Tester (left) and Impact Hammer Angle Measurement Setup (right)

In order to collect a sufficiently-large data set for each impact test, a high rate data acquisition (DAQ) system was required. The DAQ system used in this experimental setup was a National Instruments (NI) PXIe-6358 data acquisition card mounted to an NI PXIe-8821 chassis. The Omega, high precision strain gauges used during impact and tensile testing required an excitation voltage of approximately 10 V. The PXIe-6358 system does not have an integrated power supply capability; therefore, it was necessary to supply an excitation voltage via a Wheatstone quarter-bridge apparatus created by CDR Joseph Darcy (PhD), U.S. Navy. An input/output cable was run from the DAQ system to an NI input/output board, from which input/output, excitation voltage, and ground wires were routed to the relevant ports on the Wheatstone quarter-bridge. The opposing ports on the Wheatstone quarter-bridge were then connected to the actual strain gauge wires that routed the excitation voltage and input/output signals to and from the gauge itself. Images of the Wheatstone quarter-bridge and the NI input/output board are shown in Figure 9.

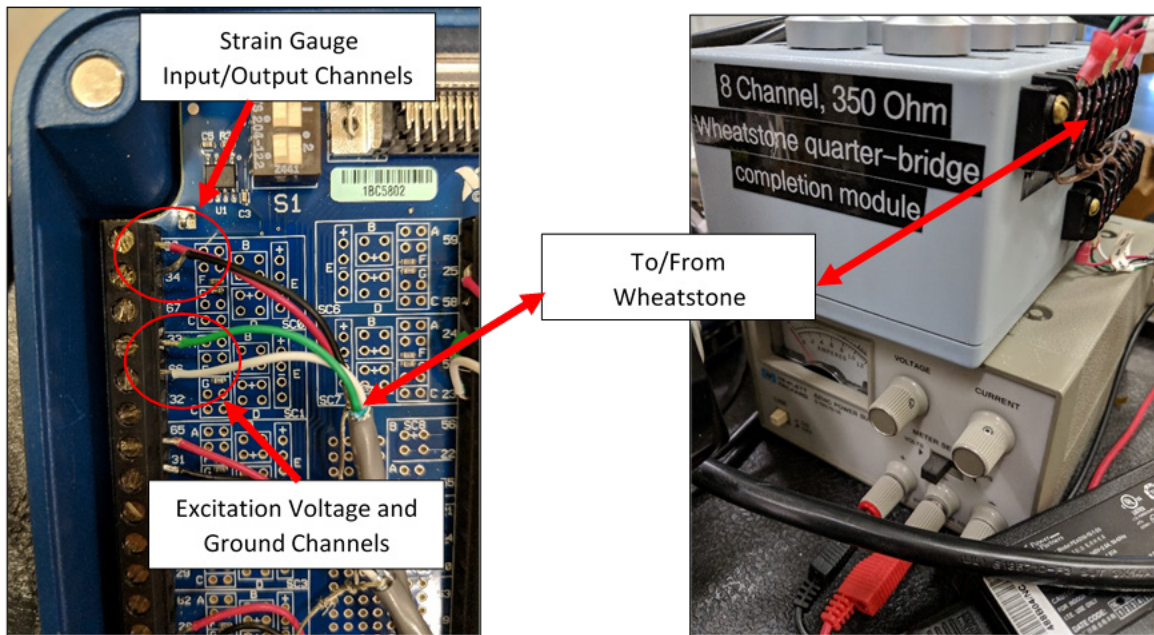


Figure 9. Wiring Configuration for Impact Test Strain Gauges

Figure 10 displays a specimen placed in the impact tester and the appropriate strain gauge wiring configuration.



Figure 10. Example of Test-Ready Impact Specimen and Strain Gauge Wiring

For each impact test, the supplied voltage was held constant at approximately 10 V. Minor resistance adjustments were made prior to each impact test in order to zero the strain gauges readings. Due to the strain gauge mounting procedure, small residual strain readings were observed prior to each impact test, thus zeroing each gauge was deemed necessary. The data acquisition rate used in each impact test varied between 100 kHz and 500 kHz on a per-specimen basis due to the wide range of strain rates that were observed.

A total of 18 quasi-isotropic impact specimens were tested (9 specimens for each orthogonal orientation), and 20 bidirectional and unidirectional impact specimens were tested (10 per orthogonal orientation). Specimen subset QI1 was the first set of impact specimens to be tested at impact hammer angles of 20°, 30°, and 45°. After gathering the relevant strain data from the QI1 specimen set, the impact hammer angle spread was increased to 20°, 45°, and 75° for the remaining impact specimen sets. A table listing the number of impact specimens tested, the data acquisition rates used, and the supplied impact hammer angles is shown in Table 6.

Table 6. Impact Test Specimens, Impact Hammer Angles, and Data Acquisition Rates

Specimen Name	# Specimens Tested	Data Acquisition Rate (Hz)	Impact Hammer Angles (°)
U1	10	100 kHz (1-4), 300 kHz (5-10)	20°(3), 45°(3), 75°(4)
U2	10	100 kHz (1-3), 300 kHz (4-6), 400 kHz (7), 500 kHz (8-10)	20°(3), 45°(3), 75°(4)
B1	10	100 kHz (1-6), 300 kHz (7-10)	20°(3), 45°(3), 75°(4)
B2	10	100 kHz (1-6), 300 kHz (7-10)	20°(3), 45°(3), 75°(4)
QI1	9	100 kHz (all)	20°(3), 30°(3), 45°(3)
QI2	9	100 kHz (1-6), 300 kHz (7-9)	20°(3), 45°(3), 75°(3)

2. Quasi-Static (Tensile) Test Setup

Quasi-static testing was performed using a simple tensile testing apparatus with a 2 mm/s extension rate and a 100 kN maximum tensile load. As aforementioned, the Instron tensile tester utilized a clamping mechanism that would have created data-altering stress concentrations on a brittle material like carbon fiber. Therefore, aluminum tabs were secured to each quasi-static specimen to avoid this stress concentration issue (refer to Figures 6 and 7). The quasi-static tests did not require the same robust DAQ capabilities as the high strain rate tests; therefore, a less complex NI cDAQ-9174 system as shown in Figure 11.



Figure 11. Image of Quasi-Static DAQ System

As shown in Figure 7, each quasi-static specimen had two orthogonal strain gauges mounted. The lateral strain gauges were mounted in an attempt to obtain Poisson's ratio information from the specimens; however, these lateral strain data proved unreliable. With respect to the actual tensile testing machine, a constant displacement rate of 2 mm/s was stipulated for each test. The tensile tester was fitted with its own onboard data acquisition system, so load versus deflection data were obtained separate from the strain readings provided by the mounted gauges. The strain gauges fed strain readings directly into the NI cDAQ-9174 system, which was integrated with its own software in order to display and record the relevant data. The slowest possible data acquisition rate of approximately 2 kHz was used during quasi-static testing; this lead to rather large data files. Two quasi-static specimens from each composite layup type and orthogonal orientation were manufactured, thus leading to a total of 12 quasi-static specimens. All quasi-static tests were carried out using the same tensile test parameters and data acquisition parameters, and each tensile test was carried out to *total* specimen failure. In some instances, intermittent failures like laminate separation occurred prior to final failure.

D. ADDITIONAL TEST SETUP FOR IMPACT LOAD READINGS

Upon completion of high and quasi-static testing, it was deemed necessary to obtain impact load data (with respect to time) from the Charpy impact hammer. The impact load data for each impact specimen type and orientation were used as the input force column vectors during FE modelling. To obtain load versus time data for each impact specimen, a Honeywell Model 13 Subminiature Compression load cell was used in conjunction with the PXIe-6358 DAQ card. The load cell specifications provided by the manufacturer are displayed in the following table:

Table 7. Load Cell Specifications. Adapted from [10].

Maximum Load (N)	Calibration Factor (mV/V)	Excitation Voltage (V)	Voltage to Force Conversion Factor (N/V)
2,224.11 N	1.9174 mV/V	5 V	232,130.44 N/V

The load cell impact tests were setup and performed in a similar manner to the high strain rate impact tests. Four impact specimens from specimen subsets QI1 and QI2 were subject to load cell impact testing; while six impact specimens from subsets U1, U2, B1, and B2 were load-impact tested. In each case, a data acquisition rate of 200 kHz was used, and every load impact test utilized a 45° impact hammer angle (a single angle was chosen due to a lack of variability in the results of the high strain rate tests). The required excitation voltage (specified by the load cell manufacturer) during load-impact testing was 5 V.

In addition to integrating the load cell with the PXIe-6358 DAQ system, an adapter had to be manufactured in order to secure the load cell to the impact hammer. A structural steel sheath that fit over the actual Charpy impact device was machined, and the load cell was fitted to the sheath with high strength metal-bonding epoxy. This setup is shown in Figure 12:

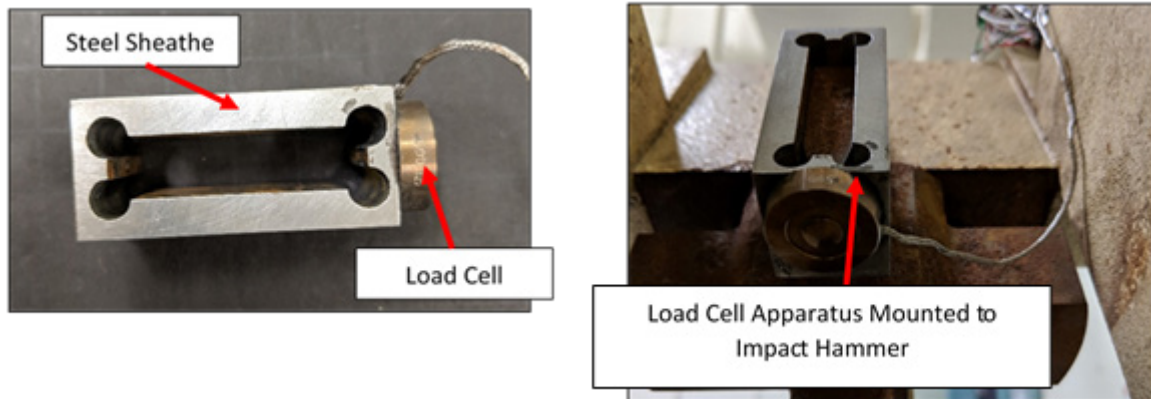


Figure 12. Images of Load Cell Apparatus and Impact Hammer Integration

The load shown in Figure 12 was positioned in such a way that the load cell actuator impacted the center of each impact specimen. The load cell produced raw data in the form of voltage readings. In order to convert the voltage data to units of Newtons, a voltage to force conversion factor of 232130.44 N/V (see Table 7) was used. This conversion allowed force versus time data to be recorded for each load-impact test.

Eight additional strain gauge impact tests were performed on impact specimen subset B1 in order to verify the load cell contact surface was producing similar failure

results and strain rates when compared to the contact surface of the Charpy impact hammer. For these validation tests, the load cell remained fixed to the impact hammer, but the PXIe-6358 system was used to take strain readings from strain gauges mounted on the relevant impact specimens. Strain gauges for the validation impact tests were mounted in an identical manner to the previous impact tests. The validation impact tests used a 200 kHz data acquisition rate and an impact hammer angle of 45°.

THIS PAGE INTENTIONALLY LEFT BLANK

III. HIGH STRAIN RATE TEST RESULTS

All high strain rate (impact) tests were carried out in accordance with the parameters and experimentation equipment described in Chapter II, Section C.1. All strain data obtained during impact testing were processed, plotted, and analyzed using MATLAB software. Appendix A provides an example MATLAB script that outlines the data processing procedure for impact specimen subset, U1. An identical data processing method was used for every impact specimen subset. The aforementioned impact specimens were measured prior to testing, and any excess material from the machining process was sanded off, thus ensuring appropriate clearance with the impact hammer apparatus. Prior to any recording of data, each specimen was placed in the Charpy impact tester, and each strain gauge was wired and supplied with the required 10 V excitation voltage. Each specimen was then deflected slightly, and any strain readings were observed. This process ensured each strain gauge was functioning properly prior to carrying out destructive testing.

A. UNIDIRECTIONAL SPECIMEN HIGH STRAIN RATE RESULTS

The following results display the high strain rate data obtained during Charpy impact testing.

1. Impact Specimen Subset U1

All ten specimens from subset U1 were tested in the Charpy impact tester (refer to Table 6 for specific test parameters). The composite layers of each specimen in subset U1 had their fibers oriented parallel to the expected tensile bending stress produced during impact testing; therefore, a high stiffness along the strain gauge measurement direction was expected due to the favorable fiber material properties. Figure 13 displays the strain readings (with respect to time) from the 20°, 45°, and 75° impact hammer angle tests. Plotted data were cutoff at the point at which the strain versus time curve diverged significantly from linearity. The diverging region of the strain versus time curve produced strain readings greater than 1, thus indicating failure of the strain gauges.

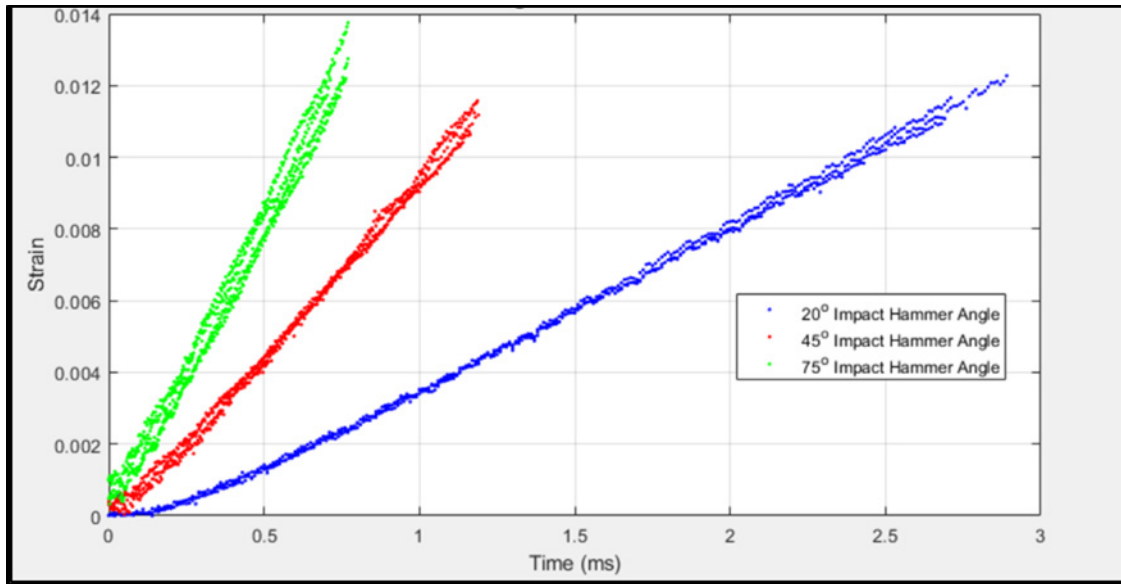


Figure 13. Strain versus Time Curves for Specimen Subset U1 – High Strain Rate

The results of impact testing on subset U1 produced linear curves that indicate varying high strain rates (based on the supplied impact hammer angle) and failure strains ranging from approximately 0.0112 to 0.01375. The differing strain rates are easily observable in Figure 13 based on the slopes of each strain versus time curve. Failure strains for each test were taken to be the final strain value prior to significant divergence from linearity of each strain versus time curve. Strain rates were determined by applying a linear curve fit to each data set, and extrapolating the determined slope. The mean failure strain and mean strain rate were calculated for the data sets pertaining to each impact hammer angle.

2. Impact Specimen Subset U2

An identical impact test procedure to specimen subset U1 (with the same impact hammer angles) was applied to impact specimen subset U2. Based on the fiber orientation and composite layering in subset U2, the fibers in each layer ran parallel to the expected tensile bending stress on the outer layer of each specimen. Therefore, the majority of mechanical stress was expected to be taken on by the matrix material. The strain versus

time data for each specimen subset U2 impact test (at the specified impact hammer angles) are shown in Figure 14.

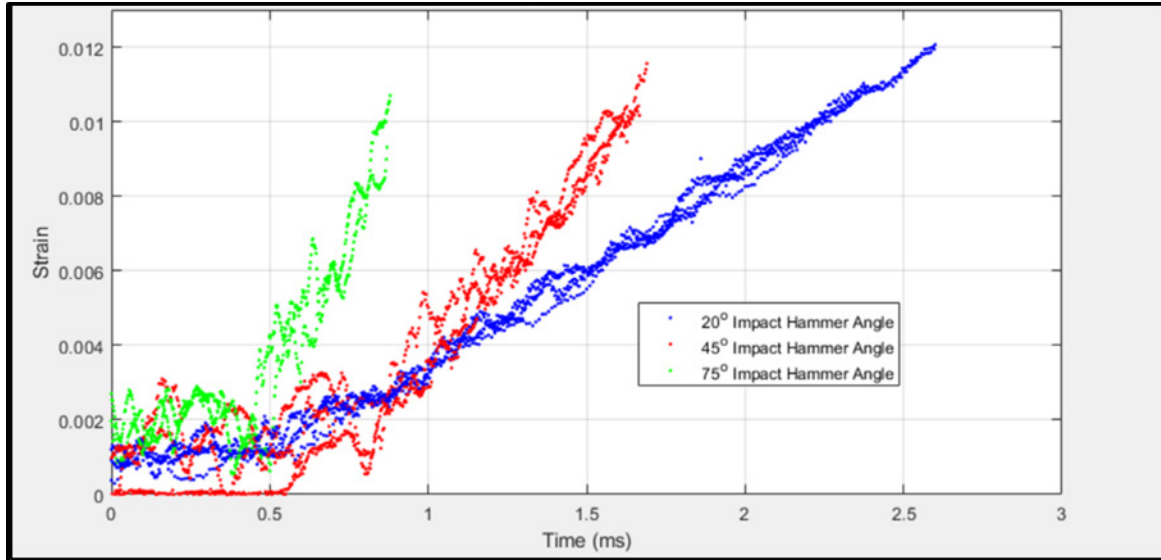


Figure 14. Strain versus Time Curves for Specimen Subset U2– High Strain Rate

The strain versus time curves in Figure 14 are not as distinctly linear as those in Figure 13. This is likely due to the markedly different material properties of the matrix material compared to carbon fiber. Several of the impact tests on specimen subset U2 did produce clear indications of the onset of impact; therefore, the curves in Figure 14 are not zeroed on a distinct start point. Instead, each curve was adjusted so that the slopes of each hammer angle group lined up for better visualization. The final two impact tests performed on specimen subset U2 at a hammer angle of 75° produced no meaningful data, and were removed from the data analysis. The most linear region of each of curve in Figure 14 was used to calculate a linear curve fit for strain rate approximation. Again, the failure strain was taken to be the point in each data set that strain readings diverged significantly from linearity. All calculated strain rates and failure strains were averaged in groups based on the designated impact hammer angle.

3. Interpretation of High Strain Rate Results – Subsets U1 and U2

The high strain rate behaviors of the unidirectional impact specimens were highly dependent on their machined orientations. Specimen subset U1 placed most of the impact load and associated stress on the specimens' carbon fibers, while the orthogonal orientation, U2, placed a majority of the impact load and stress on the matrix material. Simply orienting an identical material by 90° allowed for vastly different strain behavior under Charpy impact conditions. The matrix-dominated specimens produced notable fluctuations in ϵ_{11} strain readings, potentially due to the propagation of stress waves in the material during impact. These fluctuations were not observed to any extent in the fiber-dominated specimens. Additionally, the matrix-dominated impact specimens produced a distinct nonlinear strain versus time response at strain values less than 0.002; this initial nonlinearity was not observed in the fiber-dominated specimens.

Despite apparent differences in the stress versus time curves of specimen subsets U1 and U2, the actual failure strains and strain rates were relatively consistent and on the same order of magnitude. Figure 15 displays a comparison of the failure strain as a function of supplied strain rate for impact subsets U1 and U2 (strain rates are based on the three impact hammer angles used).

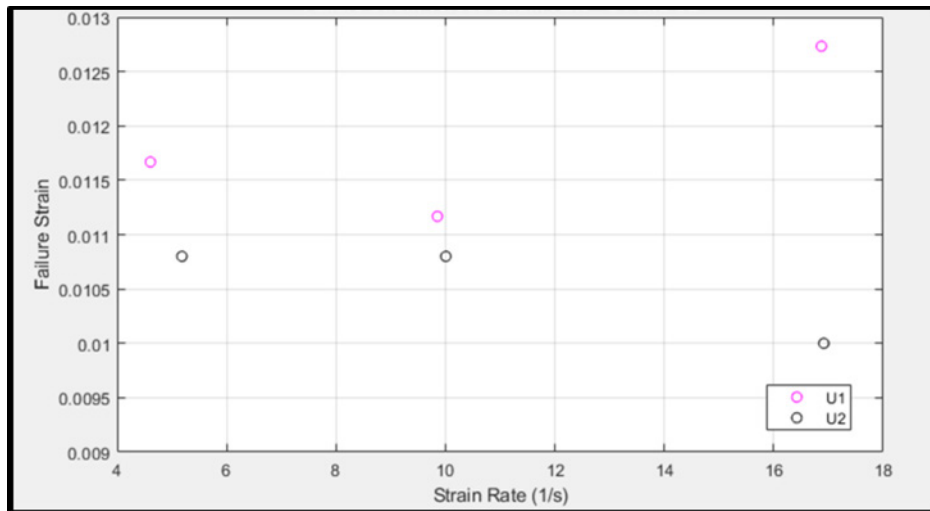


Figure 15. Failure Strain as a Function of Supplied High Strain Rate – U1 and U2

As displayed in Figure 15, the calculated high strain rates at each impact hammer angle were consistent between the two unidirectional composite layup orientations. The mean failure strain of each matrix-dominated impact specimen (U2) was less at every strain rate value, with the greatest disparity occurring at the highest strain rate of approximately 17 s^{-1} . Visually inspecting the failure strain distribution of specimen U1, there appears to be no distinct pattern that could indicate a rate-dependent failure strain increase or decrease. The mean failure strain of specimen subset decreased as the supplied strain rate was increased. This behavior is contradictory to the expected result. In Hsiao and Daniel, drop tower tests that produced similar bending stress conditions and strain rates of approximately 10 s^{-1} yielded an increase in Young's modulus and rate-insensitive UTS [7]. A number of previous high strain rate research involved a comparison of fiber-matrix material behaviors under strain rates of different orders of magnitude. The strain rates for both subsets U1 and U2 ranged from approximately 4.5 s^{-1} to 17 s^{-1} as the impact hammer angle was increased from 20° to 75° . The Charpy impact tester produced only a limited strain rate range; additionally, the average failure strains as a function of supplied strain rate for subsets U1 and U2 did not produce a distinct, functional relationship. Therefore, the strain rates and failure strains of for all three impact hammer angles of subsets U1 and U2 were averaged. This allowed for the calculation of a single high strain rate and associated failure strain for impact specimen subsets U1 and U2. Narrowing down the scope of the high strain rate data allowed for a single calculation to be made (using Equation (2)) that generated a logarithmic relationship between failure strain and the supplied strain rate spanning several orders of magnitude. For the purpose of consistency, the same mean failure strain and associated high strain rate calculation was applied to the high strain rate data produced by specimen subsets B1, B2, QI1, and QI2.

B. BIDIRECTIONAL SPECIMEN HIGH STRAIN RATE RESULTS

The same strain rate test parameters and experimental conditions were maintained when performing high strain rate tests on subsets B1 and B2. Table 6 gives further reference on data rates, impact angles, and the number of bidirectional impact specimens tested.

1. Impact Specimen Subset B1

Impact specimen subset B1 was loaded in accordance with a layer orientation of $[0^\circ/90^\circ/0^\circ/0^\circ/90^\circ/0^\circ]$, with 0° indicating fibers oriented parallel to the ε_{11} strain tensor element. The impact specimens in subset B1, therefore, contained twice as many composite layers with fibers oriented parallel to the measured strain tensor component and the direction of primary impact bending stress. Due to this particular composite layup, it was predicted that the specimens in subset B1 would produce higher failure strains when compared to the orthogonal subset B2. For the specimen subset B1, high strain rate test results at impact hammer angles of 20° , 45° , and 75° are shown in Figure 16.

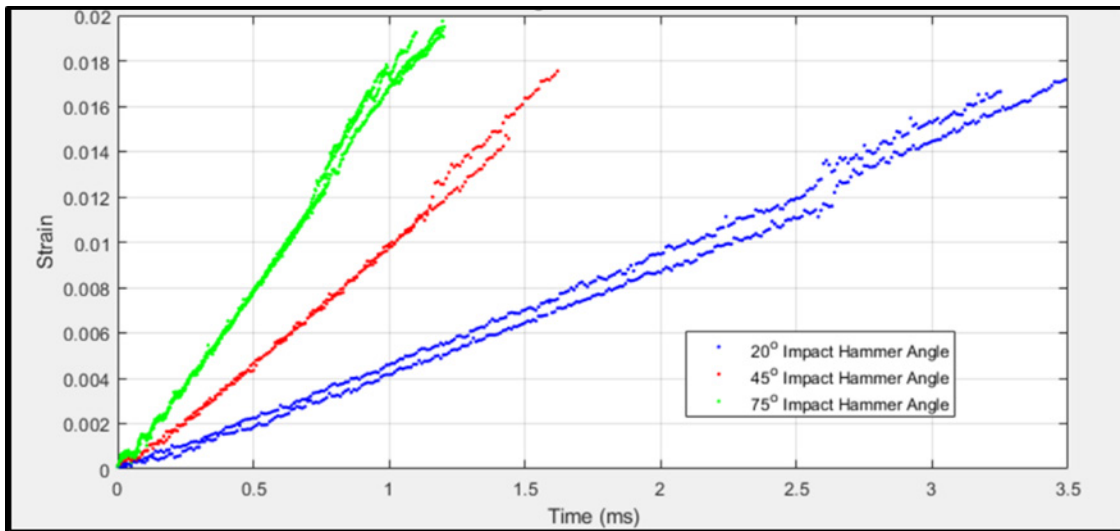


Figure 16. Strain versus Time Curves for Specimen Subset B1– High Strain Rate

Subset B1 high strain rate tests for the specimen number 2, 4, and 9 were omitted due to early strain gauge delamination and failure that prevented complete and accurate data capture. The high strain rate results of specimen subset B1 produced higher overall failure strains when compared to subset U1. Additionally, the B1 test duration at each impact hammer angle was slightly longer when compared to the unidirectional, fiber-dominated impact tests. At an extreme impact angle of 75° , bidirectional subset, B1 produced a nearly 0.006 increase in failure strain when compared to unidirectional subset,

U1. This apparent increase in ductility may have been a result of a greater distribution of the impact bending stress and load between the fiber and matrix-dominate composite layers. High strain rates and failure strains for bidirectional subset B1 were determined in an identical manner to the unidirectional subsets.

2. Impact Specimen Subset B2

Impact specimen subset B2 was loaded in accordance with a layer orientation of $[90^\circ/0^\circ/90^\circ/90^\circ/0^\circ/90^\circ]$, with 0° indicating fibers oriented parallel to the ε_{11} strain tensor element. The impact specimens in subset B2 contained twice as many composite layers with the *matrix* material oriented parallel to the measured strain tensor component and the direction of primary impact bending stress. For the specimen subset B2, high strain rate test results at impact hammer angles of 20° , 45° , and 75° are shown in Figure 17.

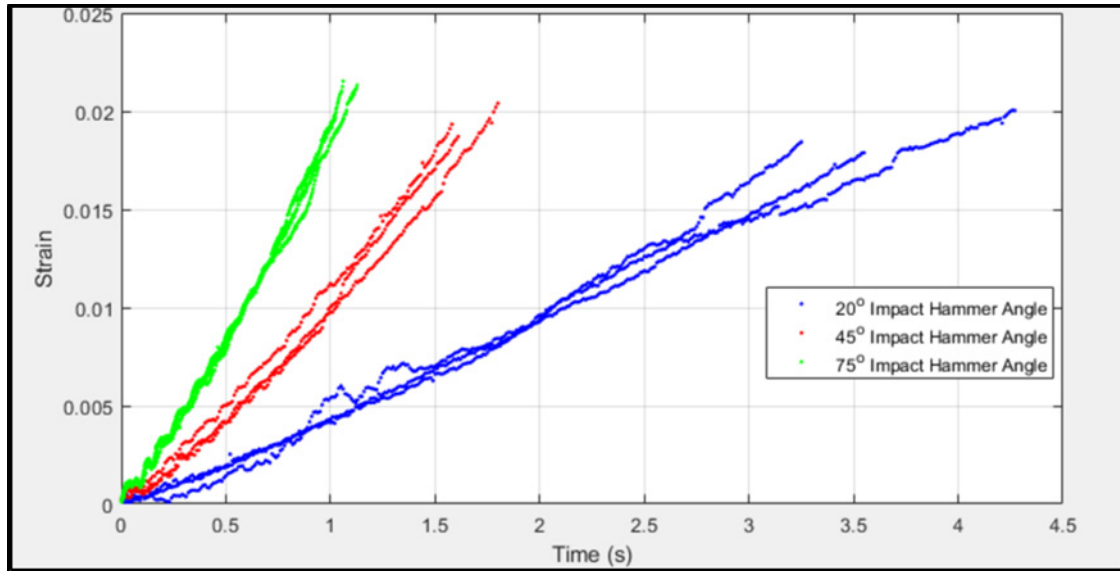


Figure 17. Strain versus Time Curves for Specimen Subset B2 – High Strain Rate

Contrary to the expected result, bidirectional subset B2 produced failure strains that were, in general, greater than bidirectional subset B1. Simply measuring strain along one strain tensor direction, however, does not completely reveal the mechanical failure behavior of a fiber-matrix composite material. As shall be shown in the proceeding

sections, the impact load data and stress versus strain data at low strain rates revealed notable failure characteristic differences between the orthogonal bidirectional specimen orientations. One of the 20° hammer angle impact tests on subset B2 displayed fluctuations in strain readings that were similar to those of unidirectional subset U2. Again, strain rates for each B2 impact test were calculated using linear curve fitting, and the failure strains were taken to be the point at which strain readings significantly diverged from linearity.

3. Interpretation of High Strain Rate Results – Subsets B1 and B2

The orthogonal orientation of the bidirectional impact specimens appeared to have less of an effect on the observed high strain rate behavior when compared to the unidirectional impact specimens. This was likely because only 2/3 of the fiber-dominated composite layers were replaced by matrix-dominated layers (with respect to the ϵ_{11} strain tensor component) when rotating subset B1 90° to subset B2. This is in contrast to the unidirectional impact specimens, in which every composite layer switched from fiber-dominated to matrix-dominated characteristics parallel to the ϵ_{11} strain tensor component. Similar to the results of unidirectional subsets U1 and U2, the orthogonal B1 and B2 specimen subsets produced strain rates and failure strains (at each impact angle) that were similar in magnitude. Figure 18 displays a comparison of the failure strain and supplied strain rate for impact subsets B1 and B2.

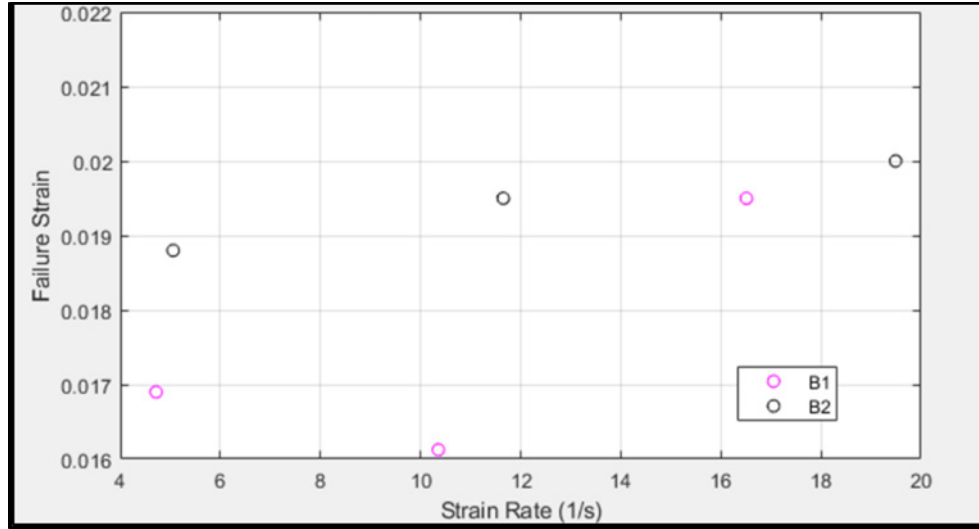


Figure 18. Failure Strain as a Function of Supplied High Strain Rate – B1 and B2

As shown in Figure 18, the strain rates and failure strains of bidirectional subsets B1 and B2 are relatively close in magnitude at each of the three impact hammer angles. Interestingly, at highest impact angle of 75° , subset B2 produced a mean strain rate that was nearly 2 s^{-1} greater than subset B1 at the same impact angle. The two lower angle strain rate values did not differ by as significant of an amount. Following the same criteria as the unidirectional impact specimen subsets, the three strain rate and failure strain values for subsets B1 and B2 were averaged to produce a single high strain rate and failure strain value for each orthogonal subset.

C. QUASI-ISOTROPIC SPECIMEN HIGH STRAIN RATE RESULTS

Quasi-isotropic specimen subsets, QI1 and QI2 were subject to the same impact test setup as the bidirectional and unidirectional subsets. Subset QI1 was the first impact specimen to be tested, therefore, a smaller impact angle variation of $20^\circ - 30^\circ - 45^\circ$ was used. After determining this impact angle spread was too narrow, the variation was increased to $20^\circ - 45^\circ - 75^\circ$ (refer to Table 6 for more impact test parameters).

1. Impact Specimen Subset QI1

As shown in Table 4, both quasi-isotropic subsets were manufactured in such a way that the composite layer fiber orientations were evenly distributed so that no single orientation angle was assigned to a greater number of composite layers. Because of this quasi-anisotropy, the composite layer orientations of subsets QI1 and QI2 were *identical* despite the specimen subsets orthogonal machining orientation. In theory, this quasi-isotropic manufacturing should produce near-identical high strain rate failure characteristics at equivalent impact hammer angles. For the specimen subset QI1, high strain rate test results at impact hammer angles of 20°, 30°, and 45° are shown in Figure 19.

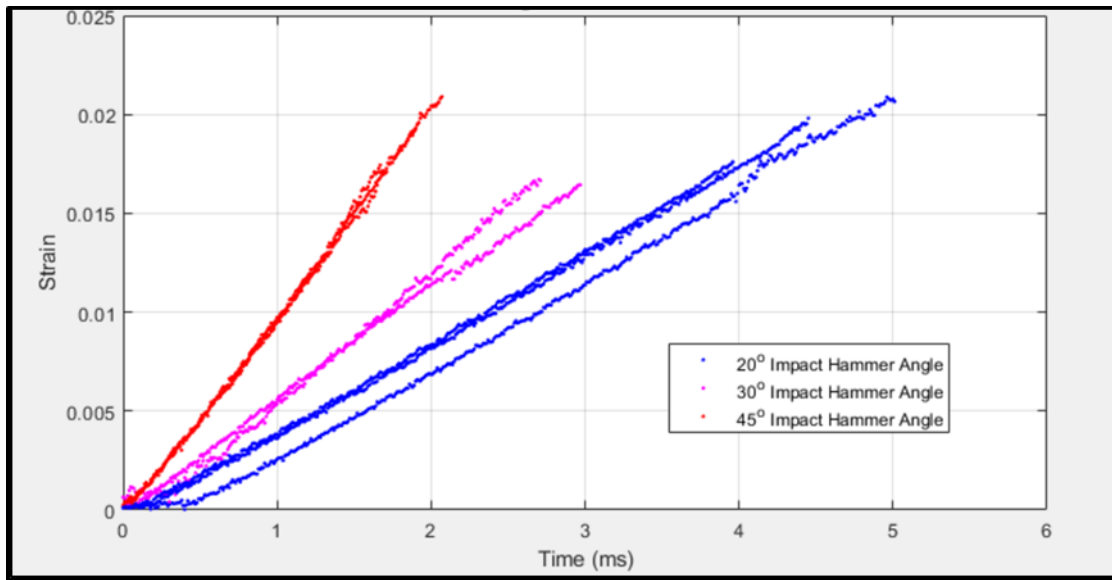


Figure 19. Strain versus Time Curves for Specimen Subset QI1 – High Strain Rate

At impact angles of 20° and 45°, subset QI1 produced impact test durations and failure strains that were comparable in magnitude to the bidirectional impact specimens. One of the 30° impact tests was unable to produce meaningful strain readings due to strain gauge delamination, so the corresponding data set was omitted. The quasi-isotropic QI1 high strain rate results produced highly-linear strain versus time curves (when compared to the bidirectional and unidirectional subsets). All high strain rates and failure strains

associated with subset QI1 were calculated in an identical manner to the previous two impact specimen composite layups.

2. Impact Specimen Subset QI2

Quasi-isotropic impact specimen subset, QI2 was subject to impact hammer angles of 20° , 45° , and 75° . The relevant strain versus time responses for each QI2 impact test are displayed in Figure 20.

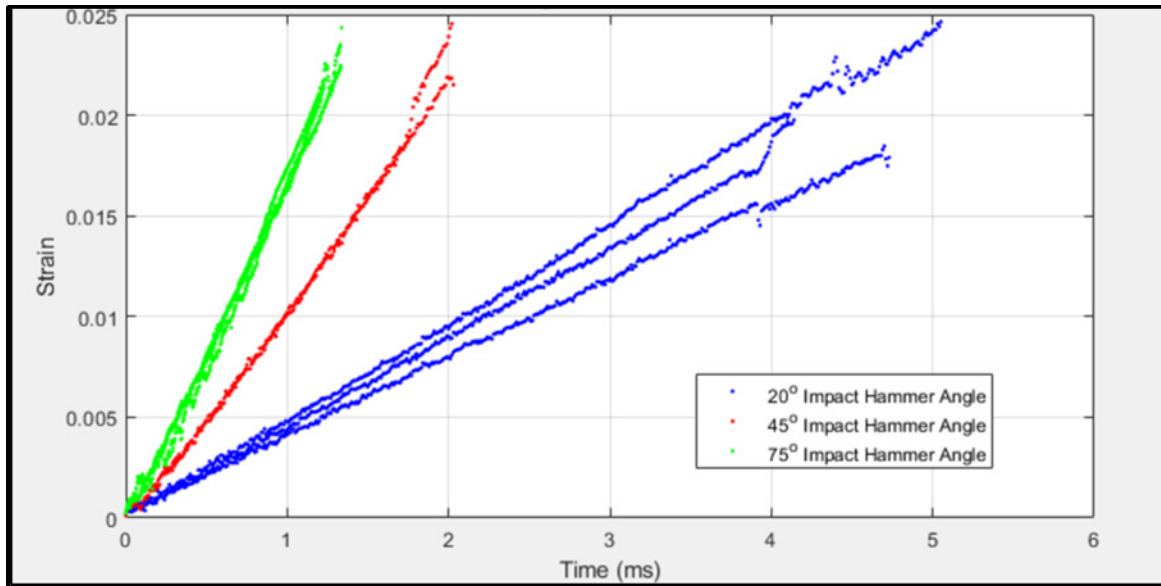


Figure 20. Strain versus Time Curves for Specimen Subset QI2 – High Strain Rate

The measured failure strains for the 20° and 45° QI2 impact were comparable in magnitude to those of subset QI1 at identical impact hammer angles. One QI2 20° impact angle test yielded a relatively high failure strain value compared to the other two impact tests at the same hammer angle. The QI2 impact test durations were also similar in magnitude to subset QI1. Additionally, quasi-isotropic subset QI2 produced highly linear strain versus time curves, thus enabling simple calculation of the associated strain rates.

3. Interpretation of High Strain Rate Results – Subsets QI1 and QI2

The impact test results of the quasi-isotropic impact specimen subsets were relatively consistent. Figure 21 displays the relationship between applied strain rate and failure strain (averaged at each impact hammer angle) for subsets QI1 and QI2.

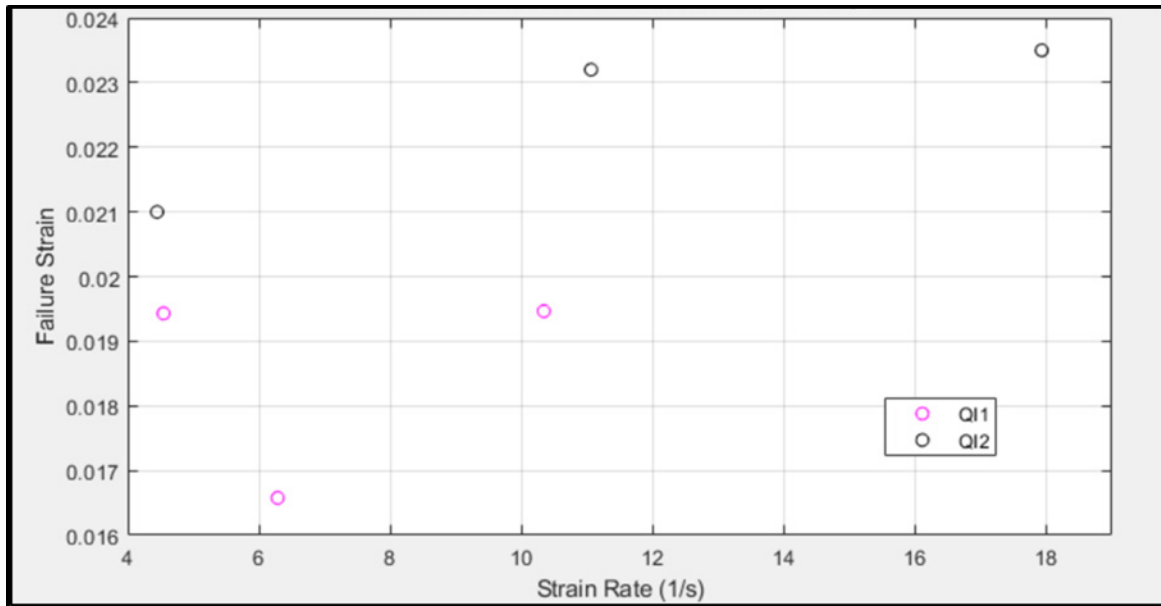


Figure 21. Failure Strain as a Function of Supplied High Strain Rate – QI1 and QI2

Subset QI1 produced slightly lower average failure strains at impact hammer angles of 20° and 45° when compared to subset QI2. The relevant average strain rates were consistent between the two orthogonal quasi-isotropic subsets. As in the unidirectional and bidirectional impact test results, all high strain rates and associated failure strains were averaged to produce a single data point for each quasi-isotropic subset.

D. SUMMARY OF HIGH STRAIN RATE (IMPACT) RESULTS

Analysis of the high strain rate data for each specimen type and orthogonal orientation revealed useful results that were then compared to low strain rate data for model finite element model development and implementation. Table 8 lists the mean high strain

rates and failure strains; each strain rate and failure strain value represents the average of all high strain rate data sets for each composite layup type and machined orientation.

Table 8. Mean High Strain Rate and Associated Failure Strain Values

	U1	U2	B1	B2	QI1	QI2
Mean Strain Rate (45° Impact Hammer) (s^{-1})	9.964 s^{-1}	10.007 s^{-1}	10.357 s^{-1}	11.653 s^{-1}	10.343 s^{-1}	11.157 s^{-1}
Mean Failure Strain	0.0119	0.0106	0.0178	0.0195	0.0187	0.0225
Failure Strain Standard Deviation	8.9223e-04	9.4906e-04	0.0018	0.0014	0.0019	0.0023
Strain Rate Standard Deviation (45° Hammer Angle)	0.5137 s^{-1}	0.8533 s^{-1}	0.0114 s^{-1}	0.2916 s^{-1}	0.1179 s^{-1}	0.0209 s^{-1}

The mean strain rates produced by each impact specimen type were relatively consistent. Quasi-isotropic subset, QI1 produced a lower mean strain rate due to the lack of 75° impact test results. Both unidirectional specimen subsets produced significantly lower failure strain values when compared to the bidirectional and quasi-isotropic subsets. Additionally, strain rate-dependence in every impact specimen subset was minimal when comparing high strain rates at impact hammer angles ranging from 20° to 75°. This lead to the conclusion that strain rate dependence may only be easily observed over a significant range in strain rate order of magnitude—as the proceeding quasi-static results shall support. The mean strain rate and failure strain values in Table 8 were used in the development of a logarithmic relationship between applied strain rate and predicted failure strain (described in Chapter V). This relationship was then supplied to the aforementioned ABAQUS finite element model for numerical validation.

THIS PAGE INTENTIONALLY LEFT BLANK

IV. QUASI-STATIC TEST RESULTS

All quasi-static (quasi-static/tensile) tests were carried out in accordance with the parameters and experimentation equipment described in Chapter II, Section C.2. All strain data obtained during tensile testing were processed, plotted, and analyzed using MATLAB software. Appendix B provides an example MATLAB script that outlines the data processing procedure for tensile specimen subset, U1. An identical data processing method was used for every tensile specimen subset. The aforementioned tensile test specimens were measured prior to testing. All relevant strain gauges were slightly deflected prior to testing in order to ensure proper strain gauge function. A standard tensile tester extension rate of 2 mm/s was applied to every tensile specimen.

A. UNIDIRECTIONAL SPECIMEN QUASI-STATIC RESULTS

All tensile specimen subsets contained a total of two specimens due to the limited machinability of the carbon fiber plates. All six composite layers in tensile subset U1 contained fibers that were oriented parallel to the applied tensile load. Conversely, the fibers in each composite layer of subset U2 were oriented orthogonal to the applied tensile load. Intermittent failure mechanisms such as single-layer fiber fracture and layer delamination were observed prior to final specimen failure (in some cases); however, all tensile tests were run up to the point of *complete* specimen fracture. Strain data were recorded at an approximately 2 kHz data rate. The first tensile specimen from subset U1 was tested at a strain data rate of 25 kHz. This high DAQ rate produced an oversized data set that was not able to be processed; therefore, only one strain-time curve from subset U1 is displayed in the following figure. Failure strain data from the oversized U1 data set were still obtained for use in calculating the mean tensile failure strain. Figure 22 displays the strain with respect to time for tensile subsets U1 and U2.

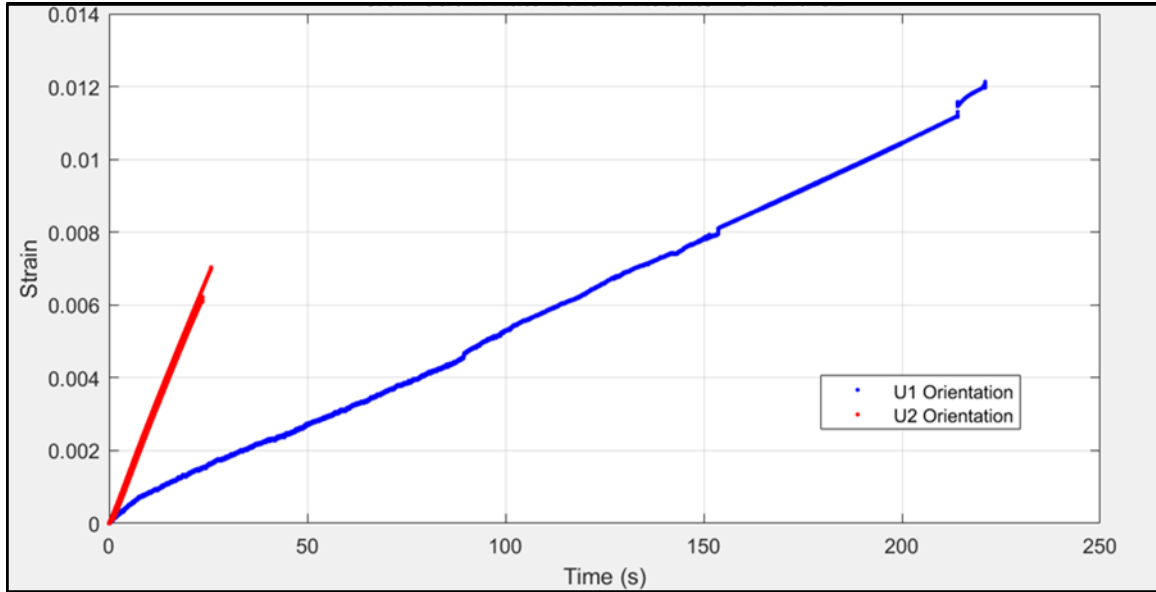


Figure 22. Strain versus Time Curves for Specimen Subsets U1 and U2 – Quasi-Static

Figure 22 displays a notable difference in the quasi-static behavior of the two orthogonal unidirectional subsets. High strain rate tests produced mean failure strains of 0.0119 and 0.0106 for subsets U1 and U2 (respectively). Quasi-static testing produced a significant reduction in mean failure strain in the matrix-dominated unidirectional specimens, and a negligible change in mean failure strain in the fiber-dominated specimens ($\sim 1.681\%$). The mean failure strain of the quasi-static results of subset U2 evaluated to 0.0066; this indicated a 37.74 % reduction in failure strain when compared to the high strain rate results! The reduction in failure strain of the matrix-dominated unidirectional specimens was an expected result. Daniel, Werner, and Fenner [11], applied Hopkinson pressure bar tests to AS4/3501-6 carbon-epoxy composite yielded reductions in ultimate tensile strength (when comparing strain rates of 10^{-4} s^{-1} and 380 s^{-1}) from 400 MPa to 250 MPa. Although a different failure characteristic (ultimate tensile strength), the strengthening effect of an applied high strain rate observed in Daniel, Werner, and Fenner is consistent with the failure strain results observed in this research.

In addition to measured strain with respect to time data, load-displacement data from the tensile tester apparatus were collected for each test. In order to convert applied

load values to axial engineering stress, the measured load data were divided by the average cross-sectional area of the tensile specimens. Because the tensile tester does not produce highly accurate displacement measurements, converting displacement data to engineering strain was not as straightforward. To accomplish this conversion, displacement data were divided by the original length of each specimen to achieve a ‘baseline’ strain measurement. These converted strain data were then plotted with the gauge-measured strain data. Then, a scalar multiplier was applied to the converted strain data in order to closely-match the gauge-measured data. This displacement-to-strain conversion process allowed for accurate production of stress-strain curves for each tensile specimen (this conversion method was applied to the bidirectional and quasi-isotropic tensile specimens as well). The engineering stress-strain curves for tensile subsets U1 and U2 are shown in Figure 23.

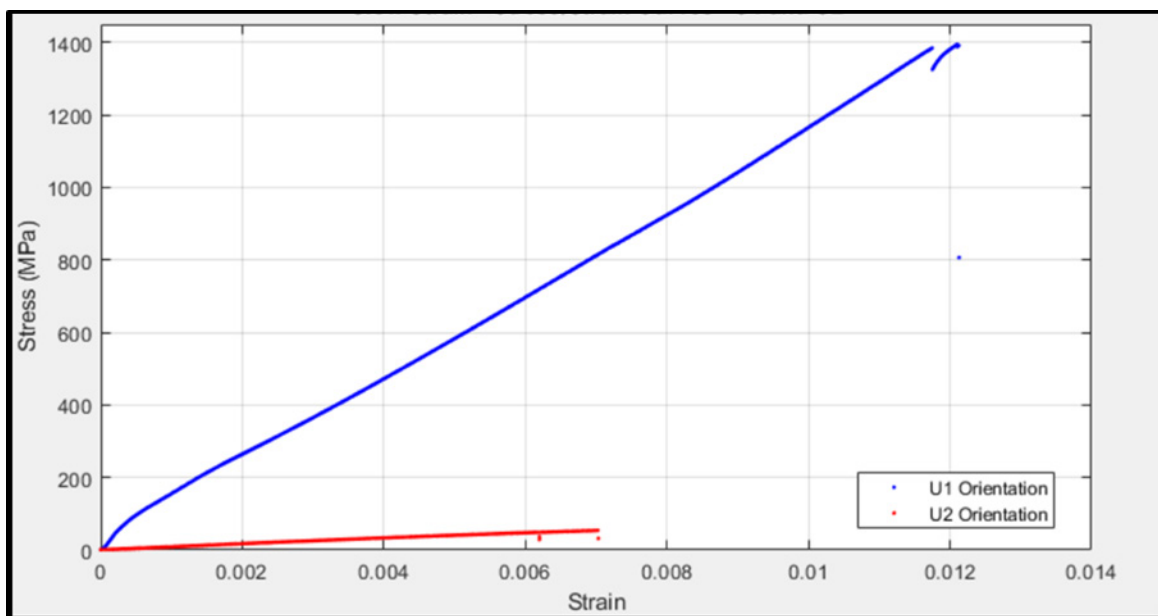


Figure 23. Engineering Stress-Strain Curves – Specimen Subsets U1 and U2

In Figure 23, the drastic difference in strength between the matrix and fiber material is easily observed. The mean UTS of subset U1 was approximately 1350.5 MPa, while the mean UTS of subset U2 evaluated to only 51.458 MPa. Despite the observed intermittent failure mechanisms during quasi-static testing, the engineering stress-strain curves for subsets U1 and U2 appear consistently linear throughout each test duration.

B. BIDIRECTIONAL SPECIMEN QUASI-STATIC RESULTS

Tensile subsets B1 and B2 each contained two specimens for quasi-static testing. All tensile testing applied to the bidirectional subsets was subject to a 2 mm/s extension rate. Due to the orientation of the outer composite layers on tensile subset B2, matrix fracture—orthogonal to the applied tensile load—occurred on said outer layers. This proved problematic for the strain gauges mounted to the tensile subset B2 specimens because the intermittent matrix failure also caused the strain gauges to fail prior to final specimen failure. An image of this intermittent failure mechanism in tensile subset B2 is shown in Figure 24.

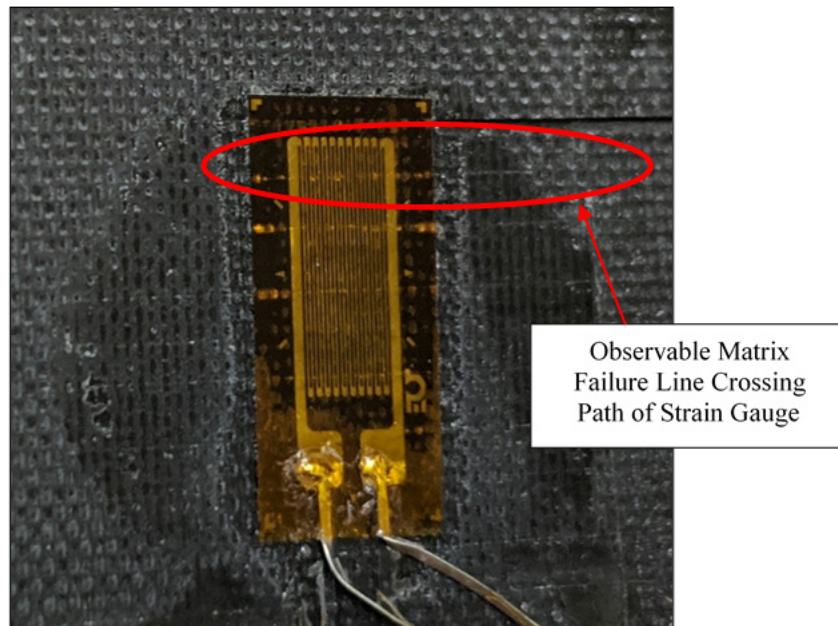


Figure 24. Image of Intermittent Failure of Outer, Matrix-Dominated Layer in Tensile Subset B2

To compensate for the loss of subset B2 strain-time data, the mean slow rate failure strain in this case was calculated from the converted tensile tester data. Figure 25 displays the time-dependent strain data for tensile specimen subsets B1 and B2.

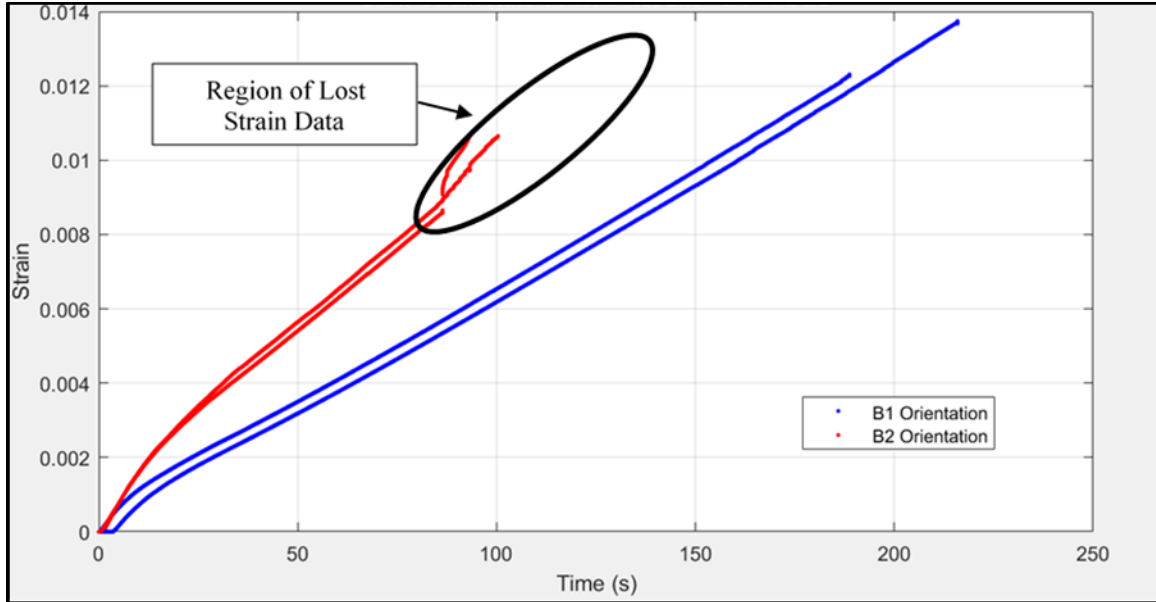


Figure 25. Strain versus Time Curves for Specimen Subsets B1 and B2 – Quasi-Static

Despite the constant extension rate of 2 mm/s, the particular composite layup associated with subset B2 produced a slightly higher tensile test strain rate compared to its B1 counterpart. Aside from the region of lost data in subset B2, all of the strain response curves in Figure 25 appear to be highly linear. Additionally, the mean failure strain of tensile subset B1 was approximately 0.0130. The mean failure strain of tensile subset B1 indicates a 26.97 % reduction in failure strain when compared to the corresponding high strain rate results. An identical conversion from load and displacement to engineering stress and strain was applied to tensile subset B1 and B2 data produced by the tensile tester. The corresponding stress and strain plots are shown in Figure 26.

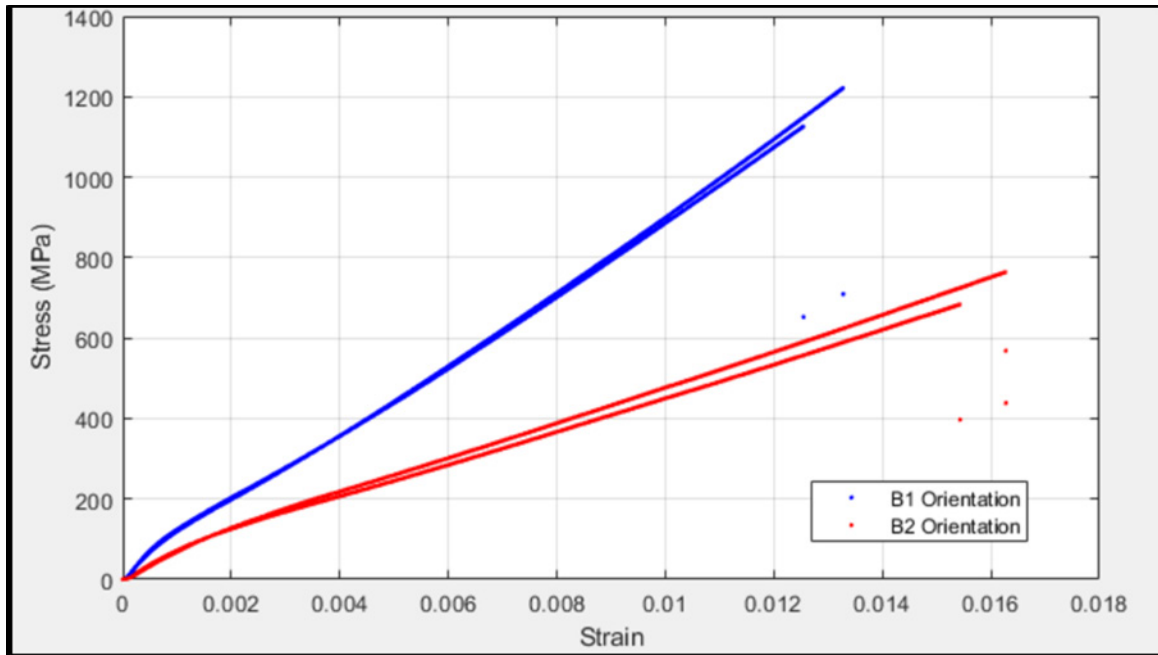


Figure 26. Engineering Stress-Strain Curves – Specimen Subsets B1 and B2

The engineering stress-strain plots in Figure 26 produced a reasonable estimate of the tensile subset B2 failure strain after following the appropriate displacement-to-strain conversion procedure outlined in Chapter IV, Section A. The slow rate failure strain data of tensile subset B2 produced a mean failure strain of approximately 0.0159. This evaluates to an approximate reduction in failure strain of 18.46 %. Due to the greater number of composite layers dominated by the matrix material in subset B2, a greater reduction in failure strain (when compared to subset B1) was expected. The relatively small reduction in failure strain in subset B2 may have been attributed to the small inaccuracy in displacement measurements by the tensile tester. More accurate subset B2 failure strain data could have been obtained by using samples with a 0° outer composite layer orientation (as opposed to 90°).

C. QUASI-ISOTROPIC SPECIMEN QUASI-STATIC RESULTS

Tensile subsets QI1 and QI2 each contained two specimens for quasi-static testing. All tensile testing applied to the quasi-isotropic subsets was subject to a 2 mm/s extension rate. The orthogonal quasi-isotropic subsets contained composite layers that were layered

in such a way that the fiber orientation angle of the specimens were unchanged with respect to machined orientation. The only difference in composite layup orientation between subsets QI1 and QI2 was the fiber orientation assigned on an individual layer basis; however, this orientation assignment order should not have produced any effect on the results. Due to the aforementioned machined orientation independence, the tensile results of subsets QI1 and QI2 were expected to be reasonably similar. Figure 27 displays the time-dependent strain data gathered during quasi-static testing of subsets QI1 and QI2.

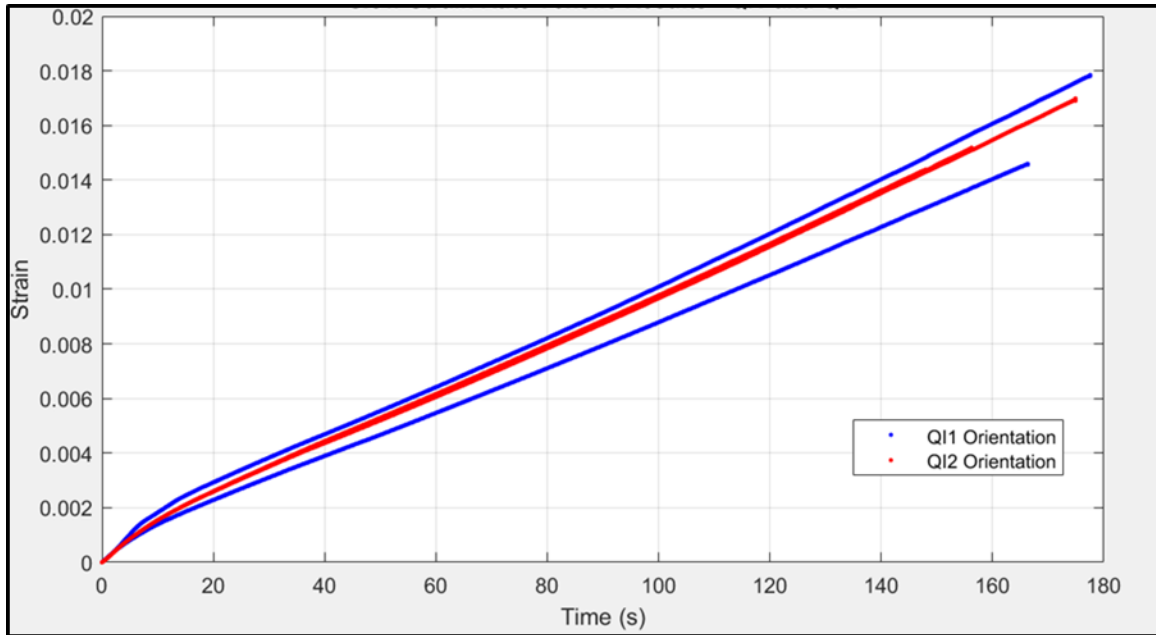


Figure 27. Strain versus Time Curves for Specimen Subsets QI1 and QI2 – Quasi-Static

As expected, the measured strain values of tensile subsets QI1 and QI2 were similar in magnitude; strain rates also appeared to be concurrent between the two tensile subsets. The mean slow rate failure strains of subsets QI1 and QI2 evaluated to 0.0162 and 0.0161 (respectively). These slow rate failure strain values yielded a 13.37 % reduction in subset QI1 failure strain and a 28.44 % reduction in subset QI2 failure strain when compared to high strain rate tests. This difference in strain rate dependence may have been a result of the lack of 75° impact hammer data for subset QI1 that could have increased the mean high

rate failure strain of the subset. A conversion from load and displacement to engineering stress and strain (as described in the analysis of unidirectional tensile data) was applied to tensile subset QI1 and QI2 data produced by the tensile tester. The corresponding stress and strain plots are shown in Figure 28.

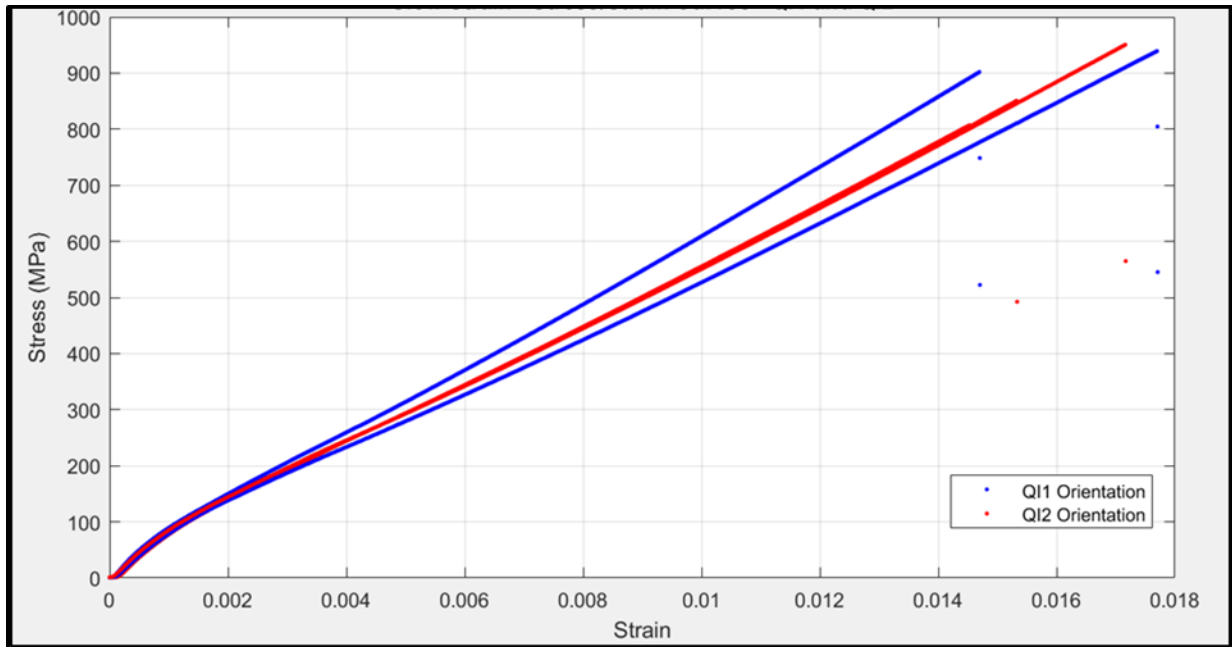


Figure 28. Engineering Stress-Strain Curves – Specimen Subsets QI1 and QI2

The stress-strain curves in Figure 28 indicate a high level of agreement between the QI1 and QI2 stress-strain data sets. The mean ultimate tensile strength produced by tensile subset QI1 was 920.53 MPa, while the mean UTS of tensile subset QI2 was relatively similar in magnitude at 900.63 MPa. This level of UTS agreement was an expected result inherent to the quasi-isotropic composite layup orientation.

D. SUMMARY OF QUASI-STATIC (TENSILE) RESULTS

The low strain rate results of all specimen subsets produced results that expressed either strain-invariant failure strains or failure strain reductions when compared to the corresponding high strain rate results. Tensile subset U1 produced a failure strain reduction of only 1.681 % when comparing high to low strain rate data. This low failure strain reduction

indicated that the carbon fibers do not exhibit significant rate-dependent failure strain properties. Conversely, the primarily matrix-dominated tensile subset U2 witnessed a 37.74 % reduction in failure strain (again, compared to high strain rate results). This relatively high failure strain reduction indicates a strong rate dependence in matrix material failure strain. The remaining subsets also produced rate-dependent failure strains, but to a lesser degree due to the presence of some fiber-dominated 0° composite layers. A table identifying the relevant stress and strain failure properties of each tensile subset is provided in Table 9.

Table 9. Mean Low Strain Rate and Associated Failure Stress and Strain Values

	U1	U2	B1	B2	QI1	QI2
Mean Strain Rate (s⁻¹)	5.124e-5 s ⁻¹	2.7537e-4 s ⁻¹	6.200e-5 s ⁻¹	8.581e-5 s ⁻¹	8.886e-5 s ⁻¹	9.177e-5 s ⁻¹
Mean Failure Strain	0.0117	0.0066	0.0130	0.0159	0.0162	0.0161
Failure Strain Reduction (%)	1.681 %	37.74 %	26.97 %	18.46 %	13.37 %	28.44 %
Mean UTS (MPa)	1351 MPa	51.46 MPa	1174 MPa	723.4 MPa	920.5 MPa	900.6 MPa

The quasi-static results displayed in Table 9 were used to produce a logarithmic relationship between the supplied strain rate and the resulting failure strain of each specimen subset. The failure strain reduction associated with each subset was a necessary metric in order to validate predicted outcomes. The fact that tensile subset U1 produced rate-independent failure strain behavior, and subset U2 produced high rate dependence is a good indication of the validity of the low and high strain rate tests. Additional validation is provided by the intermediate failure strain reduction magnitudes adopted by subsets B1, B2, QI1, and QI2. The following chapter describes the integration of high strain rate and quasi-static results into a logarithmic model capable of finite element application.

THIS PAGE INTENTIONALLY LEFT BLANK

V. STRAIN RATE CORRECTION FUNCTION IMPLEMENTATION

The aforementioned failure strain and strain rate data for all specimen subsets subject to tensile and impact testing were gathered in an effort to relate the failure strain behavior of the carbon fiber material as a function of the applied strain rate. The development of this relationship was deemed critical to the accurate and sufficient modeling of fiber-matrix composite materials in an implicit finite element model. Although the failure strain of the carbon fibers proved to be strain rate invariant, the failure strain of the matrix material was found to be highly strain rate dependent. Because of this matrix material strain rate dependence, quasi-static material failure properties were insufficient for FE material damage model implementation.

A. STRAIN RATE CORRECTION FUNCTION DERIVATION

The strain rate correction function used in this thesis was derived from functions provided in [2]. The function models the predicted failure strain of a material as a logarithmic function of the ratio of high and low applied strain rates. A constant, α , is also required as this constant serves as the connection between the general logarithmic relationship and the specific material in question. The general strain rate correction function is shown in Equation (2) [2].

$$\frac{\varepsilon_f^*}{\varepsilon_f} = \left(\alpha \log_{10} \left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_o} \right) + 1 \right) \quad (2)$$

In Equation (2) the predicted high rate failure strain is calculated by multiplying the slow rate failure strain by the aforementioned logarithmic relationship of the high and low strain rate ratio, the material-specific constant (α), and added unity. This model allowed for an accurate prediction of failure strain in multiple composite layup orientations based on the supplied high and low strain rate test data.

B. CALCULATION OF CORRECTION CONSTANTS

Due to the nature of the experimentally-gathered high and low strain rate test data, all terms in Equation (2)—except the correction constants, α —were known. Each impact and tensile specimen subset produced relevant mean strain rate and failure strain data that were applied to Equation (2). Rearranging Equation (2) and applying the known high and low failure strain and rate values, the correction constants for each composite specimen subset were calculated. These α values are displayed in Table 10.

Table 10. Strain Rate Correction Constants for Each Composite Subset

	U1	U2	B1	B2	QI1	QI2
Calculated α Values	0.0032	0.1330	0.0701	0.0438	0.0315	0.0782

Table 10 displays a nearly two order of magnitude difference between the subset U1 and U2 correction constants. This was caused by the rate-invariance of subset U1 failure strain (due to fibers adopting most of the applied load) and the high rate dependence of subset U2 failure strain. Subsets B1, B2, QI1, and QI2 expressed overall lower correction constants when compared to subset U2, but said constants proved an intermediate rate dependence compared to subset U1. The above correction constants were applied to Equation (2), and then the correction functions for each subset were run in MATLAB over a range of multiple strain rate orders of magnitude. The predicted failure strain as a function of applied strain rate were plotted for each specimen subset; the resulting correction function plots are provided in Figures 29, 30, and 31.

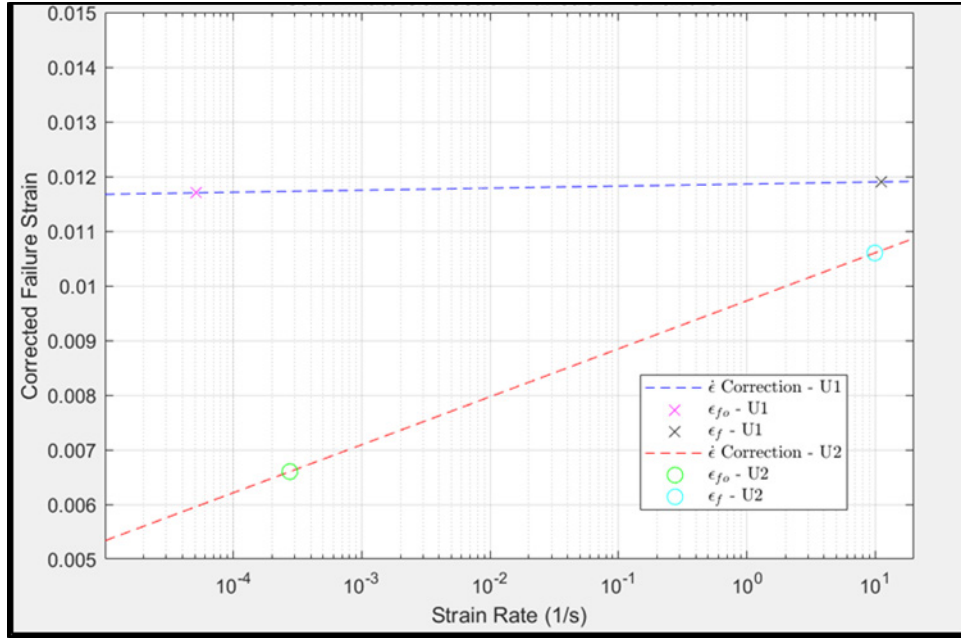


Figure 29. Strain Rate Correction Function Visualization– Subsets U1 and U2

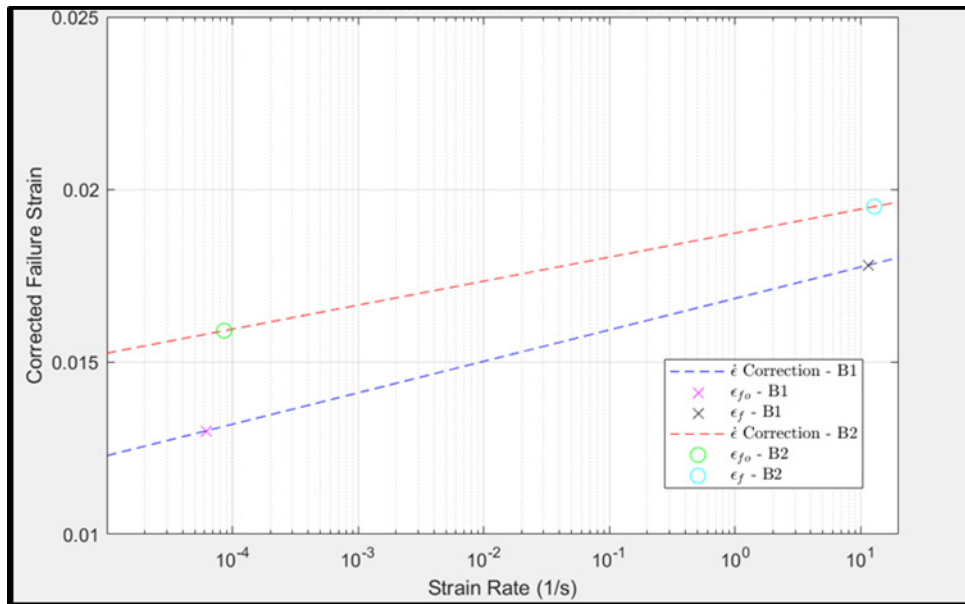


Figure 30. Strain Rate Correction Function Visualization– Subsets B1 and B2

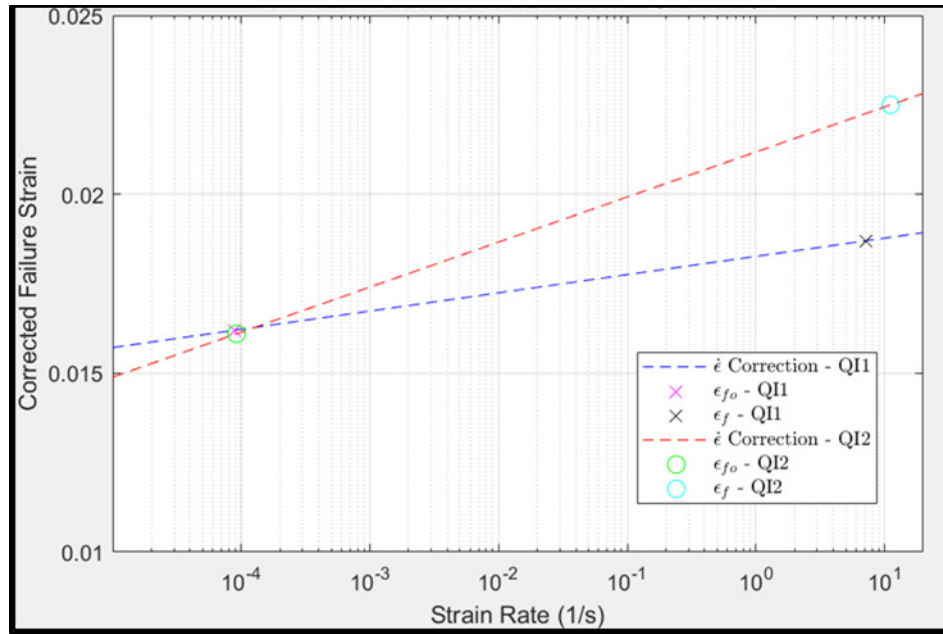


Figure 31. Strain Rate Correction Function Visualization– Subsets QI1 and QI2

In Figure 29, the significant difference in failure strain rate dependence between specimen subsets U1 and U2 is clearly displayed by each subsets' respective correction function. The bidirectional and quasi-isotropic correction functions did not display as great of a disparity between their respective orthogonal subsets. The strain rate correction function data provided above was used as a method of calculating the high strain rate failure properties of the carbon fiber material (in multiple composite layup orientations). These composite failure properties were then passed into the finite element model used to predict fiber-matrix composite damage under both low (quasi-static) and high (dynamic) strain rate loading conditions. However, prior to creating the finite element model, the input load conditions relevant to each impact specimen subset were required. The following chapter describes the process of obtaining the necessary impact load data that were passed into the FE model.

C. STRAIN RATE CORRECTION CRITERIA FOR FINITE ELEMENT MODELING

Upon defining the relationship between high and low strain rate failure strains for each specimen subset, a determination of the specific criteria to be implemented in FE modeling was made. Although useful for future strain rate effect analyses, the correction function constants and the associated plots for the bidirectional and quasi-isotropic subsets were not applied to the finite element model. Instead, the strain rate correction constants associated with subset U1 and U2 data were taken to be the pure fiber and matrix correction constants. This assumption was made due to the minimal contribution of fiber-matrix interface strength to the overall failure properties of the carbon fiber material (as dictated by composite failure theory).

The finite element model created for this thesis relied on defined composite layers, associated fiber orientations, and a 15-property material property array (further explained in Chapter VII). The primary strain-rate-effected material properties in the FE material property array were fiber and matrix failure strain. To account for strain rate effects in FE modeling, the mean quasi-static failure strains of the fiber and matrix (gathered from tensile testing) were multiplied by a specific factor determined from Equation (2) and the associated correction constants (Table 10). The resulting failure strain multiplication factors were applied to the quasi-static failure strain, and then passed into the FE model's material property array. A table of the strain rate multiplication factors (and failure strain values) for the fiber and matrix is provided in Table 11.

Table 11. Failure Strain Multiplication Factors for FE Modeling

	ϵ_f , low strain rate	Multiplication Factor	ϵ_f , high strain rate
Fiber	0.0117	1.0171	0.0119
Matrix	0.0066	1.6061	0.0106

THIS PAGE INTENTIONALLY LEFT BLANK

VI. IMPACT LOAD DATA FOR FINITE ELEMENT MODELING

Due to a lack of mass and rotational inertia data for the Charpy impact tester used during high strain rate testing, the mean impact force and impact duration for each specimen subset needed to be determined experimentally. These impact force (as a function of time) data were then supplied to each specimen finite element model as the relevant input column vector. All input forces were measured using a Honeywell Model 13 Subminiature Compression load cell. For further information regarding the impact load measurement apparatus and procedure, refer to Chapter II, Section D.

A. IMPACT LOAD MEASUREMENT CHALLENGES

Impact specimen subset B1 was the first subset subject to impact load measurements. In addition to impact load readings, strain gauge readings were also taken on each B1 impact load specimen in order to ensure similar strain rates and failure strains when compared to the high strain rate tests (without the applied load cell). Figure 32 shows the strain readings taken during load-impact testing at a 45° impact hammer angle compared to the raw strain data from high strain rate testing.

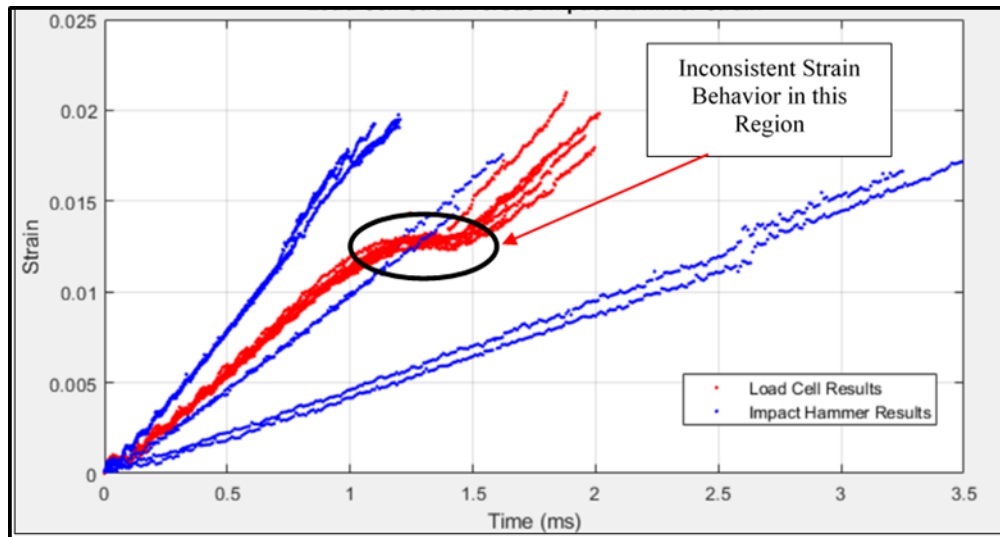


Figure 32. Impact Subset B1 Strain Readings – Load Cell and Impact Hammer Contact Surfaces

As shown in Figure 32, the impact strain behavior produced by the load cell generated similar strain rates and overall failure strains when compared to those of the pure impact hammer tests (at a hammer angle of 45°). However, at an impact time of approximately 1.00 ms, the load cell impact tests produced a region of strain invariance for approximately 0.50 ms prior to final specimen failure. The cause of this temporary strain invariance was unknown until physical inspection of the subset B1 load-impact specimens. Each specimen exhibited markings on the outer rim of the impact site that appeared to be caused by the casing of the load cell making contact with each specimen as bending deflection (and strain) increased. Figure 33 provides a visualization of this occurrence.

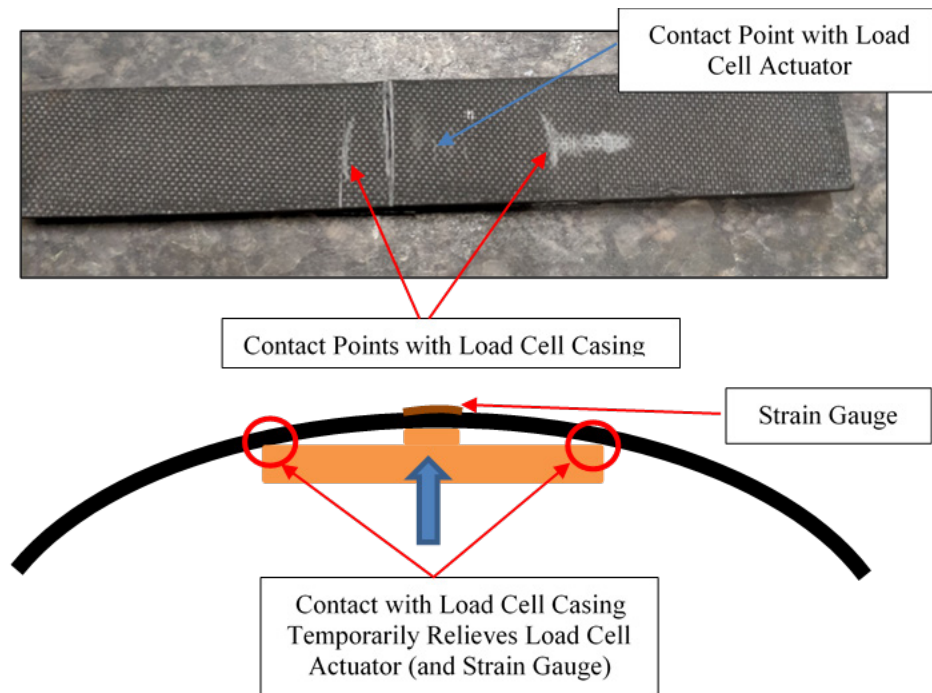


Figure 33. Image (top) and Graphic (bottom) Displaying Load Cell Casing Contact

As shown in Figure 33, the sides of the load cell casing (that do not contribute to load measurements) may have partially relieved the strain gauge and load cell actuator during the aforementioned 0.50 ms interval displayed in Figure 32. Load cell size limitations denied the implementation of a larger load cell actuator tip that could have

prevented this issue. To compensate for the strain/load relief phenomenon, the curve fits applied to all load data were modified to disregard the 0.50 ms strain/load relief interval in order to more accurately mimic the pure impact test strain behaviors. This curve-fitting and modification process are explained in the following section.

B. LOAD DATA CURVE FITTING AND MODIFICATIONS

All load-impact data (for each specimen subset) were plotted against the recorded impact duration in each case. All load-impact specimen subsets contained a total of six specimens, thus producing six load versus time curves to fit to a general subset-specific curve for FE implementation. The curve fits were made using a seventh-order curve fit approximation. To compensate for the aforementioned strain/load relief phenomenon, each curve fit was transformed into a first-order approximation halfway through each impact duration. The curve fit load data were then cutoff at the approximate time of failure as indicated by the pure impact hammer strain data (see Chapter III). After performing FE analysis on the U2 impact subset model, the quasi-linearized input force method produced axial strain calculations that were nearly 300 % greater than the experimental values. Instead of using a quasi-linear input force for the FE model of impact subset U2, the seventh-order curve fit was used continuously. The subset U2 load-time curve fit was cutoff at the associated mean impact duration derived from the high strain rate test data (at a hammer angle of 45°). The first-order portion of each impact load curve fit was extended well beyond the failure envelope of the material. This was done in order to ensure the FE model would produce material failure. Any FE data past the modeled failure point were discarded. The curve fitting processes for load-impact subsets U1 and U2 are shown in Figure 34.

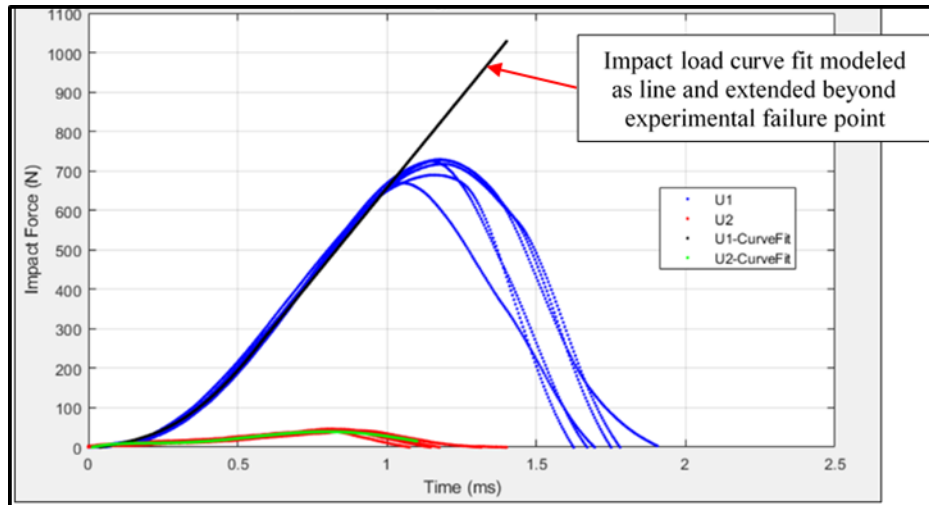


Figure 34. Time-Dependent Impact Force Data and Curve Fits – Subsets U1 and U2

The linearized curve fit regions are clearly displayed in Figure 34. As will be proven in the description of the finite element results, this linearization process of the input force column vector data allowed for more accurate impact modeling. The impact force data and respective curve fits for the bidirectional and quasi-isotropic subsets are shown Figures 35 and 36.

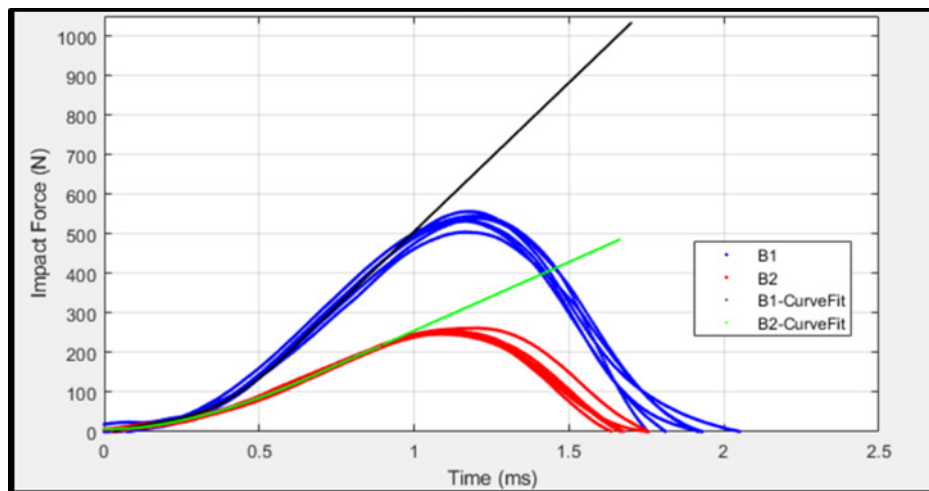


Figure 35. Time-Dependent Impact Force Data and Curve Fits – Subsets B1 and B2

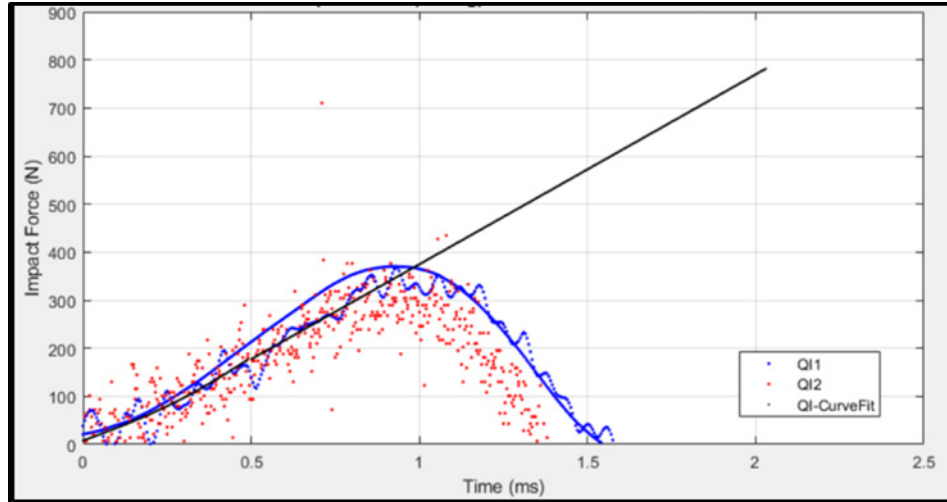


Figure 36. Time-Dependent Impact Force Data and Curve Fits – Subsets QI1 and QI2

Linearization of the unidirectional force-time curve fits produced less drastic differences between the fit approximations and the measured load data when compared to the bidirectional and quasi-isotropic curve fits. The first set of quasi-isotropic impact load measurements were subject to the incorrect DAQ rate and thus produced the highly variable load readings shown in Figure 36. Additionally, only one load-time curve fit was generated for both quasi-isotropic subsets due to similar load-time behaviors. The quasi-isotropic load data produced impact duration times that were nearly 0.50 ms less than those measured in the pure impact tests. This may have been a result of complete relief of the load cell actuator during load-impact testing, thus producing the short measured impact durations and measured peak loads. The mean quasi-static UTS of the quasi-isotropic subsets was approximately 300 MPa less than the observed subset U1 UTS (1351 MPa). However, the subset U1 load-impact specimens produced peak-measured loads of nearly 850 N, while identical quasi-isotropic load-impact specimens produced measured peak loads slightly less than 400 N. This disparity does not agree with the overall material strength data, and the same statement could be applied to the bidirectional specimens. Therefore, the mean impact duration (at a hammer angle of 45°) was calculated for each specimen subset from the pure impact data (without the applied load cell). These impact duration times were then applied to the linear portion of each force-time curve fit, thus extending the peak force

values to more reasonable magnitudes that correspond with the measured impact performance of each specimen subset during pure impact testing. Upon producing the necessary time-dependent impact load curve fits for each specimen subset, the load-time data were exported for finite element implementation in the implicit impact model to be described in the proceeding sections.

As displayed in Figures 34-36, the method of measuring impact force for each subset contained sources of error that led to potentially improper force-time curves. As aforementioned, the physical design of the load cell and its actuator caused strain/load relief to occur as each impact specimen deflected enough to contact the edges of the load cell casing. In certain cases—such as the quasi-isotropic load-impact specimens—this casing contact may have completely relieved the load cell actuator, thus cutting off load measurements prior to final specimen failure. Due to the brittle nature of the carbon fiber material, a linear load-time approximation up to final failure is relatively accurate; however, small nonlinear deformation effects cannot be accounted for by a first-order curve fit. The lack of plastic deformation effects in the stress-strain curves of each specimen subset points to a negligible plastic load response under impact testing, but further experimentation beyond the scope of this thesis is necessary in order to completely prove this assumption. The quasi-linear load response curve fit data were implanted in the FE model of each impact specimen without the consideration of significant plastic deformation prior to final failure.

For further proof of the brittle assumption applied to each impact load curve fit, note the lack of significant nonlinear deformation in the stress-strain curves shown in Figure 23, Figure 26, and Figure 28 (Chapter IV).

VII. FINITE ELEMENT MODEL DEVELOPMENT

All strain rate, failure strain, and impact load data were critical to the proper development of a dynamic finite element model capable of reasonably handling fiber-matrix composite material failure (dynamic and quasi-static). The UMAT subroutine described in Ref. [5] was used as the primary numerical method to handle fiber, matrix, and fiber-matrix interface failure. Implementation of the subroutine in ABAQUS finite element software required a specific process that is explained in detail in Appendix C.

A. IMPACT SPECIMEN FINITE ELEMENT MODEL DEVELOPMENT

The impact (high strain rate) finite element model was developed in accordance with material properties, geometries, and boundary conditions that most closely represent the experimental impact setup. An important distinction between tensile (quasi-static) and impact (high strain rate) FE modeling aside from obvious specimen geometry differences; the input material properties in the impact FE model were modified to account for high strain rate effects based on the gathered experimental data. The impact specimen FE simulation was based on a dynamic-implicit model structure with a defined time step of $1\text{e-}06$ s (derived from the time step size of the load cell data). Pre-determined simulation run times were based on the mean test duration of each impact subset, gathered from experimental high strain rate data.

1. Impact Specimen FE Geometry

The established geometry of the impact specimen FE model was based on the mean dimensions of all impact specimens. The impact specimen FE part (used in each subset FE analysis) was partitioned along its thickness direction in accordance with the manufacturer-determined composite layer counts provided in Table 4. This thickness direction partitioning allowed for the assignment of individual layer fiber orientations using the ABAQUS graphical user interface (GUI). The Figure 37 provides an ABAQUS image of the impact specimen part used in all high strain rate FE modeling, and a graphic displaying the applied dimensions.

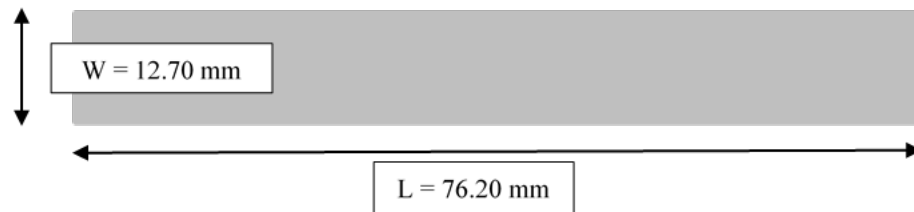
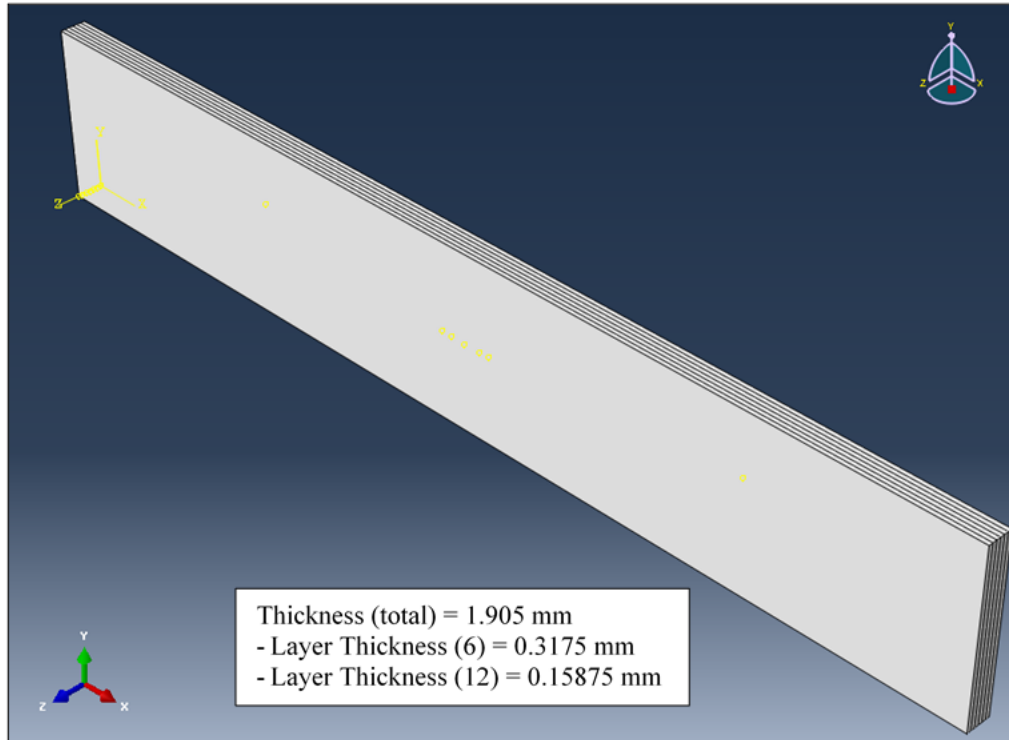


Figure 37. Basic Impact Specimen FE Model Geometry

Figure 38 displays a single partitioned layer (from the impact subset U1 FE model) and its associated fiber orientation. All fiber orientation angle in ABAQUS were defined as rotation angles about the thickness axis (z-axis in ABAQUS) of the impact specimen model.

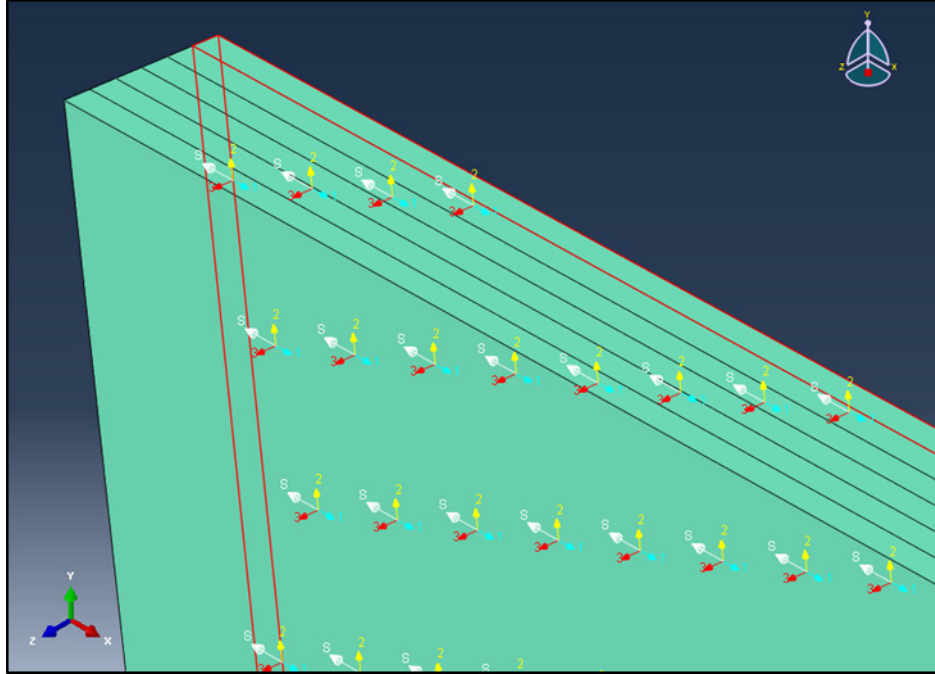


Figure 38. Graphic Displaying Fiber Orientation of Single FE Impact Specimen Composite Layer

The use of composite layer partitioning allowed for simple adjustment of the fiber orientation angles at each partitioned later in order to accommodate different composite layup schemes (U1, U2, B1, etc.). The quasi-isotropic specimens utilized a 12-layer composite layup; therefore, the FE model and its associated partition layers were adjusted accordingly.

2. Impact Specimen Material Property Inputs

The particular material property inputs required for the impact and tensile FE models were dictated by the UMAT subroutine. The UMAT subroutine required 15 material property inputs that were passed from the ABAQUS GUI into the UMAT subroutine at each simulation time step. For further information regarding the material property subroutine process, see [5]. The determination of the required material properties relied on manufacturer specifications, experimental high strain rate effects, and calculations based from a unit cell model. Material properties that were not experimentally available or provided by the manufacturer were estimated using the unit cell model, which

is used for the multiscale analysis of composite materials, such that the predicted composite material properties agree with experimentally determined data. The unit cell model and the associated composite material failure criteria are described in detail in Y. Kwon and J. Darcy [12]. A MATLAB script—created by Dr. Young Kwon, Naval Postgraduate School—utilized unit cell failure criteria and experimental data to predict material properties of the fiber and matrix. To accomplish this, the Young's moduli and shear moduli of the fiber and matrix materials were adjusted within the MATLAB script until the calculated failure strains (produced by the program) matched the experimental data. The experimentally determined fiber and matrix failure strains were derived from the mean failure strains of impact subsets U1 and U2 (respectively).

As indicated in [5] and [12], the unit cell model and associated material property calculations were developed for quasi-static loading conditions. To account for dynamic (impact) loading conditions, some material properties needed to be altered slightly after initial calculation of the required material property array. All calculated material properties were determined based on the experimental fiber and matrix failure strains. Previous experimentation on similar fiber matrix composites indicated a strain rate dependence of matrix elastic and shear moduli. In Daniel et al. [11] the elastic and shear moduli of the matrix material in a unidirectional carbon/epoxy composite saw a 20% to 30% increase in response to a strain rate increase from 0.0001 s^{-1} to 10 s^{-1} . Similarly, the observed transverse and shear strengths increased by the same amount for an identical strain rate increase [11]. The failure strain multiplication factor in this thesis was notably higher (at approximately 1.6061); however, the overall rate-dependent material response agrees with [11]. Due to a lack of experimentally available high strain rate stiffness values, the matrix material elastic modulus used in the FE model was not multiplied by a high strain rate factor. Further experimentation using a different high strain rate testing method to determine true matrix stiffness values is beyond the scope of this research.

Two additional material property inputs were affected by the high strain rate effect: the perpendicular tensile strength of the fiber-matrix interface, and the associated interface shear strength. The perpendicular failure stress calculated by unit cell failure theory was multiplied by the matrix strain rate correction multiplier of 1.6061. The shear strength was

determined by multiplying the high strain rate perpendicular strength by a factor of $2/3^{1/2}$ (~ 0.866). The 15 material property inputs used in every high strain rate FE simulation are provided in Table 12.

Table 12. High Strain Rate FEM Material Input List

Material Property	Value
Longitudinal Fiber Elastic Modulus (Pa)	220e+09 Pa
Transverse Fiber Elastic Modulus (Pa)	28e+09 Pa
Fiber Shear Modulus, G_{12} (Pa)	30e+09 Pa
Fiber Poisson's Ratio, ν_{12}	0.12
Fiber Poisson's Ratio, ν_{23}	0.4
Matrix Elastic Modulus (Pa)	3e+09 Pa
Matrix Shear Modulus (Pa)	1.154e+09 Pa
Matrix Poisson's Ratio	0.3
Volume Fraction of Fiber	0.5
Fiber Tensile Failure Strain	(0.0117)*1.0171
Fiber Compressive Failure Strain	(0.0117)*1.0171
Matrix Tensile Failure Strain	(0.0066)*1.6061
Matrix Compressive Failure Strain	(0.0066)*1.6061
Interface Shear Strength (Pa)	(80e+06)*1.8546 Pa
Interface Perpendicular Strength (Pa)	(54e+06)*1.6061 Pa

In Table 12, the highlighted material property inputs were the relevant properties effected by the applied strain rate. It should be noted that there was no available experimental method to determine high rate compressive failure strain, so the compressive failure strains of the fiber and matrix material were set to their corresponding tensile failure values. The material property inputs show in Table 12 were passed into the impact specimen FE model (for each subset) in order to properly interface with the composite failure subroutine.

It should be noted that considerable assumptions went into the development of the material properties in Table 12. No compressive failure data or stress data during impact testing were available. As a result, elastic moduli, shear moduli, compressive failure strains, and interface strength (perpendicular and shear) had to be assumed from the aforementioned unit cell model. These values were not necessarily best suited for the FE

simulation at hand. Further research, using more advanced data recording capabilities, is required in order to refine the material property inputs to the FE model.

3. Impact Specimen FE Boundary Conditions

FE boundary conditions were established in such a way that closely mimicked actual impact test conditions. In order to properly model, the impact specimens subject to the constraints of the impact tester itself, two rigid blocks were created in the FE model, and a rigid contact condition was defined at the block-specimen interface. The positioning of the blocks in the FE model was based on the measurements made on an impact specimen's position in the Charpy impact tester. In order to remove any unnecessary specimen motion, the impact specimen FE model was constrained along the y-axis (vertical), and only rotation about the vertical y-axis was permitted. An additional boundary condition was placed at the center of the impact specimen, in which only z-axis (direction of impact force) motion was permitted. An ABAQUS image of the impact specimen boundary conditions is shown in Figure 39.

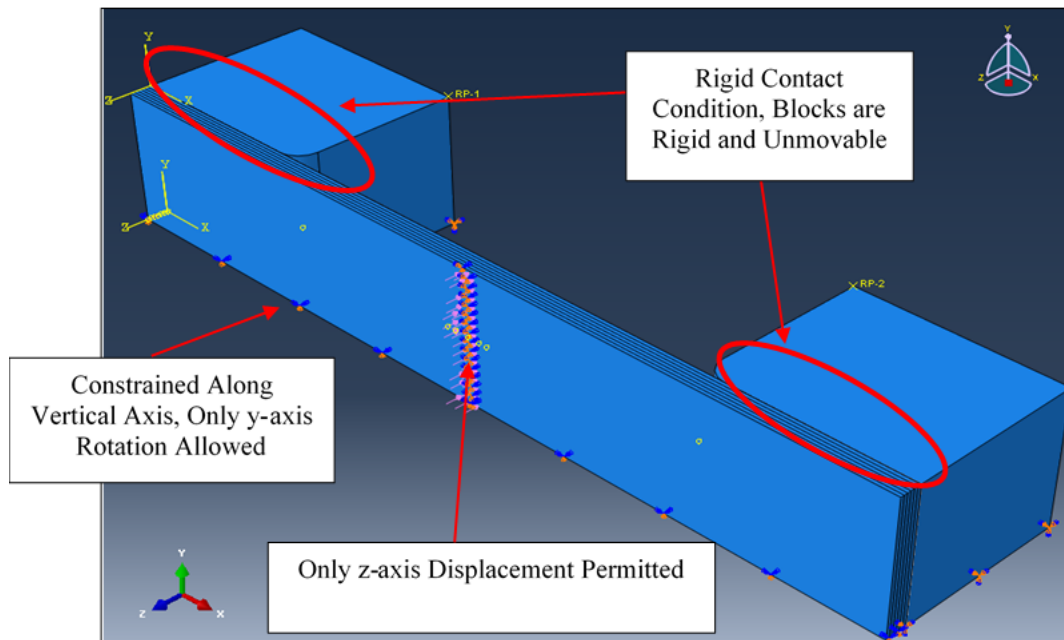


Figure 39. Impact Specimen FE Boundary Conditions

4. Impact Specimen FE Meshing

Both the FE impact specimens and tensile specimens used linear hexahedral elements. This element choice was made as a compromise between computational expense and model fidelity. An initial attempt to use shell elements was made, but interfacing this element type with the UMAT subroutine did not produce a functioning model. Each composite layer partition was assigned a single-element thickness. The long-axis (x-direction) of each composite layer was assigned 171 elements, and the orthogonal y-axis of each layer was assigned 30 elements. This produced a total element count of 30,780 elements with 37,324 nodes. An image of the final impact specimen mesh is provided in Figure 40.

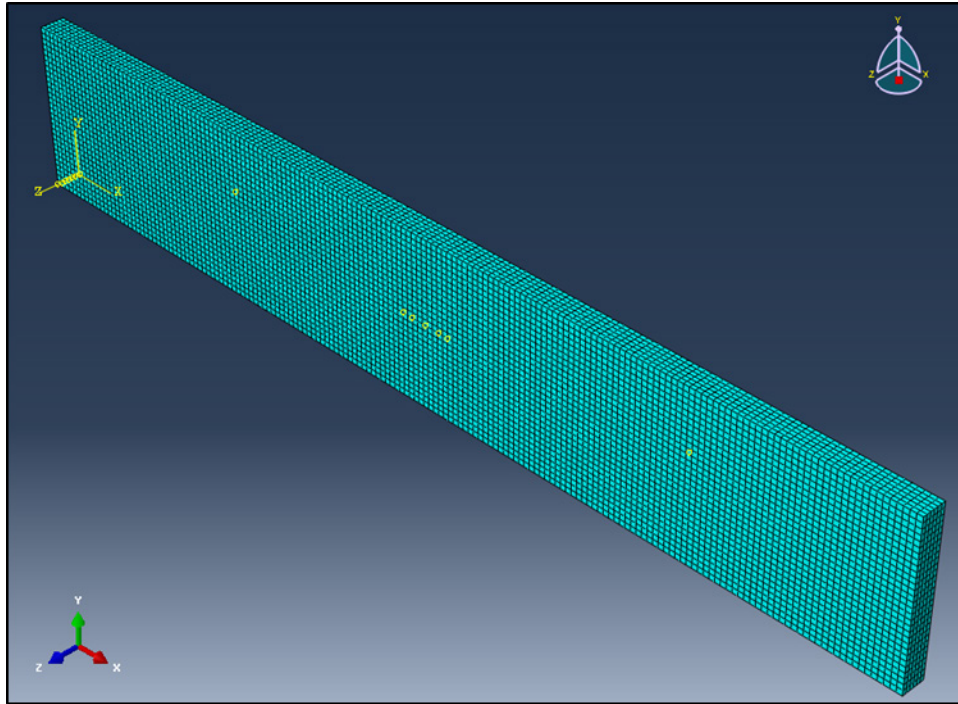


Figure 40. Impact Specimen FE Mesh

Mesh fidelity was verified by decreasing the element count to 27,000 on the impact subset U1 model (150 elements along the long axis). Finite element simulation results at the lower element count produced no notable divergence from the 30,780 element solution. Although computationally expensive, the high element count of 30,780 was maintained in

order to fully-ensure FE model fidelity. The quasi-isotropic impact specimen models required 12 composite layer partitions, so their respective meshes utilized 61,560 elements, which was double the 30,780 element mesh in the bidirectional and unidirectional models.

5. Impact Specimen FE Loading Conditions

The applied load in the impact specimen FE model varied based on the particular subset simulation. ABAQUS allows for the development of tabular, time-dependent load data that are read in at each dynamic implicit time step. To accommodate the experimentally determined impact load values, each impact load curve fit (see Figures 34, 35, and 36) was exported into tabular form. The tabular load data were then copied into the ABAQUS ‘Amplitude’ feature. The amplitude feature dictated the magnitude of the applied load at each 1e-06 s time step, thus fully integrating the experimental load data with the FE model. The applied load was modeled as a distributed force, occupying the measured area of the Charpy impact hammer contact surface. An ABAQUS image of the applied impact loading condition is shown in Figure 41.

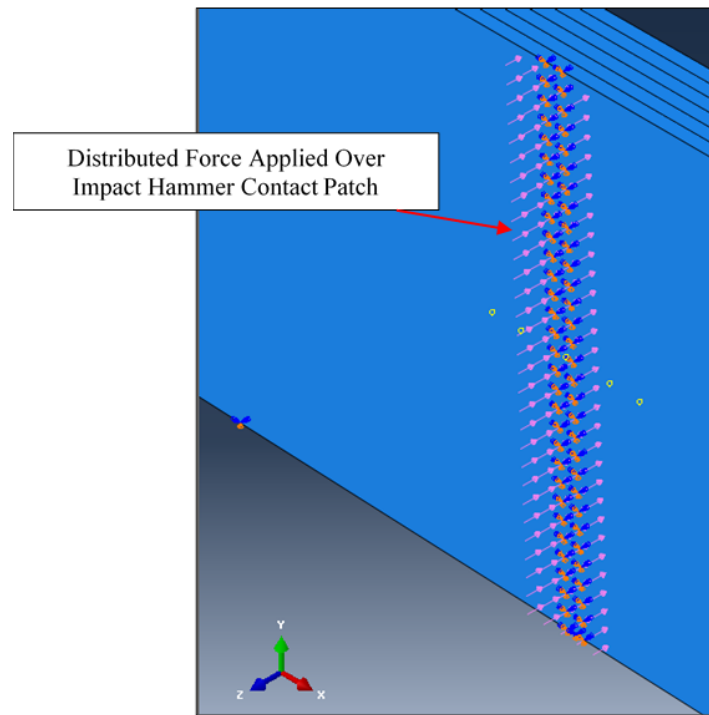


Figure 41. Impact Specimen Applied FE Loading Condition

The impact specimen FE model only produced relevant and viable results when the dynamic-implicit time step was set to 1e-06 s as dictated by the load cell data acquisition rate.

B. QUASI-STATIC SPECIMEN FINITE ELEMENT MODEL DEVELOPMENT

The development of the tensile (quasi-static) FE model utilized identical element types and characteristics of the high strain rate model. The material property inputs to the tensile model were input without the high strain rate effect multipliers. Additionally, the quasi-static FE model utilized a quasi-static format in conjunction with the aforementioned UMAT subroutine.

1. Quasi-Static Specimen FE Geometry

The quasi-static specimen FE geometry was set to a single set of dimensions for consistency amongst the various subsets. This geometry is shown in Figure 42.

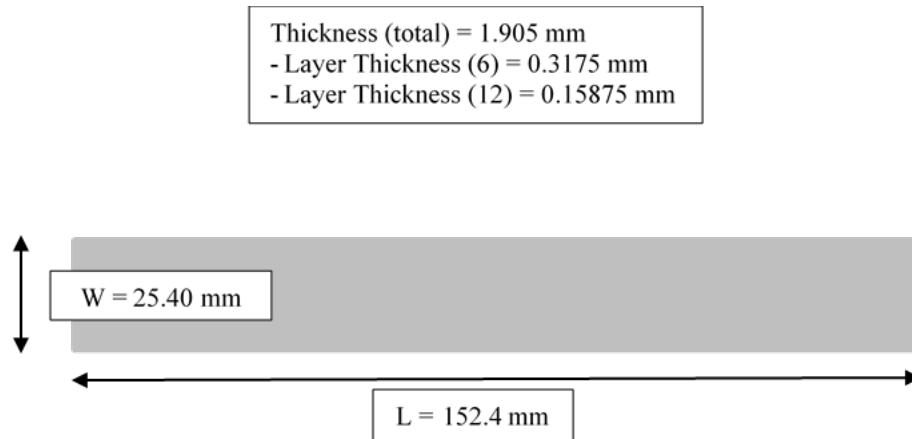


Figure 42. Basic Quasi-Static Specimen FE Model Geometry

The composite layup portioning scheme applied to the impact FE specimens was also applied to the quasi-static model in accordance with each manufacturer-supplied fiber orientation layup (see Figure 38 for further reference).

2. Quasi-Static Specimen Material Property Inputs

The material property inputs for the quasi-static FE model were also based off of the unit cell model provided in Ref. [12]. Generating quasi-static material property inputs was nearly identical to that of the high strain rate model; however, the high strain rate correction factors were suppressed for the quasi-static case. The required 15 material property inputs are shown in Table 13.

Table 13. Quasi-Static FEM Material Input List

Material Property	Value
Longitudinal Fiber Elastic Modulus (Pa)	220e+09 Pa
Transverse Fiber Elastic Modulus (Pa)	28e+09 Pa
Fiber Shear Modulus, G_{12} (Pa)	30e+09 Pa
Fiber Poisson's Ratio, ν_{12}	0.12
Fiber Poisson's Ratio, ν_{23}	0.4
Matrix Elastic Modulus (Pa)	3e+09 Pa
Matrix Shear Modulus (Pa)	1.154e+09 Pa
Matrix Poisson's Ratio	0.3
Volume Fraction of Fiber	0.5
Fiber Tensile Failure Strain	0.0117
Fiber Compressive Failure Strain	0.0117
Matrix Tensile Failure Strain	0.0066
Matrix Compressive Failure Strain	0.0066
Interface Shear Strength (Pa)	80e+06 Pa
Interface Perpendicular Strength (Pa)	54e+06 Pa

3. Quasi-Static Specimen FE Boundary Conditions, Meshing, and Loading

Due to the relative simplicity of the quasi-static FE model, the boundary condition and loading condition development process required less precision and refinement when compared to the high strain rate model. Only one boundary condition on each end of the quasi-static specimen model was required in order to constrain the specimen in a manner that produced concurrent FE results. For this model, the faces perpendicular to the longitudinal axis of the tensile specimen were constrained to allow motion only along the

specimen longitudinal axis. This was deemed the most accurate way of modeling the actual boundary conditions supplied by the tensile tester.

The tensile tester supplied a uniaxial load to each quasi-static specimen; therefore, the FE model was fitted with the same load type. To avoid unrealistic point loading, a distributed tensile load was applied to each end face of the quasi-static FE model. The magnitudes of each supplied FE load were gathered from the relevant tensile tester failure load data (provided for each tensile subset). An ABAQUS image of the supplied quasi-static boundary conditions and loading conditions is shown in Figure 43.

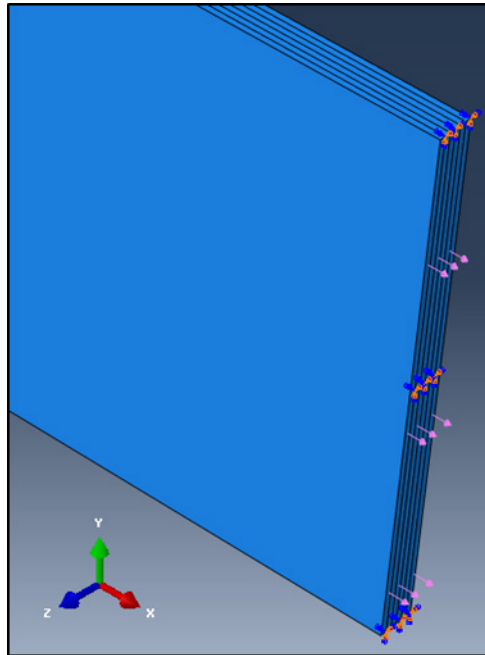


Figure 43. Quasi-Static Specimen FE Load and Boundary Conditions

The element type (linear-hexagonal, 3D solid) and sizing was kept consistent between the impact and quasi-static FE models. Although this resulted in a very fine mesh for the quasi-static model, the required computation time was still low for the quasi-static case since a dynamic-implicit timestep iteration was not required. The resulting quasi-static mesh produced 101,745 nodes and 85,200 elements for the six-layer samples (unidirectional and bidirectional). The element count was reduced for the 12-layer (quasi-isotropic) model to 60,000 elements. An increase above 85,200 elements for the quasi-

static model proved too cumbersome for the computational capabilities at hand, so the quasi-isotropic element count was reduced in order to reduce computation time. The mesh proved to be sufficiently refined even with only 60,000 elements. An ABAQUS image of the six-layer quasi-static mesh is provided in Figure 44.

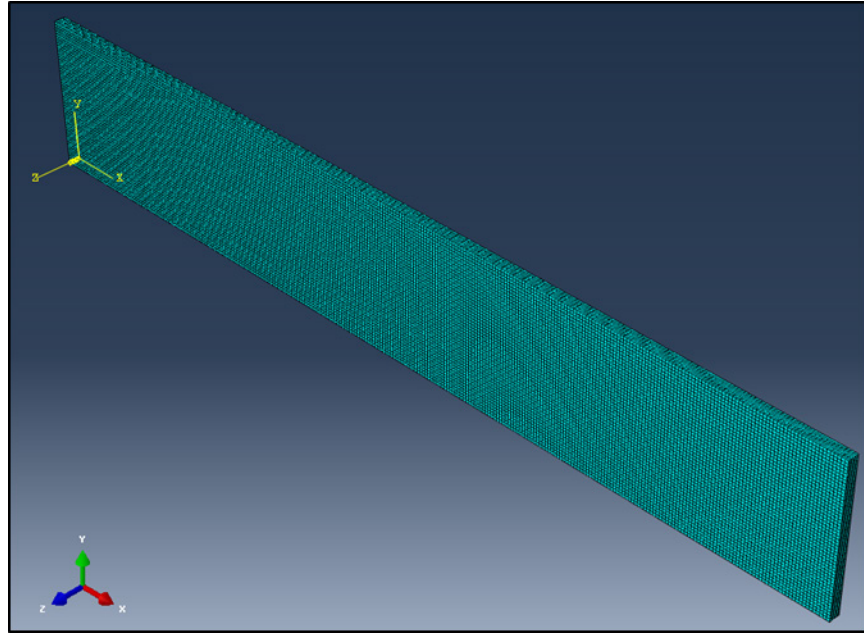


Figure 44. Quasi-Static Specimen FE Mesh

The aforementioned impact and quasi-static specimen FE model setups were utilized as the final iterations in the modeling process. All FE data were recorded directly from the ABAQUS (and corresponding UMAT subroutine) results gathered from the models described in Chapter VII, Section A. and B.

VIII. COMPARISON OF FINITE ELEMENT AND EXPERIMENTAL RESULTS

In accordance with the finite element model setups outlined in Chapter VII, the impact and quasi-static models were run, and all relevant stress and strain data were collected. Data were processed and plotted in MATLAB in order to compare the experimental and simulated results. All FE simulations were run in tandem with the composite model user subroutine described in [5]. In order to save computational effort and time, the impact specimen FE runs were distributed amongst four processor cores. All dynamic-implicit simulations were run using a 1e-06 s time step. This particular time step was chosen because it represented the exact data rate of the load cell that was used to take input load measurements.

A. IMPACT SPECIMEN FINITE ELEMENT STRESS RESULTS

With respect to the impact specimens, this thesis was primarily concerned with the comparison of the simulated versus experimental longitudinal strain results. However, the FE simulation provided insight into potential stresses experienced by each impact subset that could not be experimentally determined. The equivalent Von Mises stress distribution in several of the impact FE specimens offered insight into the inter-layer behavior of the composite layups. Figure 45 provides an overview of the equivalent Von Mises distribution in FE impact specimen, B2. The layers bearing a majority of the load are associated with fibers oriented parallel to the specimen long axis (0°). Regardless of the magnitude of the associated Von Mises stress, Figure 45 shows that the impact FE model was capable of appropriately handling the various fiber orientation layups.

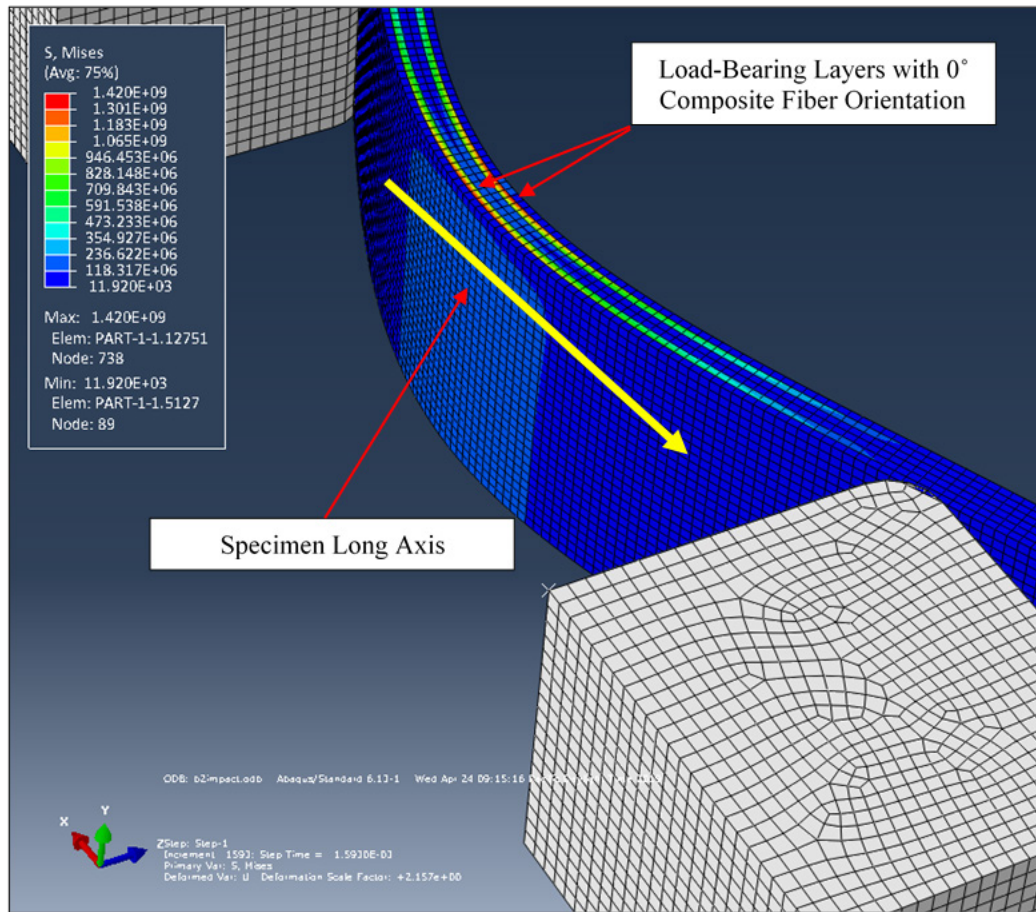


Figure 45. Equivalent Von-Mises Stress Distribution (MPa) – FE Impact Specimen B2

Simulated maximum stress values were collected to identify if the model was able to predict failure stress increases due to high strain rate effects; however, these stress values were bending stresses, so they differed in form from the experimental quasi-static uniaxial stresses. The maximum stress values collected from the simulated data were taken from the outermost layer of each FE impact specimen in the same location as the physically applied strain gauge. Instead of using equivalent Von Mises stresses at the aforementioned locations, the axial stresses parallel to each specimens' longitudinal axis were used. These predicted stress values, although of the bending stress variety, were most similar to the uniaxial stresses calculated from the tensile test load data during quasi-static testing. This allowed for a reasonable comparison between quasi-static stress data and predicted

increases in failure stress due to high strain rate effects. The FE-predicted maximum impact stress values are displayed in Table 14.

Table 14. Axial Failure Stresses – Impact FE and Quasi-Static Experimental

	U1	U2	B1	B2	QI1	QI2
FE Impact Axial Failure Stress (MPa)	1327 MPa	32.82 MPa	1398 MPa	1380 MPa	1368 MPa	1355 MPa
Quasi-Static Experimental Axial Failure Stress (MPa)	1351 MPa	51.46 MPa	1174 MPa	723.4 MPa	920.5 MPa	900.6 MPa

The results in Table 14 display a clear strain rate dependence in axial failure stress for subsets U1, B1, B2, QI1, and QI2. Contradictory to the expected result, the predicted high strain rate axial failure stress in subset U2 was lower than the experimental quasi-static value. This was not an expected result since the matrix-dominated subsets experienced the most significant strain rate dependence based on experimental results. Additionally, the high rate axial failure stress predicted by the subset B2 simulation appears to be too close in magnitude to its significantly stronger B1 counterpart. Tensile test data produced a quasi-static axial failure stress in subset B2 that was approximately 450.6 MPa less than subset B1. Although an increase in the strength behaviors of the matrix material (as a function of applied strain rate) was observed, the predicted high strain rate axial failure stress of subset B2 appears too great in magnitude.

It is likely that the difference in stress type (i.e. bending versus axial stress) can be associated with the unexpected results shown in Table 14. The FE mode may be accurately predicting stress values based on the various directional failure mechanisms, strain-rate strengthening effects, and composite layup interactions inherent to fiber-matrix composite high strain rate failure. In order to validate the tensile bending stress values produced by the impact FE models, future physical testing should include a mechanism for measuring accurate axial loads and strains (such as that used in a Hopkinson pressure bar test) so that experimental stress values may be derived.

B. IMPACT SPECIMEN FINITE ELEMENT STRAIN RESULTS

Finite element impact strain results were gathered from the center-most node on the outer layer of each FE impact specimen. In order to stay consistent with the strain readings from the experimental strain gauges, the ϵ_{11} (along the specimen's longitudinal axis) strain data at the aforementioned nodal location were extracted and plotted against the respective simulation time step values. This allowed for time-dependent strain plots to be extracted from each impact FE simulation with data gathered from the approximate location of the physical strain gauges. A graphic identifying the nodal strain data extraction point is shown in Figure 46.

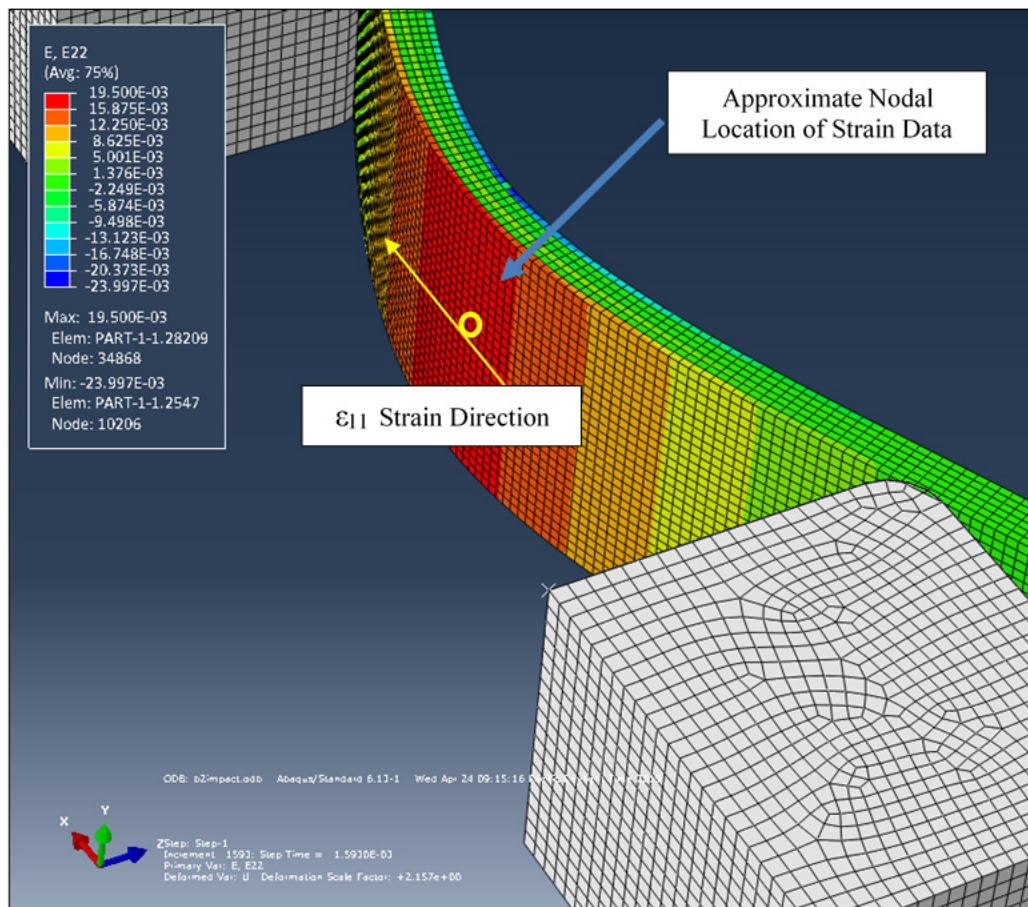


Figure 46. ABAQUS Image of Simulated Strain Data Nodal Location

FE strain data were collected up to the point of significant divergence from linearity in the strain versus time curve. This divergence was determined to be representative of model failure produced by user subroutine stiffness reductions. This allowed for a reasonable comparison of the model's performance leading up to the point of predicted specimen failure. All FE data produced after the failure point of each impact subset were not evaluated in this thesis.

1. Unidirectional FE Impact Strain Results

The entire FE high strain rate model was refined using the simulation data from FE impact subsets, U1 and U2. The high strain rate experimental data (45° impact hammer data only) are plotted in conjunction with the FE-determined high strain rate data in Figure 47.

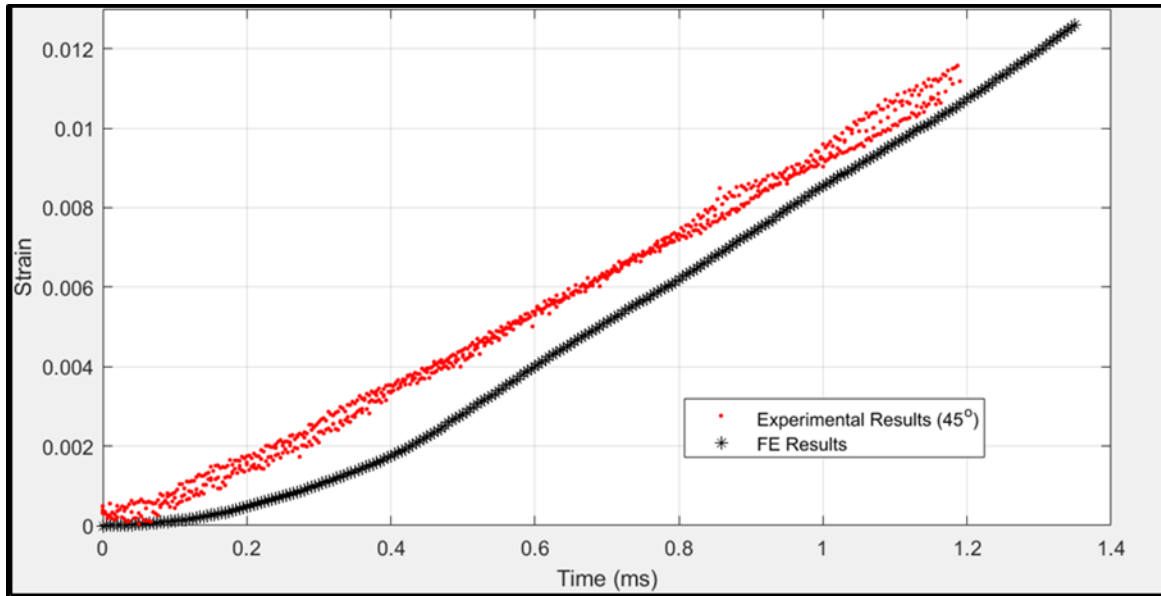


Figure 47. FE Impact Strain Results (with Experimental Data Shown) – Subset U1

The results in Figure 47 show a clear connection between the experimental and simulated strain data. The initial 0.4 ms of the FE strain data appear nonlinear. This effect was most likely caused by the small differences between the loading conditions of the

impact hammer contact surface and the load cell. After the initial 0.4 ms of simulated data, the FE model produced a similar strain rate to the experimental condition. The highly linear strain curve adopted by subset U1 was maintained by the high strain rate finite element model. Applying a linear curve fit to the FE data of impact subset U1 produced a strain rate of approximately 10.069 s^{-1} ; compared to the experimental 45° impact hammer strain rate of 9.853 s^{-1} . The proximity of the simulated and experimental strain rate values lends credence to the finite element model's fidelity for stiff fiber-matrix composite orientations such as that exhibited by subset U1. The FE model predicted a high rate failure strain of approximately 0.0126 in subset U1; compared to the experimental mean failure strain of 0.0119.

The FE high strain rate plot and experimental strain data with respect to a 45° impact hammer angle, for impact subset U2 is shown in Figure 48.

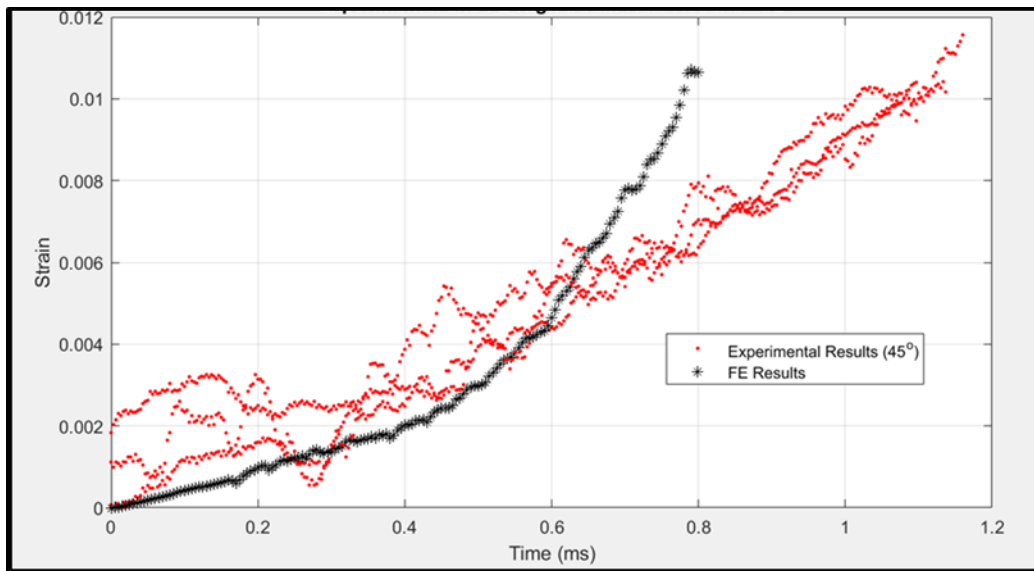


Figure 48. FE Impact Strain Results (with Experimental Data Shown) – Subset U2

The finite element data produced by impact subset U2 disagreed to a greater extent with the experimental data than its orthogonal, U1 counterpart. Additionally, the FE model did not produce a distinct failure point with notable divergence from strain curve linearity.

Due to this, the impact FE strain data for subset U2 were cutoff at the experimental mean failure strain of 0.0106. The initial FE data reasonably modeled the experimental condition, but after approximately 0.4 ms, the FE high strain rate model caused subset U2's strain readings to diverge from linearity. This may have been caused by the method in which the user subroutine handles subcell failure. [5] indicates that the unit cell model reduces the modulus of elasticity of a subcell by 99 % when failure occurs. This reduction value may need to be adjusted in order to produce more accurate impact FE results for subset U2. Additionally, the difference in loading conditions between the impact load cell and the pure Charpy impact hammer surface could have produced a higher strain rate in the finite element results since the FE model relied on load cell data. The subset U2 high strain rate model produced a linearized strain rate of approximately 11.737 s^{-1} . This simulated strain rate was slightly larger than the experimental rate of 10.007 s^{-1} .

2. Bidirectional FE Impact Strain Results

The high strain rate experimental data for impact subset B1 (45° impact hammer data only) are plotted in conjunction with the FE-determined high strain rate data in Figure 49.

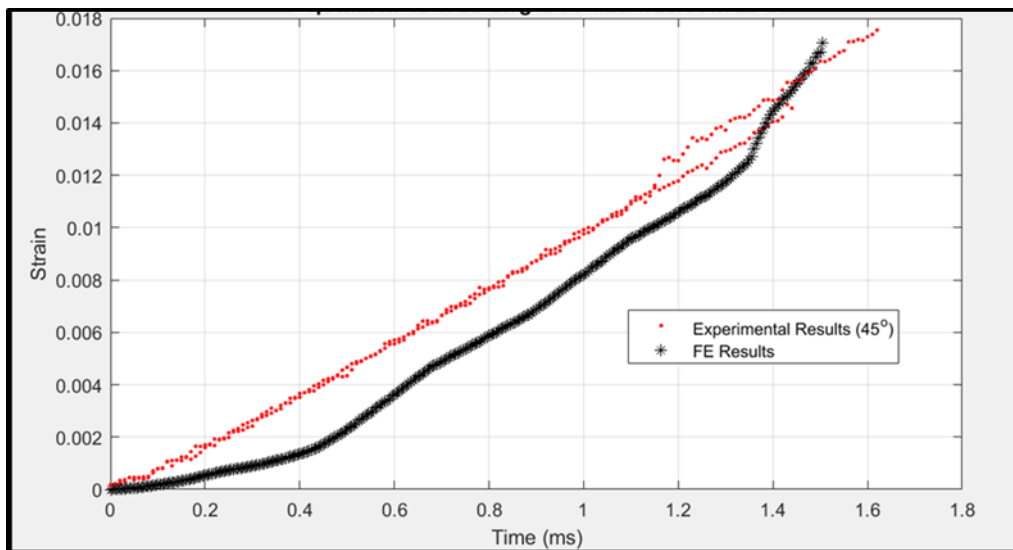


Figure 49. FE Impact Strain Results (with Experimental Data Shown) – Subset B1

The high strain rate finite element data generated by subset B1 experienced the same initial nonlinearity as subset U1. The remaining finite element strain data were in agreement with experimental strain readings. Interestingly, the finite element model produced a slight increase in strain rate during the final 0.1 ms of the simulated impact. This is indicative of fiber-dominated subcell failure, as dictated by the stiffness-reduction process carried out by the user subroutine. The subset B1 high strain rate finite element model produced a linearized strain rate of approximately 10.902 s^{-1} , which was 0.645 s^{-1} greater than the experimental mean strain rate of 10.357 s^{-1} . Additionally, the high strain rate subset B1 FE model predicted a failure strain of 0.0171, which is notably similar in magnitude to the experimental mean failure strain of 0.0178.

The high strain rate experimental data for impact subset B2 are plotted in conjunction with the FE-determined high strain rate data in Figure 50.

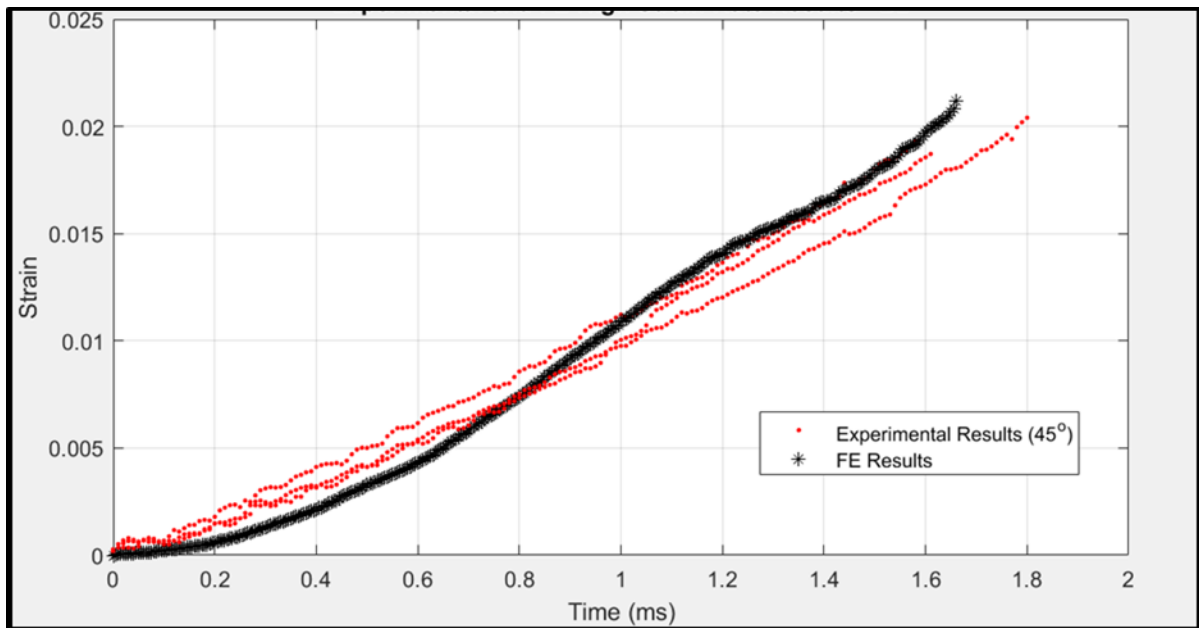


Figure 50. FE Impact Strain Results (with Experimental Data Shown) – Subset B2

Visual inspection of Figure 50 indicated that the high strain rate FE model produces strong agreement between simulated and experimental subset B2 impact strain behavior.

The FE model predicted a linearized strain rate of approximately 13.495 s^{-1} ; compared to the experimental value of 11.653 s^{-1} . Interestingly, the impact subset B2 FE model predicted a higher linearized strain rate in a similar manner to the subset U2 model. Subset B2 contained composite fiber orientations that resulted in more matrix-dominated layers, thus causing subset B2 to exhibit failure behaviors most similar to the fully-matrix-dominated subset, U2. The high strain rate finite element model also expressed this subset similarity. The FE-predicted high rate failure strain for subset B2 was approximately 0.0208, which is slightly greater than the experimental mean failure strain of 0.0195.

3. Quasi-Isotropic FE Impact Strain Results

The high strain rate experimental data for impact subset QI1 (45° impact hammer data only) are plotted in conjunction with the FE-determined high strain rate data in Figure 51.

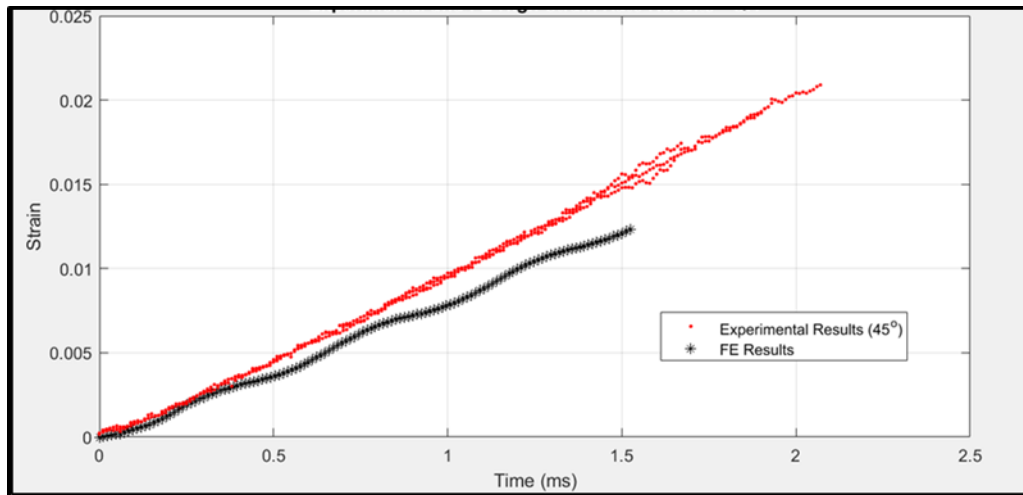


Figure 51. FE Impact Strain Results (with Experimental Data Shown) – Subset QI1

As shown in Figure 51, the impact finite element model did not entirely predict the high strain rate failure behavior of subset QI1. The subset QI1 impact FE model produces a strain curve that significantly diverges from linearity at a failure strain of approximately 0.0123, which is significantly less than the mean experimental failure strain of 0.0187. This

may have been indicative of the model predicting fiber failure in the outer layer (where strain measurements were taken) prior to total specimen failure. The FE strain data for impact subset QI1 produced behavior similar to that of subset U1. This concurs with reasonable expectations since the outer layers (where FE strain readings were taken) of subsets U1 and QI1 expressed identical fiber orientation. Further refinement of the quasi-isotropic high strain rate FE model could include adapting the user subroutine to better account for fiber-matrix interface interactions, as well as adjusting the stiffness reduction criteria of the subroutine as outlined in [5]. Additionally, the simulation produced a linearized strain rate of approximately 8.388 s^{-1} ; compared to the experimental 10.343 s^{-1} strain rate.

The high strain rate experimental data for impact subset QI2 (45° impact hammer data only) are plotted in conjunction with the FE-determined high strain rate data in Figure 52.

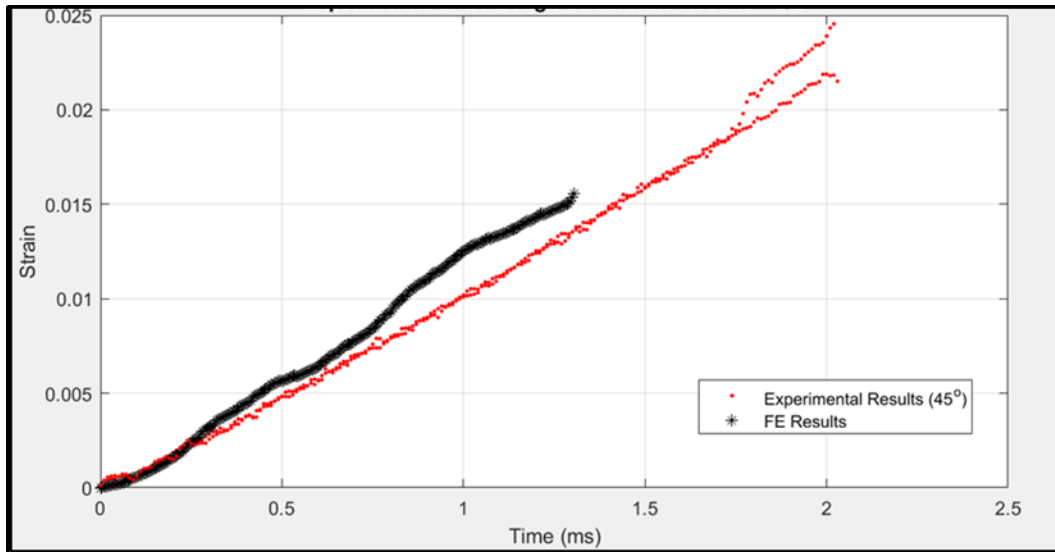


Figure 52. FE Impact Strain Results (with Experimental Data Shown) – Subset QI2

Figure 52 displays a similar result to the FE subset QI1 strain data, in which the simulated failure strain is significantly lower than the experimental value. The predicted failure strain of the impact subset QI2 model was 0.0156; compared to 0.0225 from the

experimental data. Additionally, the predicted strain rate of the subset QI2 high strain rate model was approximately 12.862 s^{-1} , which is notably higher than the 11.062 s^{-1} experimental mean strain rate.

Another potentially problematic modeling error could have originated in the magnitude of the time-dependent impact force used in the quasi-isotropic impact model. The actual peak forces measured during quasi-isotropic load cell testing were 400 N with a test duration of 0.9 ms. The linear extrapolation applied to the load curve (see Figure 36), however, was extended to nearly 800 N with a test duration of 1.6 ms. It is likely that the input load data used in the quasi-isotropic finite element model needs further refinement in order to accurately represent the physical system. Experimental impact subsets QI1 and QI2 produced the greatest failure strain values out of any of the three supplied carbon fiber types. This means that the quasi-isotropic impact specimens, due to their greater deformation, were effected the most by the load cell contact issue shown in Figure 33. To account for this error, future research should take load cell measurements using a load cell impact surface that does not allow the impact specimen to contact the outer load cell casing.

4. Comparison of FE and Experimental Impact Failure Loads

Upon obtaining FE impact failure strain data, the applied FE loads at each failure strain value were extracted and compared to the peak loads measured by the physical load cell. These impact failure load results are shown in Table 15.

Table 15. FE and Experimental Impact Failure Loads

	U1	U2	B1	B2	QI1	QI2
FE Impact Failure Load (N)	1018.93 N	39.28 N	909.67 N	484.17 N	581.74 N	495.23 N
Experimental Impact Failure Load (N)	709.54 N	44.45 N	534.99 N	252.06 N	470.85 N	470.85 N

Table 15 identifies the notable differences in magnitude between the experimental and FE failure load data. Subset U2 was the only specimen that did not produce a

significantly larger FE impact failure load when compared to its physically-measured counterpart. Despite the major differences in impact failure load order of magnitude, the FE-produced failure strains for the unidirectional and bidirectional impact subsets closely agreed with the experimental failure strain values. This lends proof to the observed intermittent unloading of the impact load cell, which caused low peak load measurements. Interestingly, the quasi-isotropic FE impact subsets produced peak failure loads that were notably closer in magnitude to the physically measured peak impact loads. Conversely, the quasi-isotropic FE impact subsets produced the least agreement between FE and experimental impact failure strain data. These two observations (with respect to quasi-isotropic impact data) indicate the necessity to refine the experimental impact load measurement process and the quasi-isotropic FE model itself.

5. Summary of FE Impact Strain Results

The high strain rate finite element model produced reasonable results in the stiffer specimens, and the specimens with a simpler composite layup structure (i.e. unidirectional and bidirectional). An accounting of the major FE impact strain data as they compare to experimental results are provided in Table 16.

Table 16. Summary of FE Impact Strain Results

	U1	U2	B1	B2	QI1	QI2
FE Failure Strain	0.0126	Indistinguishable	0.0171	0.0208	0.0123	0.0156
Experimental Failure Strain	0.0119	0.0106	0.0178	0.0195	0.0187	0.0225
Failure Strain Percent Error (%)	5.882 %	N/A	3.933 %	6.667 %	34.225 %	30.667 %
FE Strain Rate (s⁻¹)	10.069 s ⁻¹	11.737 s ⁻¹	10.902 s ⁻¹	13.495 s ⁻¹	8.388 s ⁻¹	12.862 s ⁻¹
Experimental Strain Rate (s⁻¹)	9.853 s ⁻¹	10.007 s ⁻¹	10.357 s ⁻¹	11.653 s ⁻¹	10.343 s ⁻¹	11.062 s ⁻¹
Strain Rate Percent Error (%)	2.192 %	17.288 %	5.262 %	15.807 %	18.902 %	16.272 %

The columns containing the subset QI1 and QI2 data are highlighted due to their significant under-prediction of the impact specimen failure strain. It should be noted that a single point of significant divergence from strain curve linearity was not identifiable in the impact strain results of FE subset U2; therefore, no percent error calculations were made with respect to U2 impact failure strain.

Table 16 shows that the complexity of the quasi-isotropic FE model and its corresponding inaccurate input force vector may have contributed to the major irregularities in FE impact data when compared to the experimental results. The specimen subsets with the highest 0°- oriented carbon fiber layers, U1 and B1, produced the least disparity between the experimental and finite element impact results. This indicates that further research is necessary with respect to strain rate effects as they apply to the matrix material as well as the fiber-matrix interface interaction. Additionally, loading conditions in the finite element model were slightly inaccurate due to the linear extrapolation assumption applied to each input load curve. To refine the model for future use, it is necessary to obtain as accurate of load data as possible. Additionally, the user subroutine must be adapted to better handle to properties inherent to the material at hand. The 99 % stiffness reduction in failed unit cell strength (dictated by the user subroutine) may be too significant of a reduction with respect to the matrix material (and potentially the fibers). Additional properties that are handled by the user subroutine—like the fiber and matrix interface strength—can also be adjusted to refine the impact model's fidelity. An additional source of error could have arisen in the element thickness of the quasi-isotropic impact model. Thin 3-D solid elements can produce FE error if the thickness of each element is too thin relative to the element width and height. The quasi-isotropic element thickness was reduced by half due to the increase to a 12-layer model. In order to maintain relative element thickness consistency, the QI1 and QI2 FE models would have required a prohibitively large amount of elements that the computation system could not handle.

Overall, the fiber-matrix composite impact FE model produced reasonable results with respect to the supplied carbon fiber material. There were notable discrepancies in strain rate and failure strain values for the quasi-isotropic high strain rate simulations, and subset U2 produced a large strain rate compared to its experimental value; but the overall

agreement between the finite element impact strain results and the experimental data was significant. This simulation-experimental agreement was enough to deem the finite element model a success in early development (with room for material property and force input improvements).

C. QUASI-STATIC SPECIMEN FINITE ELEMENT RESULTS

All quasi-static FE simulations were run in accordance with the model described in Chapter VII. Tensile specimen simulations were subject to a quasi-static finite element model rather than the time-stepped, dynamic implicit model applied to the high strain rate specimens. Strain data were gathered from the relevant node on the FE model that corresponded with the approximate location of the applied strain gauge on the experimental tensile specimens. Applied load magnitudes in the quasi-static model were derived from the maximum load measured by the tensile tester after each run.

The ABAQUS quasi-static model—in coordination with the user subroutine—produced reasonable stress distributions as shown in the example of tensile FE subset U1 in Figure 53.

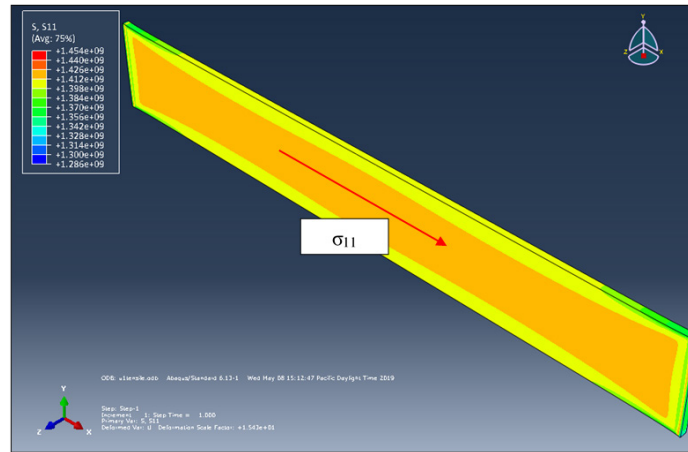


Figure 53. FE Stress Distribution (σ_{11}) in Tensile Subset U1

In order to stay consistent with the experimentally measured directional stresses and strains, both stress and strain data obtained from the quasi-static FE models were

gathered along the specimen's longitudinal axis, thus extracting the ε_{11} and σ_{11} global strain and stress tensor components (respectively). Obtaining mean engineering stress values from both unidirectional quasi-static subsets was trivial as each composite layer exhibited the same fiber orientation angle. Gathering mean engineering stress from the bidirectional and quasi-isotropic FE models, however, required examining the mean stress of each layer. Due to the nature of applying the partitioned-layer orientation scheme to each FE model, ABAQUS output stress tensor component distributions relative to local layer orientations. So, the σ_{11} distributions in the 90°- oriented layers were displayed orthogonal to those of the 0° layers. This same effect applied to the 45° layers in the quasi-isotropic model. To counter this effect, the local σ_{11} and σ_{22} values were gathered from a relevant node on each composite layer with a fiber orientation angle greater than or less than 0°. These 2-D stress tensor components were then multiplied by a coordinate transformation matrix defined by the orientation angle of the respective composite layer. The resulting coordinate-transformed σ_{11} components of each layer were used to find the mean engineering stress in each quasi-static FE model. Mean engineering stresses were obtained by multiplying the magnitude of FE-generated σ_{11} values (from a relevant node on each composite layer) by the number of layers with that specific orientation; then dividing the sum of all layer-multiplied stress readings by the total number of composite layers in the model. Table 17 displays the FE-obtained quasi-static engineering stress values of each specimen subset.

Table 17. Quasi-Static FE and Experimental Stress Results

	U1	U2	B1	B2	QI1	QI2
FE Engineering Stress (MPa)	1413 MPa	52.23 MPa	1215 MPa	728.4 MPa	820.8 MPa	793.6 MPa
Experimental UTS (MPa)	1351 MPa	51.46 MPa	1174 MPa	723.4 MPa	920.5 MPa	900.6 MPa
% Error	4.589 %	1.496 %	3.492 %	0.691 %	10.831 %	11.881 %

The accuracy of the quasi-static FE model produced much more reliable stress results when compared to the impact FE model. This is most likely due to the nature of the user subroutine. The user subroutine, as indicated in [5], was designed to handle quasi-

static models, so it would be advantageous to vectorize the user subroutine to better deal with dynamic loading conditions. The quasi-isotropic FE model, again, produced lower-than-expected mean engineering stress magnitudes. This may have been due to the coordinate transformation method that was required in order to obtain a reasonable estimate of FE-generated stresses. Composite finite element modeling does not produce uniform stress tensor component distributions throughout the thickness of a model because of the anisotropic nature of each distinct composite layer. To combat this effect, it could be beneficial to add an additional step to the user subroutine that transforms the stress tensor components of each element to a global (not local, layer-based) scale upon completion of the simulation.

Obtaining failure strain measurements from the FE model was much more trivial. All strain measurements on each quasi-static FE model were gathered from the approximate nodal location on the models that corresponded with the experimental strain gauge location. Since each outer layer where FE strain data were gathered was either 0° or 90° -oriented, the complex coordinate transformation process was not required. An example of the ϵ_{11} strain distribution on quasi-static FE subset U1 is shown in Figure 54.

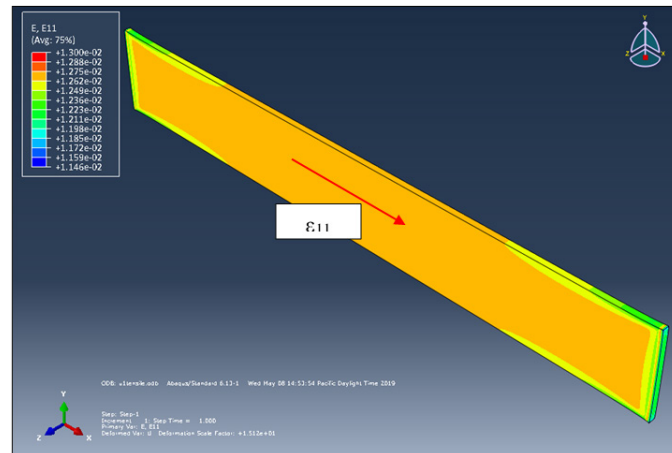


Figure 54. FE Strain Distribution (ϵ_{11}) in Tensile Subset U1

Quasi-static FE failure strain data from each subset are compared with the experimentally obtained strain results in Table 18.

Table 18. Quasi-Static FE and Experimental Strain Results

	U1	U2	B1	B2	QI1	QI2
FE Strain, ϵ_{11}	0.0127	0.0069	0.0155	0.0172	0.0183	0.0177
Experimental Strain at Failure, ϵ_{11}	0.0117	0.0066	0.0130	0.0159	0.0162	0.0161
% Error	8.547 %	4.546 %	19.231 %	8.176 %	12.963 %	9.938 %

Contrary to the mean stress underprediction by the quasi-static FE model, Table 18 indicates that quasi-isotropic specimens overpredicted the strain at failure of the relevant tensile samples. Interestingly, every quasi-static FE subset slightly overpredicted the ϵ_{11} strain at failure. This lends credence to the notion that the 99 % stiffness reduction criteria applied to failed unit cells—as dictated by the composite model user subroutine—is too significant for this particular material. Perhaps lower the unit cell stiffness reduction factor within the user subroutine could produce better results in future fiber-matrix composite simulations (both quasi-static and dynamic). Despite slight overpredictions in strain at failure, the quasi-static finite element model produced reliable results that closely match the experimentally obtained strain data.

THIS PAGE INTENTIONALLY LEFT BLANK

IX. CONCLUSIONS AND FUTURE WORK

A. CONCLUSIONS

Manufacture-supplied carbon fiber samples with three distinct fiber layer orientation schemes—unidirectional, bidirectional, and quasi-isotropic—were used in a comprehensive analysis of the high strain rate effects in fiber-matrix composite materials. The results of this experimental study were used to generate a finite element model capable of simulating high and quasi-static strain rate failure behaviors and characteristics of the fiber-matrix composite material at hand. The finite element model relied on a composite material user subroutine that utilized a multi-scale unit cell model technique—as described in [5] and [12].

High strain rate testing involved using a Charpy impact tester in conjunction with a high data rate DAQ system linked to strain gauges mounted along the longitudinal axis on the outer bending surface of each impact specimen. Impact testing proved successful in generating highly linear time-dependent strain curves, with distinct failure strain values, for impact specimen subsets U1, B1, B2, QI1, and QI2. Impact specimen subset U2 produced abnormal impact strain curves that were not distinctly linear, and a potential cause of this deviation from linearity may have arisen from the propagation of stress waves produced throughout each impact test duration. Despite difficulties with impact subset U2, impact test results allowed for a reasonable determination of the mean high strain rate and failure strain (averaged over multiple impact hammer angles) for each specimen subset. Quasi-static testing was performed using a standard tensile tester and a similar strain gauge setup. Quasi-static test results provided highly linear stress-strain curves and time-dependent strain curves that allowed for accurate determination of failure stresses and strains as well as applied quasi-static strain rates. Quasi-static subset B2 did not produce complete strain-time data sets due to premature outer layer failure that interfered with the applied strain gauges; however, the stress-strain data obtained from the tensile tester were accurately scaled in order to salvage the subset B2 quasi-static test data.

Upon completion of high strain rate and quasi-static testing, the relevant failure strain and strain rate data were applied to a failure strain correction function (Equation (2)) that produced a failure strain correction constant for each specimen subset. The correction constants were used to modify the material properties (primarily failure strain) of the fiber-matrix composite material in order to comply with property changes that result from an increased strain rate. The correction constants gathered from the subset U1 (fiber-dominated) and U2 (matrix-dominated) experimental data were used as multiplication factors to alter the material properties of the fiber and matrix material as functions of the applied strain rate. The matrix material failure strain multiplication factor was determined to be approximately 1.6061 (60.61 % increase) when subject to a high strain rate of approximately 10.007 s^{-1} . The fiber material produced a nearly rate-invariant failure strain multiplication factor at only 1.0171 (1.71 % increase) when subject to a high strain rate of 9.964 s^{-1} . In addition to strain rate corrections to material properties, impact load were required prior to generating the FE model. Impact load data were gathered using a load cell applied to the Charpy impact hammer contact surface. Load data produced relatively smooth time-dependent force curves; however, it was discovered that the sides of the load cell were partially relieving the load cell actuator upon significant deformation of the impact specimens. To counteract this effect, the load cell curve fits (used as the FE force inputs for the impact models) were fitted with linear curve fit extensions that extended beyond the potential load cell actuator relief points on each curve.

The finite element model was created using ABAQUS FE software in conjunction with the aforementioned user subroutine. The user subroutine required 15 material inputs in order to properly function. These material inputs were derived using the unit cell model outlined in [5] and [12]. Material inputs (such as fiber/matrix failure strain) for the high strain rate FE model were modified based on the relevant failure strain correction factors obtained from experimental data. Impact specimen FE results produced strain curves that were in close agreement with the experimental data for impact subsets U1, B1, and B2. The impact FE strain curves from subset U2 diverged from linearity in a similar manner to the experimental results, but at a notably higher strain rate. The FE impact strain curves for subsets QI1 and QI2 underestimated the experimental failure strain in both cases. This

deviation was determined to most likely be a result of the following conditions: too great of a specified stiffness reduction in failed unit cells (handled by the user subroutine), inaccuracies in load data inputs that could have produced premature failure, and 3-D element thickness issues due to the halving of element thickness in the quasi-isotropic model. The quasi-static FE model produced accurate failure stress and strain values that agreed with the experimental results. A potential source of small error in the quasi-static FE data potentially originated in the necessity to apply coordinate transformation techniques in order to obtain mean engineering stress values from the model. Overall, both the high strain rate and quasi-static finite element models predicted the fiber-matrix material behavior within a reasonable degree of agreement with experimental data. Further refinement is necessary in order to improve the performance of the high strain rate model—especially with respect to more complicated composite layer orientation schemes—but the finite element model can be considered an initial success due to its close agreement with experimental data in a majority of the specimen subsets.

B. FUTURE WORK

Future improvements to both the experimentation techniques and the FE model inputs used in this thesis can be made in order to increase the fidelity of fiber-matrix material high strain rate modeling. In order to better the performance of the FE model, the user subroutine can be tailored to match the material at hand. It may be beneficial in future studies to alter the aforementioned unit cell stiffness reduction percentage that is dictated by the user subroutine. The current percentage sits at 99%, but the FE results in this thesis proved that this percentage may not be entirely suitable for the particular fiber-matrix material at hand. The stiffness reduction factor could be incrementally reduced until the high strain rate FE model simulates experimental conditions to the greatest possible degree of fidelity. On the experimental side, future research should focus on creating a different method of high strain rate testing. The current Charpy impact method produced reliable data, but certain subsets (like subset U2) produced abnormal strain curves that require further refinement. A high strain rate test that allows for both strain and applied load measurements would best aid in generating more accurate material and load inputs for the FE model. Should the same high strain rate testing method be required, a load cell with a

longer actuator could be used to avoid the issue of the impact specimens relieving the load actuator through contact with the load cell casing. With respect to the user subroutine, a major improvement could be made to the way in which dynamic models are handled. The current user subroutine is tailored for use in a quasi-static simulation. The subroutine worked under dynamic-implicit conditions; however, vectorizing the subroutine for use in dynamic-explicit conditions could drastically improve the performance of the FE model.

APPENDIX A. EXAMPLE OF IMPACT DATA PROCESSING

```
[dat1 txt]=xlsread('U1_1.xlsx');
%Extrapolate strain data
e1=dat1(:,2)./4;
%Generate array indexing time steps for 100kHz specimens
t100(1)=1;
for i=2:length(dat1)
    t100(i)=t100(i-1)+1;
end
[dat2 txt]=xlsread('U1_2.xlsx');
e2=dat2(:,2)./4;
[dat3 txt]=xlsread('U1_3.xlsx');
e3=dat3(:,2)./4;
[dat4 txt]=xlsread('U1_4.xlsx');
e4=dat4(:,2)./4;
[dat5 txt]=xlsread('U1_5.xlsx');
e5=dat5(:,2)./4;
%Generate array indexing time steps for 300kHz specimens
t300(1)=1;
for i=2:length(dat5)
    t300(i)=t300(i-1)+1;
end
[dat6 txt]=xlsread('U1_6.xlsx');
e6=dat6(:,2)./4;
[dat7 txt]=xlsread('U1_7.xlsx');
e7=dat7(:,2)./4;
[dat8 txt]=xlsread('U1_8.xlsx');
e8=dat8(:,2)./4;
[dat9 txt]=xlsread('U1_9.xlsx');
e9=dat9(:,2)./4;
[dat10 txt]=xlsread('U1_10.xlsx');
e10=dat10(:,2)./4;
%Create time arrays for each data set based on relevant strain response
%region
ef1=e1(39839:40128);
t1=[0:1:length(ef1)-1].*1e-5;
ef2=e2(27456:27725);
t2=[0:1:length(ef2)-1].*1e-5;
ef3=e3(26249:26520);
t3=[0:1:length(ef3)-1].*1e-5;
ef4=e4(14950:15069);
t4=[0:1:length(ef4)-1].*1e-5;
ef5=e5(3412:3761);
t5=[0:1:length(ef5)-1].*((1/3)*1e-5);
ef6=e6(22970:23326);
t6=[0:1:length(ef6)-1].*((1/3)*1e-5);
ef7=e7(15279:15510);
t7=[0:1:length(ef7)-1].*((1/3)*1e-5);
ef8=e8(82616:82845);
t8=[0:1:length(ef8)-1].*((1/3)*1e-5);
ef9=e9(60028:60259);
t9=[0:1:length(ef9)-1].*((1/3)*1e-5);
ef10=e10(36021:36235);
t10=[0:1:length(ef10)-1].*((1/3)*1e-5);

figure(1);
plot(t1.*1000,ef1,'.b');
hold on;
plot(t4.*1000,ef4,'.r');
hold on;
plot(t7.*1000,ef7,'.g');
hold on;
plot(t2.*1000,ef2,'.b');
hold on;
plot(t3.*1000,ef3,'.b');
```

```

hold on;
plot(t5.*1000,ef5,'.r');
hold on;
plot(t6.*1000,ef6,'.r');
hold on;
plot(t8.*1000,ef8,'.g');
hold on;
plot(t9.*1000,ef9,'.g');
hold on;
plot(t10.*1000,ef10,'.g');
title('Variable High Strain Rate Plot - U1');
xlabel('Time (ms)');
ylabel('Strain');
grid on;
axis([0 3 0 0.014]);
legend('20° Impact Hammer Angle','45° Impact Hammer Angle','75° Impact Hammer Angle');

```

```

y1=ef1(70:250);
x1=t1(70:250)';
p1=polyfit(x1,y1,1);
y2=ef2(70:250);
x2=t2(70:250)';
p2=polyfit(x2,y2,1);
y3=ef3(70:250);
x3=t3(70:250)';
p3=polyfit(x3,y3,1);
y4=ef4(30:100);
x4=t4(30:100)';
p4=polyfit(x4,y4,1);
y5=ef5(70:320);
x5=t5(70:320)';
p5=polyfit(x5,y5,1);
y6=ef6(70:320);
x6=t6(70:320)';
p6=polyfit(x6,y6,1);
y7=ef7(70:205);
x7=t7(70:205)';
p7=polyfit(x7,y7,1);
y8=ef8(70:205);
x8=t8(70:205)';
p8=polyfit(x8,y8,1);
y9=ef9(70:205);
x9=t9(70:205)';
p9=polyfit(x9,y9,1);
y10=ef10(70:205);
x10=t10(70:205)';
p10=polyfit(x10,y10,1);

```

```

max20=[max(ef1),max(ef2),max(ef3)];
max45=[max(ef4),max(ef5),max(ef6)];
max75=[max(ef7),max(ef8),max(ef9),max(ef10)];

```

```

%Failure Strain
strainf20=mean(max20);
strainf45=mean(max45);
strainf75=mean(max75);
%Strain Rate
rate20=[p1(1),p2(1),p3(1)];
strainr20=mean(rate20);
rate45=[p4(1),p5(1),p6(1)];
strainr45=mean(rate45);
rate75=[p7(1),p8(1),p9(1),p10(1)];
strainr75=mean(rate75);
%Compile Results into Single Array
rate=[strainr20,strainr45,strainr75];
failstrain=[strainf20,strainf45,strainf75];
%Plot Results (U1 and U2)
rate2=[5.1834 10.0074 16.9139];
failstrain2=[0.0108 0.0108 0.0100];
figure(2);

```

```

plot(rate,failstrain,'om');
hold on;
plot(rate2,failstrain2,'ok');
grid on;
title('Failure Strain versus Strain Rate - U1 and U2');
xlabel('Strain Rate (1/s)');
ylabel('Failure Strain');
axis([4 18 0.009 0.013]);
legend('U1','U2');

emax=[max(ef1),max(ef2),max(ef3),max(ef4),max(ef5),max(ef6),max(ef7),max(ef8),max(ef9),max(ef10)];
e_dot_max=[p1(1),p2(1),p3(1),p4(1),p5(1),p6(1),p7(1),p8(1),p9(1),p10(1)];
ef_U1=mean(emax);
e_dot_U1=mean(e_dot_max);

[dat11 txt]=xlsread('U1impact_FE.xls');
tfe1=dat11(1:271,1);
efe1=dat11(1:271,2);
figure(4);
plot(t4.*1000,ef4,'.r');
hold on;
plot(tfe1.*1000,efe1,'*k');
hold on;
plot(t6.*1000,ef6,'.r');
hold on;
plot(t5.*1000,ef5,'.r');
title('Experimental and FE High Strain Rate Results - U1');
xlabel('Time (ms)');
ylabel('Strain');
grid on;axis([0 1.4 0 0.013]);
legend('Experimental Results (45^o)','FE Results');
%Calculate FE Strain Rate
y11=efe1(:);
x11=tfe1(:);
p11=polyfit(x11,y11,1);
strainrFE=p11(1);

```

Published with MATLAB® R2018a

THIS PAGE INTENTIONALLY LEFT BLANK

APPENDIX B. EXAMPLE OF QUASI-STATIC DATA PROCESSING

```
%[u1_1t,txt]=textread('U1_2_transverse_txt.txt');
%[u1_1r,txt]=textread('U1_2_radial_txt.txt');
[u1_2t,txt]=xlsread('U1_3_transverse.csv');
[u1_2r,txt]=xlsread('U1_3_radial.csv');
[u2_1t,txt]=xlsread('U2_1_transverse.csv');
[u2_1r,txt]=xlsread('U2_1_radial.csv');
[u2_2t,txt]=xlsread('U2_2_transverse.csv');
[u2_2r,txt]=xlsread('U2_2_radial.csv');

%U1_1 - DATA FILE TOO LARGE TO PROCESS
%strain_u11t=u1_1t(:,2);
%strain_u11t=strain_u11t-strain_u11t(1);
%time_u11t=u1_1t(:,1);
%time_u11t=time_u11t-time_u11t(1);
%strain_u11r=u1_1r(:,2);
%strain_u11r=strain_u11r-strain_u11r(1);
%time_u11r=u1_1r(:,1);
%time_u11r=time_u11r-time_u11r(1);
%U1_2
strain_u12t=u1_2t(17298:459235,2);
strain_u12t=strain_u12t-strain_u12t(1);
time_u12t=u1_2t(17298:459235,1);
time_u12t=time_u12t-time_u12t(1);
strain_u12r=u1_2r(17298:459235,2);
strain_u12r=strain_u12r-strain_u12r(1);
time_u12r=u1_2r(17298:459235,1);
time_u12r=time_u12r-time_u12r(1);
%B2_1
strain_u21t=u2_1t(5494:57006,2);
strain_u21t=strain_u21t-strain_u21t(1);
time_u21t=u2_1t(5494:57006,1);
time_u21t=time_u21t-time_u21t(1);
strain_u21r=u2_1r(5494:57006,2);
strain_u21r=strain_u21r-strain_u21r(1);
time_u21r=u2_1r(5494:57006,1);
time_u21r=time_u21r-time_u21r(1);
%B2_2
strain_u22t=u2_2t(4641:51817,2);
strain_u22t=strain_u22t-strain_u22t(1);
time_u22t=u2_2t(4641:51817,1);
time_u22t=time_u22t-time_u22t(1);
strain_u22r=u2_2r(4641:51817,2);
strain_u22r=strain_u22r-strain_u22r(1);
time_u22r=u2_2r(4641:51817,1);
time_u22r=time_u22r-time_u22r(1);

%Tensile
figure(1);
%plot(time_u11t,strain_u11t,'.b'); - DATA FILE TOO LARGE
%hold on;
plot(time_u12t,strain_u12t,'.b');
hold on;
plot(time_u21t,strain_u21t,'.r');
hold on;
plot(time_u22t,strain_u22t,'.r');
grid on;
title('Slow Strain Rate Tensile Results - U1 and U2');
xlabel('Times (s)');
ylabel('Strain');
legend('U1 Orientation','U2 Orientation');
axis([0 250 0 0.014]);
ax = gca;
ax.XRuler.Exponent=0;
ay=gca;
```

```

ay.YRuler.Exponent=0;
    %Transverse
figure(2);
%plot(time_u11r,strain_u11r,'.b'); - DATA FILE TOO LARGE
%hold on;
plot(time_u12r,strain_u12r,'.b');
hold on;
plot(time_u21r,strain_u21r,'.r');
hold on;
plot(time_u22r,strain_u22r,'.r');
grid on;
title('Slow Strain Rate Transverse Results - B1 and B2');
xlabel('Times (s)');
ylabel('Strain');
ax = gca;
ax.XRuler.Exponent=0;
ay=gca;
ay.YRuler.Exponent=0;

```

```

    %Calculate Strain Rates
%y1=strain_u11t(:);
%x1=time_u11t(:);
    %[mb11 hb11]=polyfit(x1,y1,1);
y2=strain_u12t(39774:402169);
x2=time_u12t(39774:402169);
    [mu12 hu12]=polyfit(x2,y2,1);
y3=strain_u21t(6015:39934);
x3=time_u21t(6015:39934);
    [mu21 hu21]=polyfit(x3,y3,1);
y4=strain_u22t(6040:39177);
x4=time_u22t(6040:39177);
    [mu22 hu22]=polyfit(x4,y4,1);
    %Calculate Final Failure Strain
%fail_u11t=max(strain_b11t);
fail_u11t=0.01124; %From large .txt file
fail_u12t=max(strain_u12t);
fail_u21t=max(strain_u21t);
fail_u22t=max(strain_u22t);

```

```

eSlow_dot_U1=mean([mu12(1)]);
eSlow_dot_U2=mean([mu21(1),mu22(1)]);
efSlow_U1=mean([fail_u11t,fail_u12t]);
efSlow_U2=mean([fail_u21t,fail_u22t]);

```

```

[u11,txt]=xlsread('U1_2_1.csv');
t11=u11(:,1); %time in seconds
e11=(u11(:,2)./91.3638); %strain
s11=u11(:,3)./(0.07474*645.16); %stress in MPa
[u12,txt]=xlsread('U1_3_1.csv');
t12=u12(:,1); %time in seconds
e12=0.15.*(u12(:,2)./91.6432); %strain
e12=e12-0.0002761;
s12=u12(:,3)./(0.076125*645.16); %stress in MPa
[u21,txt]=xlsread('U2_1_1.csv');
t21=u21(:,1); %time in seconds
e21=0.769.*(u21(:,2)./94.7674); %strain
s21=u21(:,3)./(0.075675*645.16); %stress in MPa
[u22,txt]=xlsread('U2_2_1.csv');
t22=u22(:,1); %time in seconds
e22=0.746.*(u22(:,2)./94.6404); %strain
s22=u22(:,3)./(0.076608*645.16); %stress in MPa

```

```

figure(3);
plot(e12,s12,'.b');
hold on;
plot(e21,s21,'.r');
hold on;
plot(e22,s22,'.r');
grid on;
title('Slow Strain - Stress/Strain Curves - U1 and U2');

```

```

xlabel('Strain');
ylabel('Stress (MPa)');
legend('U1 Orientation','U2 Orientation');
axis([0 0.014 0 1450]);
ax = gca;
ax.XRuler.Exponent=0;
ay=gca;
ay.YRuler.Exponent=0;
figure(4);
plot(e21,s21,'.r');
hold on;
plot(e22,s22,'.r');
title('Slow Strain - Stress/Strain Curves - U2');
xlabel('Strain');
ylabel('Stress (MPa)');
grid on;
ax = gca;
ax.XRuler.Exponent=0;
ay=gca;
ay.YRuler.Exponent=0;

```

[Published with MATLAB® R2018a](#)

THIS PAGE INTENTIONALLY LEFT BLANK

APPENDIX C. INTERACTING WITH THE USER SUBROUTINE

The following appendix describes the necessary steps required to correctly interface the user subroutine with a finite element model developed in ABAQUS FE software. These steps apply to both dynamic-implicit and quasi-static FE models.

Prior to interfacing with the user subroutine, generate a complete FE model (including meshing). If the model requires composite layering (such as that in this thesis), partition the model along the composite layering direction in accordance with the required number of layers. Assign orientation angles to each layer in the ABAQUS graphic-user interface (GUI). Shell elements were found to be a source of user subroutine failure during dynamic-implicit modeling; therefore, 3-D continuum solid elements should be used in any future dynamic-implicit models that utilize this specific user subroutine. The composite layering option in the ABAQUS GUI also produced user subroutine errors, so the aforementioned model partitioning method should be used for any future dynamic models. Once the model partitioning (to include assigning orientation angles) and meshing are complete, a user material must be created. The ABAQUS GUI allows for the creation of a user material with a defined number of property inputs. The user subroutine requires 15 property inputs in accordance with those defined in Table 12 and Table 13. When defining the user material, the number of dependent variables that the subroutine interacts with must also be defined. This input is under the title, ‘Depvar’ in the ABAQUS user material GUI. With respect to the current user subroutine, the number of dependent variables must be set to 58. An additional material density input is required for dynamic-implicit simulations. An ABAQUS GUI image of the user material inputs used in the dynamic-implicit FE simulations is provided in Figure 55.

Name: unidirectional

Description:

Material Behaviors

Density

Depvar

User Material

General Mechanical Thermal Electrical/Magnetic Other

User Material

User material type: Mechanical

☐ Use unsymmetric material stiffness matrix

Data

	Mechanical Constants
1	220000000000
2	280000000000
3	300000000000
4	0.12
5	0.4
6	3000000000
7	1154000000
8	0.3
9	0.5
10	0.0119
11	0.0119
12	0.0106
13	0.0106
14	148192000
15	86729400

OK Cancel

Figure 55. Example of User Subroutine Material Property Input GUI

After generating the required material property inputs, interaction with the user subroutine is minimal. When an ABAQUS job is created, in the main job menu select the desired file path to the user subroutine Fortran code in the 'User Subroutine' input menu. Ensure that the user subroutine Fortran file is in the same folder as the ABAQUS model. After creating an ABAQUS job with the defined user subroutine file path, no further action regarding the user subroutine is necessary in order for it to properly function.

LIST OF REFERENCES

- [1] DragonPlate Composite Manufacturer, “What is carbon fiber?” [Online]. Available: www.dragonplate.com/what-is-carbon-fiber
- [2] Hexcel Inc, “HexTow IM7 carbon fiber,” 2016. [Online]. Available: www.hexcel.com/user_area/content_media/raw/IM7/HexTow_DataSheet.pdf
- [3] Hexcel Inc, “HexPly 8551-7 epoxy matrix,” 2005. [Online]. Available: hexcel.com/user_area/content_media/raw/HexPly_85571_us_DataSheet.pdf
- [4] A.C. Ugural and S.K. Fenster, *Advanced Strength and Applied Elasticity*, 4th ed. Prentice Hall, Upper Saddle River, NJ, 2003.
- [5] J. Darcy, “Failure model for fibrous composites using multiscale modeling,” PhD Dissertation, Mechanical Engineering Department, NPS, Monterey, CA, USA, 2018.
- [6] G.C. Jacob, J.M. Starbuck, J.F. Fellers, and R.G. Boeman, “Strain rate effects on the mechanical properties of polymer composite materials,” *Journal of Applied Polymer Science*, vol. 94, no. 1, pp. 298, Sep. 2004.
- [7] H.M. Hsiao and I.M. Daniel, “Strain rate behavior of composite materials,” *Composites Part B: Engineering*, vol. 29, no. 5, pp. 521, Sep. 1998.
- [8] Torray Composite Materials America Inc., “T700S: standard modulus carbon fiber,” 2019. [Online]. Available: www.toraycma.com/page.php?id=661
- [9] Newport Adhesives and Composites Inc., “Newport 301 product data sheet,” 2006. [Online]. Available: [www.rockwestcomposites.com/downloads/Newport-301-Data_\(2006\).pdf](http://www.rockwestcomposites.com/downloads/Newport-301-Data_(2006).pdf)
- [10] Honeywell International Inc., “Model 13: Subminiature Load Cell,” 2009. [Online]. Available: sensing.honeywell.com/honeywell-t&m-model-13-subminiature-load-cell-product-sheet-008641-2-en.pdf
- [11] I.M. Daniel, B.T. Werner, and J.S. Fenner, “Strain-rate-dependent failure criteria for composites,” *Composite Science and Technology*, vol. 71, no. 3, pp. 357-364, 2011.
- [12] Y.W. Kwon and J. Darcy, “Failure criteria for fibrous composites based on multiscale modeling,” *Multiscale and Multidiscip. Model. Exp. and Des.*, no. 1, pp. 3-17, 2018.

- [13] S. Swanson, M. Messick, and Z. Tian, "Failure of carbon/epoxy lamina under combined stress," *Journal of Composite Materials*, vol. 21, no. 7, pp. 620-629, Jul. 1987.
- [14] S.W. Tsai and E.M. Wu, "A general theory of strength for anisotropic materials," *Journal of Composite Materials*, vol. 5, no. 1, pp. 59-79, Jan. 1971.
- [15] Society of Automotive Engineers, *Composite Materials Handbook*, SAE International on behalf of CMH-17, a division of Wichita State University, 2012. [Online]. Available: www.worldcat.org/isbn/9780768078138
- [16] R.L. Sierakowski, "Strain rate effects in composites," *Applied Mechanics Reviews*, vol. 50, no. 12, pp. 741-759, Dec. 1997.
- [17] *Abaqus 6.13 User Manual*, Dassault Systemes Inc., 2015.
- [18] Y.W. Kwon and J. Berner, "Micromechanics model for damage and failure analyses of laminated fibrous composites," *Engineering Fracture Mechanics*, vol. 52, no. 2, pp. 231-242, Sep. 1995.
- [19] Y.W. Kwon and M.S. Park, "Versatile micromechanics model for multiscale analysis of composite structures," *Applied Composite Materials*, vol. 20, no. 1, pp. 21-39, Jan. 1998.
- [20] M. Hinton, A. Kaddour, and P. Soden, *Failure Criteria in Fibre Reinforced Polymer Composites: The World-Wide Failure Exercise*, M. Hinton, A. Kaddour, and P. Soden, Eds. Elsevier, Amsterdam, 2004.

INITIAL DISTRIBUTION LIST

1. Defense Technical Information Center
Ft. Belvoir, Virginia
2. Dudley Knox Library
Naval Postgraduate School
Monterey, California