CRASH PERFORMANCE OF PRE-IMPREGNATED PLATELET BASED MOLDED COMPOSITES

A Dissertation
Submitted to the Faculty
of
Purdue University
by
Rebecca A. Cutting

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

August 2019
Purdue University
West Lafayette, Indiana
THE PURDUE UNIVERSITY GRADUATE SCHOOL
STATEMENT OF DISSERTATION APPROVAL

Dr. R. Byron Pipes, Co-Chair
School of Aeronautics and Astronautics
Dr. Johnathan E. Goodsell, Co-Chair
School of Aeronautics and Astronautics
Dr. Weinong Chen
School of Aeronautics and Astronautics
Dr. Tyler Tallman
School of Aeronautics and Astronautics

Approved by:
Dr. Weinong Chen
Head of the School Graduate Program
For Audrey Beale and Mary Beale Cutting
ACKNOWLEDGMENTS

This work has been a journey that would not have been completed without the help and support of my family, friends, colleagues, and mentors.

First I would like to thank Dr. Goodsell for introducing me to the world of composites in 2010 and mentoring me in my life and career ever since. I am truly grateful for all that you and your family have done for me. I would also like to thank Dr. Pipes for taking a chance on me. Thank you for your kindness, understanding, and nudges in the right direction when I needed them.

Thanks to Dr. Chen for the use of his high speed test facilities, Dr. Tsutsui for his help with drop tower testing, and Gillum Machining for the light-speed turn around times on fixture designs.

Thanks to my colleagues Dr. Ben Denos, Dr. Anthony Favaloro, and Dr. Drew Sommer for their endless theoretical debates, support in times of pants-on-fire crises, and most importantly, friendship. Thanks to Federico Tascon for his tireless work over the last year in improving our manufacturing and testing capabilities.

Additional thanks are due to my mentors throughout the years: my high school calculus teacher Mr. Blue, Ken Ferguson at ATA Engineering, and Don Morris, Jen Potter, and Mark Proulx at Boeing’s SDL who taught me everything I know about testing and being a rock-star engineer.

I would like to thank my family: my mother and grandmother for giving me examples of strong, persevering women, my dad who planned my dive into engineering with our K’NEX and LEGO build sessions, my sister for always giving me a good laugh, and my brother for always bringing me back down to Earth. Thanks to my grandpa, Dr. A. F. Beale, Jr. for giving me my first chemistry set (that I promptly melted) and giving me an academic goal to strive for. Finally, thanks to my fiancé, Kyle Schwinn, for the love and support through all of this.
TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>LIST OF TABLES</td>
<td>viii</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>x</td>
</tr>
<tr>
<td>ABBREVIATIONS</td>
<td>xvi</td>
</tr>
<tr>
<td>ABSTRACT</td>
<td>xvii</td>
</tr>
<tr>
<td>1 INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>1.1 Molding Compounds</td>
<td>2</td>
</tr>
<tr>
<td>1.1.1 Bulk Molding Compound and Sheet Molding Compound</td>
<td>2</td>
</tr>
<tr>
<td>1.1.2 Prepreg Platelet Molding Compound</td>
<td>3</td>
</tr>
<tr>
<td>1.2 Crashworthiness</td>
<td>5</td>
</tr>
<tr>
<td>1.3 Problem Statement</td>
<td>6</td>
</tr>
<tr>
<td>2 BACKGROUND ON EXPERIMENTAL CRUSH TUBES</td>
<td>8</td>
</tr>
<tr>
<td>2.1 Variables of Interest</td>
<td>8</td>
</tr>
<tr>
<td>2.1.1 Failure Morphology</td>
<td>8</td>
</tr>
<tr>
<td>2.1.2 Load-Displacement Curve</td>
<td>10</td>
</tr>
<tr>
<td>2.1.3 Specific Energy Absorption</td>
<td>12</td>
</tr>
<tr>
<td>2.2 Metal Tube Studies</td>
<td>13</td>
</tr>
<tr>
<td>2.3 Continuous Fiber Tube Studies</td>
<td>15</td>
</tr>
<tr>
<td>2.3.1 Effect of Material System</td>
<td>15</td>
</tr>
<tr>
<td>2.3.2 Effect of Processing Conditions</td>
<td>17</td>
</tr>
<tr>
<td>2.3.3 Effect of Geometry</td>
<td>18</td>
</tr>
<tr>
<td>2.3.4 Effect of Testing Parameters</td>
<td>19</td>
</tr>
<tr>
<td>2.4 Discontinuous Fiber Tube Studies</td>
<td>19</td>
</tr>
<tr>
<td>3 EXPERIMENTAL WORK ON CRUSH TUBES</td>
<td>21</td>
</tr>
<tr>
<td>3.1 Manufacturing the Tubes</td>
<td>21</td>
</tr>
<tr>
<td>3.1.1 3D Printed Composite Tool</td>
<td>22</td>
</tr>
<tr>
<td>3.1.2 Aluminum Tool</td>
<td>29</td>
</tr>
<tr>
<td>3.1.3 Sample Preparation</td>
<td>32</td>
</tr>
<tr>
<td>3.2 Quasi-Static Test Method</td>
<td>33</td>
</tr>
<tr>
<td>3.3 Results</td>
<td>33</td>
</tr>
<tr>
<td>3.3.1 Metal Study</td>
<td>33</td>
</tr>
<tr>
<td>3.3.2 Continuous Fiber Tests</td>
<td>35</td>
</tr>
<tr>
<td>3.3.3 PPMC Tests</td>
<td>44</td>
</tr>
<tr>
<td>3.4 Conclusions on Quasi-Static Testing</td>
<td>58</td>
</tr>
<tr>
<td>Section</td>
<td>Title</td>
</tr>
<tr>
<td>---------</td>
<td>----------------------------------------------------------------------</td>
</tr>
<tr>
<td>3.5</td>
<td>Dynamic Testing</td>
</tr>
<tr>
<td>3.5.1</td>
<td>Drop Tower Fixture Design</td>
</tr>
<tr>
<td>3.5.2</td>
<td>Dynamic Test Method</td>
</tr>
<tr>
<td>3.5.3</td>
<td>Results</td>
</tr>
<tr>
<td>3.5.4</td>
<td>Conclusions from Dynamic Testing of PPMC Tubes</td>
</tr>
<tr>
<td>4</td>
<td>SIMULATING CRASH BEHAVIOR OF COMPOSITES</td>
</tr>
<tr>
<td>4.1</td>
<td>Finite Element Software</td>
</tr>
<tr>
<td>4.2</td>
<td>Material Models for Crash Simulation of Composites in LS Dyna</td>
</tr>
<tr>
<td>4.2.1</td>
<td>MAT 54</td>
</tr>
<tr>
<td>4.2.2</td>
<td>MAT 58</td>
</tr>
<tr>
<td>4.3</td>
<td>Modeling Methods for Composites in Crash Using LS Dyna</td>
</tr>
<tr>
<td>4.3.1</td>
<td>Element Formulations</td>
</tr>
<tr>
<td>4.3.2</td>
<td>Time Step Calculations</td>
</tr>
<tr>
<td>4.3.3</td>
<td>Contact Definitions</td>
</tr>
<tr>
<td>4.3.4</td>
<td>Post Processing Methods</td>
</tr>
<tr>
<td>4.3.5</td>
<td>Material Characterization</td>
</tr>
<tr>
<td>4.3.6</td>
<td>Modeling PPMC in Crash</td>
</tr>
<tr>
<td>5</td>
<td>UNIDIRECTIONAL MODELS IN LS DYNA</td>
</tr>
<tr>
<td>5.1</td>
<td>Creation of the MAT 54 and MAT 58 Unidirectional Material Models</td>
</tr>
<tr>
<td>5.1.1</td>
<td>Experimental Material Characterization</td>
</tr>
<tr>
<td>5.1.2</td>
<td>Material Characterization Compared to Manufacturer Reported Properties</td>
</tr>
<tr>
<td>5.1.3</td>
<td>Single-Element and Four-Element Models</td>
</tr>
<tr>
<td>5.2</td>
<td>Element Deletion in FEA with Brittle Materials in Compression</td>
</tr>
<tr>
<td>5.2.1</td>
<td>Mesh Density Study</td>
</tr>
<tr>
<td>5.2.2</td>
<td>1D Model</td>
</tr>
<tr>
<td>5.3</td>
<td>Flexure Coupons</td>
</tr>
<tr>
<td>5.3.1</td>
<td>Model Setup</td>
</tr>
<tr>
<td>5.3.2</td>
<td>Impact Speed Study</td>
</tr>
<tr>
<td>5.3.3</td>
<td>Verification of Flexure Model</td>
</tr>
<tr>
<td>5.3.4</td>
<td>Validation with Experiment</td>
</tr>
<tr>
<td>5.3.5</td>
<td>Conclusions on Unidirectional Flexure Simulations</td>
</tr>
<tr>
<td>5.4</td>
<td>Crush Tubes</td>
</tr>
<tr>
<td>5.4.1</td>
<td>Model Setup</td>
</tr>
<tr>
<td>5.4.2</td>
<td>Validation with Experiment</td>
</tr>
<tr>
<td>5.5</td>
<td>Conclusions Regarding Unidirectional Modeling in LS Dyna</td>
</tr>
<tr>
<td>6</td>
<td>MANUFACTURING-INFORMED PERFORMANCE MODELS</td>
</tr>
<tr>
<td>6.1</td>
<td>Orientation Distribution Function</td>
</tr>
<tr>
<td>6.2</td>
<td>Modeling Approach for PPMC</td>
</tr>
<tr>
<td>6.3</td>
<td>Orientation Mapping Process</td>
</tr>
<tr>
<td>6.3.1</td>
<td>Generating Digital PPMC Components</td>
</tr>
<tr>
<td>Section</td>
<td>Title</td>
</tr>
<tr>
<td>---------</td>
<td>-------</td>
</tr>
<tr>
<td>6.4</td>
<td>Mapped Flexure Samples</td>
</tr>
<tr>
<td>6.4.1</td>
<td>Platelet Collimation Sensitivity Study</td>
</tr>
<tr>
<td>6.4.2</td>
<td>Verification of PPMC Flexure Models with an Analytical Solution</td>
</tr>
<tr>
<td>6.4.3</td>
<td>Validation with Experiment</td>
</tr>
<tr>
<td>6.5</td>
<td>Mapped Crush Tubes</td>
</tr>
<tr>
<td>6.5.1</td>
<td>Sensitivity Study</td>
</tr>
<tr>
<td>6.5.2</td>
<td>t/D Study</td>
</tr>
<tr>
<td>6.5.3</td>
<td>Validation with Experiment</td>
</tr>
<tr>
<td>6.6</td>
<td>Mapping Advanced Geometries</td>
</tr>
<tr>
<td>7</td>
<td>CONCLUSIONS</td>
</tr>
<tr>
<td>7.1</td>
<td>Review of Work</td>
</tr>
<tr>
<td>7.2</td>
<td>Limitations in Crash Modeling</td>
</tr>
<tr>
<td>7.3</td>
<td>Future Work</td>
</tr>
<tr>
<td>REFERENCES</td>
<td></td>
</tr>
<tr>
<td>A</td>
<td>EXAMPLE LS DYNA KEYWORD FILE</td>
</tr>
<tr>
<td>B</td>
<td>DROP TOWER TEST FIXTURE SCHEMATICS</td>
</tr>
<tr>
<td>VITA</td>
<td></td>
</tr>
</tbody>
</table>
LIST OF TABLES

Table                                      Page
3.1 Laminates and number of tubes manufactured for continuous fiber study   42
3.2 Geometry measurements of PPMC tubes                                      46
3.3 Platelet measurements from microscopy                                    49
3.4 Performance of PPMC tubes                                               55
3.5 Drop tower test specifications                                            67
3.6 Max load, displacement, energy absorbed, and SEA calculated for drop    71
tower test samples
3.7 Energy absorption comparison between dynamic and quasi-static tests      72
4.1 LS Dyna MAT 54 inputs - Dynamic Parameters                               81
4.2 LS Dyna MAT 54 inputs - Static Parameters                                82
4.3 LS Dyna MAT 58 inputs - Dynamic Parameters                               86
4.4 LS Dyna MAT 58 inputs - Static Parameters                                87
5.1 VORAFUSE M6400™ material characterization                                99
5.2 Material properties from Dow data sheet compared to experimentally char- 103
   acterized properties
5.3 Flexural modulus, strength, and computational time for speed study models126
5.4 Estimated flexural modulus for [0/90]4s layup from different analysis methods130
5.5 Experimental flexural modulus, strength, and failure strain versus span- 132
to-thickness ratio
5.6 Experimental flexural modulus and strength versus simulation            133
5.7 Model specifications for each simulation in the axial fiber percentage study 148
5.8 Model specifications for each simulation with an increased number of in- 160
tegration points
6.1 Average flexural modulus and strength for simulation models with increas- 178
ing amounts of platelet collimation
<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.2  Manufacturing charge dimensions and final thickness for experimental flexure samples</td>
<td>182</td>
</tr>
<tr>
<td>6.3  Modulus and strength results of experimental PPMC flexure testing</td>
<td>183</td>
</tr>
<tr>
<td>6.4  Estimated level of collimation for manufactured samples according to plate and sample longitudinal axis using Jeffery’s equation</td>
<td>184</td>
</tr>
<tr>
<td>6.5  Estimated level of collimation for manufactured samples from microscopy measurements</td>
<td>187</td>
</tr>
<tr>
<td>6.6  Thickness, inner diameter, and number of platelets for t/D model study</td>
<td>193</td>
</tr>
</tbody>
</table>
## LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Example forms of PPMC</td>
<td>4</td>
</tr>
<tr>
<td>1.2</td>
<td>Ease of processing versus performance for fiber composites</td>
<td>4</td>
</tr>
<tr>
<td>2.1</td>
<td>Continuous fiber failure modes presented in literature</td>
<td>9</td>
</tr>
<tr>
<td>2.2</td>
<td>Generic load-displacement curve of composite tube during progressive failure</td>
<td>11</td>
</tr>
<tr>
<td>2.3</td>
<td>Example concertina and diamond collapse modes in metal tubes</td>
<td>14</td>
</tr>
<tr>
<td>2.4</td>
<td>A sample of specific energy absorption dependencies</td>
<td>16</td>
</tr>
<tr>
<td>2.5</td>
<td>Reported values of SEA versus material type</td>
<td>17</td>
</tr>
<tr>
<td>3.1</td>
<td>Schematic of bladder-mold manufacturing method</td>
<td>22</td>
</tr>
<tr>
<td>3.2</td>
<td>CAD model of CAMRI at Purdue CMSC</td>
<td>23</td>
</tr>
<tr>
<td>3.3</td>
<td>Detailed schematic of top half of composite mold, all values in inches</td>
<td>25</td>
</tr>
<tr>
<td>3.4</td>
<td>Detailed schematic of bottom half of composite mold, all values in inches</td>
<td>26</td>
</tr>
<tr>
<td>3.5</td>
<td>Composite 3D printed tool with metal brace after 46 manufacturing cycles</td>
<td>28</td>
</tr>
<tr>
<td>3.6</td>
<td>Detailed schematic of top half of aluminum mold, all values in inches</td>
<td>30</td>
</tr>
<tr>
<td>3.7</td>
<td>Detailed schematic of bottom half of aluminum mold, all values in inches</td>
<td>31</td>
</tr>
<tr>
<td>3.8</td>
<td>Aluminum mold after several manufacturing cycles</td>
<td>32</td>
</tr>
<tr>
<td>3.9</td>
<td>Load-displacement curves for two test rounds of aluminum samples</td>
<td>34</td>
</tr>
<tr>
<td>3.10</td>
<td>SEA of aluminum 6061 versus t/D ratio for diamond failure mode</td>
<td>35</td>
</tr>
<tr>
<td>3.11</td>
<td>Load-displacement curves for low carbon steel samples</td>
<td>36</td>
</tr>
<tr>
<td>3.12</td>
<td>Failure morphology comparison for crush tube that displayed the splaying mode</td>
<td>38</td>
</tr>
<tr>
<td>3.13</td>
<td>Failure morphology comparison for crush tube that displayed the fragmentation mode</td>
<td>39</td>
</tr>
<tr>
<td>3.14</td>
<td>Load-displacement comparison of splaying and fragmentation failure modes</td>
<td>40</td>
</tr>
<tr>
<td>3.15</td>
<td>Stability of tubes as related to placement of axial plies</td>
<td>41</td>
</tr>
<tr>
<td>3.16</td>
<td>Example crushed tubes with unstable and stable layups</td>
<td>43</td>
</tr>
</tbody>
</table>
3.17 Experimental specific energy absorption versus percentage of axial fibers for continuous fiber tubes ............................................. 44
3.18 Comparison of tubes made with composite and aluminum molds prior to sample preparation ............................................. 45
3.19 Microscopy image of an unconsolidated PPMC sheet ..................... 47
3.20 Microscopy image of a PPMC tube made with composite mold ........ 48
3.21 Graphical representation versus experimental results of PPMC failure . . . 52
3.22 Example load-displacement curves for 4 PPMC test samples ............. 53
3.23 Average load-displacement curve of experimentally tested PPMC samples . 54
3.24 Comparison of load-displacement curves for test samples produced from the same parent tube ............................................. 56
3.25 Example orientation distributions from the 2D orientation analysis application59
3.26 Experimental SEA versus t/D ratio for PPMC tubes ....................... 60
3.27 Experimental SEA of PPMC tubes versus closed form solutions for metal crush tubes ............................................. 61
3.28 Experimental SEA versus laminate stiffness for PPMC and continuous fiber tubes ............................................. 62
3.29 Drawing of 4-rail drop tower test fixture with the top plate removed .... 64
3.30 Drop tower test fixture during test setup ............................................. 66
3.31 Load, acceleration, velocity, displacement, and energy calculations for a PPMC tube crushed at a drop height of 2.0 meters ............. 68
3.32 Sequence of crash images from crush test with h = 3.0 meters showing cloud of debris and dust forming immediately after impact ........ 69
3.33 Remains of PPMC crush tubes (in order) after drop tests with h = 2.0, 2.5, 3.0, 3.5, and 4.0 m ............................................. 70
3.34 Energy absorbed versus maximum displacement for PPMC tubes tested with the drop tower ............................................. 71
3.35 Time history of dynamic load data for PPMC samples from the same parent tube ............................................. 73
4.1 MAT 54 Behavior in fiber and matrix directions ............................. 83
4.2 MAT 58 Behavior in fiber and matrix directions ............................. 89
Figure | Page
---|---
5.1  MAT 54 versus MAT 58 material behavior with measured properties | 100
5.2  Finalized MAT 54 and MAT 58 material models | 101
5.3  MAT 54 in LS Dyna format | 102
5.4  MAT 58 in LS Dyna format | 102
5.5  Single-element model boundary conditions | 104
5.6  MAT 54 single-element stress-strain results compared to closed-form solution | 105
5.7  MAT 58 single-element stress-strain results compared to closed-form solution | 106
5.8  Four-element model with boundary and loading conditions | 107
5.9  Stress-strain curve of four element model | 108
5.10 Constant-velocity rigid wall losing contact with mesh after element deletion | 110
5.11 Load-displacement results of mesh density study showing that frequency of peaks in load is related to element size | 111
5.12 Numerical instability in model with increased mesh density | 112
5.13 1D model example with node and element numbering for hollow cylinder geometry | 115
5.14 Load-displacement curve for 1D model with constant-velocity rigid wall | 116
5.15 Load-displacement curve for 1D model with adjustable boundary condition | 117
5.16 Dimensions for 3-point bend simulation | 118
5.17 Graphical representation of LS Dyna finite element model for flexure | 119
5.18 Diagram of ply orientations applied to each integration point for a shell element | 123
5.19 Example load versus displacement curve for flexure model | 124
5.20 Stress-strain results of speed study for flexure simulations using MAT 54 | 127
5.21 The effect of modulus ratio on the moment of inertia for a [0/90]s laminate | 128
5.22 Load-displacement results of cross ply laminate in flexure versus analytical solution | 129
5.23 Stress-strain results of crossply laminate in flexure for MAT 54 and MAT 58 | 131
5.24 Microscopy image of [0/90]s plate tested in flexure | 134
5.25 Experimental flexure results versus simulation results using MAT 54 | 135
<table>
<thead>
<tr>
<th>Figure Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.26 Experimental flexure results versus simulation results with E1 adjusted to</td>
<td>136</td>
</tr>
<tr>
<td>119 GPa</td>
<td></td>
</tr>
<tr>
<td>5.27 Geometry and boundary conditions for baseline crush tube simulation</td>
<td>137</td>
</tr>
<tr>
<td>5.28 Finite element model of crush tube with bevel built into bottom two rows of</td>
<td>138</td>
</tr>
<tr>
<td>elements</td>
<td></td>
</tr>
<tr>
<td>5.29 Machined bevel for experimental samples versus simulated bevel shape within</td>
<td>139</td>
</tr>
<tr>
<td>LS Dyna</td>
<td></td>
</tr>
<tr>
<td>5.30 Comparison of load-displacement results from crush tube model that was</td>
<td>141</td>
</tr>
<tr>
<td>solved 3 separate times</td>
<td></td>
</tr>
<tr>
<td>5.31 Errant spikes seen in rigid wall reaction force during crush tube baseline</td>
<td>142</td>
</tr>
<tr>
<td>setup</td>
<td></td>
</tr>
<tr>
<td>5.32 Post-processing example of the load-displacement curve from a crush tube</td>
<td>145</td>
</tr>
<tr>
<td>model</td>
<td></td>
</tr>
<tr>
<td>5.33 Experimental and simulated load-displacement curves for axial fiber</td>
<td>149</td>
</tr>
<tr>
<td>percentages of 12.5%, 20%, and 33%</td>
<td></td>
</tr>
<tr>
<td>5.34 Experimental and simulated load-displacement curves for axial fiber</td>
<td>150</td>
</tr>
<tr>
<td>percentages of 50%, 66%, and 80%</td>
<td></td>
</tr>
<tr>
<td>5.35 Specific energy absorption of continuous fiber tube simulations versus</td>
<td>151</td>
</tr>
<tr>
<td>experimental results with the same layup</td>
<td></td>
</tr>
<tr>
<td>5.36 Numerical instability causing catastrophic failure of crush tube</td>
<td>153</td>
</tr>
<tr>
<td>5.37 Example of load displacement curve from catastrophic failure in simulation</td>
<td>154</td>
</tr>
<tr>
<td>versus experimental results</td>
<td></td>
</tr>
<tr>
<td>5.38 Load displacement curve for models with 87.5% axial fibers and different</td>
<td>155</td>
</tr>
<tr>
<td>stable time increment scale factor</td>
<td></td>
</tr>
<tr>
<td>5.39 Comparison of load displacement curves for simulations with increasing</td>
<td>157</td>
</tr>
<tr>
<td>number of integration points and 20% axial fibers</td>
<td></td>
</tr>
<tr>
<td>5.40 Comparison of load displacement curves for simulations with increasing</td>
<td>159</td>
</tr>
<tr>
<td>number of integration points and 50% axial fibers</td>
<td></td>
</tr>
<tr>
<td>5.41 Specific energy absorption of continuous fiber tubes compared to simulation</td>
<td>161</td>
</tr>
<tr>
<td>results with and without additional integration points</td>
<td></td>
</tr>
<tr>
<td>5.42 Specific energy absorption of continuous fiber tubes compared to simulation</td>
<td>162</td>
</tr>
<tr>
<td>results with reduced E1</td>
<td></td>
</tr>
<tr>
<td>6.1 Orientation of a vector using spherical coordinates</td>
<td>165</td>
</tr>
<tr>
<td>Figure</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>------</td>
</tr>
<tr>
<td>6.2</td>
<td>166</td>
</tr>
<tr>
<td>6.3</td>
<td>167</td>
</tr>
<tr>
<td>6.4</td>
<td>170</td>
</tr>
<tr>
<td>6.5</td>
<td>171</td>
</tr>
<tr>
<td>6.6</td>
<td>172</td>
</tr>
<tr>
<td>6.7</td>
<td>173</td>
</tr>
<tr>
<td>6.8</td>
<td>174</td>
</tr>
<tr>
<td>6.9</td>
<td>175</td>
</tr>
<tr>
<td>6.10</td>
<td>176</td>
</tr>
<tr>
<td>6.11</td>
<td>176</td>
</tr>
<tr>
<td>6.12</td>
<td>181</td>
</tr>
<tr>
<td>6.13</td>
<td>182</td>
</tr>
<tr>
<td>6.14</td>
<td>185</td>
</tr>
<tr>
<td>6.15</td>
<td>186</td>
</tr>
<tr>
<td>6.16</td>
<td>188</td>
</tr>
<tr>
<td>6.17</td>
<td>189</td>
</tr>
<tr>
<td>6.18</td>
<td>190</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
</tr>
<tr>
<td>6.19</td>
<td>Simulated load-displacement curves for tubes using MAT 54</td>
</tr>
<tr>
<td>6.20</td>
<td>Simulated load-displacement curves for tubes using MAT 58</td>
</tr>
<tr>
<td>6.21</td>
<td>Specific energy absorption compared to the thickness-to-diameter ratio of simulated tubes</td>
</tr>
<tr>
<td>6.22</td>
<td>SEA results for t/D simulation study versus experiment</td>
</tr>
<tr>
<td>6.23</td>
<td>Load-displacement results for MAT 54 simulations versus experiment</td>
</tr>
<tr>
<td>6.24</td>
<td>Load-displacement results for MAT 58 simulations versus experiment</td>
</tr>
<tr>
<td>6.25</td>
<td>SEA results from crush tubes using MAT 54 and MAT 58 compared to experiment</td>
</tr>
<tr>
<td>B.1</td>
<td>Schematic of bottom plate for drop tower test fixture</td>
</tr>
<tr>
<td>B.2</td>
<td>Schematic of middle plate for drop tower test fixture</td>
</tr>
<tr>
<td>B.3</td>
<td>Schematic of top plate for drop tower test fixture</td>
</tr>
</tbody>
</table>
## ABBREVIATIONS

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BMC</td>
<td>Bulk Molding Compound</td>
</tr>
<tr>
<td>CAMRI</td>
<td>Composites Additive Manufacturing Research Instrument</td>
</tr>
<tr>
<td>CDM</td>
<td>Continuum Damage Mechanics</td>
</tr>
<tr>
<td>CLPT</td>
<td>Classical Laminate Plate Theory</td>
</tr>
<tr>
<td>CMSC</td>
<td>Composites Manufacturing and Simulation Center</td>
</tr>
<tr>
<td>DMS</td>
<td>Diversified Machine Systems</td>
</tr>
<tr>
<td>DOF(s)</td>
<td>Degree(s) of Freedom</td>
</tr>
<tr>
<td>HPCC</td>
<td>High Performance Computing Cluster</td>
</tr>
<tr>
<td>MLT</td>
<td>Matzenmiller, Lubliner, Taylor</td>
</tr>
<tr>
<td>NHTSA</td>
<td>National Highway Traffic Safety Administration</td>
</tr>
<tr>
<td>OEM</td>
<td>Original Equipment Manufacturer</td>
</tr>
<tr>
<td>ORNL</td>
<td>Oak Ridge National Lab</td>
</tr>
<tr>
<td>PPMC</td>
<td>Prepreg Platelet Molding Compound</td>
</tr>
<tr>
<td>RTM</td>
<td>Resin Transfer Molding</td>
</tr>
<tr>
<td>SEA</td>
<td>Specific Energy Absorption</td>
</tr>
<tr>
<td>SCL</td>
<td>Stable Crushing Load</td>
</tr>
<tr>
<td>SMC</td>
<td>Sheet Molding Compound</td>
</tr>
<tr>
<td>TMAC</td>
<td>Test Machine for Automotive Crashworthines</td>
</tr>
<tr>
<td>UD</td>
<td>Unidirectional</td>
</tr>
<tr>
<td>UMAT</td>
<td>Abaqus User Material</td>
</tr>
<tr>
<td>US EPA</td>
<td>United States Environmental Protection Agency</td>
</tr>
<tr>
<td>VARTM</td>
<td>Vacuum Assisted Resin Transfer Molding</td>
</tr>
</tbody>
</table>
ABSTRACT

Cutting, Rebecca A. Ph.D., Purdue University, August 2019. Crash Performance of Pre-Impregnated Platelet Based Molded Composites. Major Professors: R. Byron Pipes, Johnathan E. Goodsell.

Platelets made of slit and chopped unidirectional, carbon-fiber prepreg are becoming a popular option for use as a high performance molding compound because of their high fiber volume fraction and increased ability to flow compared to continuous fiber systems. As this molding compound is newly introduced to industry, increasing amounts of research have gone into understanding how platelets flow during molding and how components perform mechanically based on the final orientation state of platelets. This work investigates the performance of prepreg platelet molding compound (PPMC) as a viable alternative to continuous fiber systems for use with geometrically complex structural members on vehicles subjected to collisions. In doing so, the crash performance, energy absorption, and failure morphology of crush tubes made with PPMC are investigated and quantified. Then, a simulation methodology is developed to obtain manufacturing-informed performance models to predict the effect of platelet orientation state on mechanical behavior of PPMC components. This methodology uses a building block approach where each block in modeling is verified against closed-form solution (when available) and validated against experimental results. Once confidence is developed in a modeling block, the complexity of the simulation is increased until a component with full platelet orientation distribution is captured. The result is PPMC component models that are capable of predicting mechanical performance in orientation regimes that are not investigated experimentally.
1. INTRODUCTION

Fiber reinforced plastics were introduced in the 1940s as a light weight, high stiffness alternative to metals. First utilized in the aerospace and maritime industries, specifically for military purposes, fiber composites garnered attention in alternative sectors because of their wide range of potential applications. The patent of the first carbon fiber in 1961, expanded the performance envelope of composite materials with an increase in stiffness to weight ratios [1]. In the automotive industry, carbon fiber composites were initially limited to luxury vehicles with low production numbers. However, a push for reduced emissions by the United States Environmental Protection Agency (EPA) and Department of Transportation’s National Highway Traffic Safety Administration (NHTSA) has given the automotive industry an incentive to investigate the use of carbon fiber composites as structural members on high production rate vehicles to increase fuel economy standards [2].

Composite materials are not new to the automotive industry, the 1953 Chevrolet Corvette C1 utilized fiberglass for the outer body in the first production vehicle to contain fiber composites [3]. However, carbon fiber components on vehicles have been limited to aesthetic and/or non-structural applications due to slow manufacturing methods, material costs, and a trade-off between mechanical properties and cycle times. An example of this is a car hood that is made out of a carbon fiber weave. While there are considerable weight savings, the necessary manufacturing method (hand layup) is not suitable for high production rate vehicles. An alternative example is an injection molded part using a short fiber system. These components can be manufactured at high volumes with complex geometries, but the lower stiffness and strength properties of the molding material system prevent the components from being effective structural members.
A challenge unique to the automotive industry is the need for lightweight structural members that can be manufactured at high volumes and perform well under impact loading conditions. These requirements limit the composite material systems available for this application. Continuous fiber composites are commonly manufactured with methods that produce excessive waste and are incompatible with short cycle times, like hand layup and VARTM [4]. Short fiber systems manufactured via injection molding or compression molding are suitable for high volume manufacturing, but their mechanical properties are not good enough to serve as structural components for a vehicle.

This work investigates the performance of a relatively new material system, prepreg platelet molding compound (PPMC), as a viable option for structural members on vehicles. The manufacturing and testing of PPMC components under compressive and flexural loading conditions, to represent vehicular crash, will be detailed, and a sound methodology for modeling this material system in finite element simulations will be introduced.

1.1 Molding Compounds

1.1.1 Bulk Molding Compound and Sheet Molding Compound

There are two primary molding compounds for composite materials in the automotive industry: bulk molding compound (BMC) and sheet molding compound (SMC). Both of these molding compounds use chopped glass fibers in combination with thermoset resin and fillers to create a “ready-to-mold” material system for compression molding.

Bulk molding compound is created by combining the raw materials in an industrial mixer and producing a dough like compound. BMC generally has 10-30% glass fibers by weight with fiber lengths ranging from 6-12 mm [1,5]. During manufacturing, a lump of BMC is placed in the center of the molding tool. Once the heated tool closes, the material flows to fill the remaining gaps in the mold. Components made of BMC
are limited in size and dependent on the ability of the material system to flow without separating.

Sheet molding compound is created when a random distribution of glass fibers is poured onto a moving line of resin and filler mixture and then compressed. This process results in a sheet of molding compound that can be distributed along an open tool surface. The moving line setup allows for a larger percentage of fibers in the mixture, with glass fiber content by weight ranging from 10-60% [6]. SMC also contains longer fibers than BMC, ranging from 25-75 mm [1]. This in combination with the higher glass fiber content results in better mechanical properties for components made of SMC versus BMC.

### 1.1.2 Prepreg Platelet Molding Compound

Prepreg platelet molding compound, or PPMC for short, is a molding compound created from unidirectional carbon fiber prepreg. The aligned, continuous fiber prepreg is ran through a machine that slits and chops the material into rectangular platelets of adjustable dimensions. PPMC can be supplied as a loose platelet material system to be poured into a charge cavity for molding, or it can be provided in a sheet form similar to SMC. The sheet form of PPMC is created by letting newly chopped platelets fall onto a moving line of nonstick film that is then heated and compressed slightly. The heat makes the platelets tacky and they will stick together under compression. Figure 1.1(a) provides an example of PPMC in sheet form, similar to SMC, while Figure 1.1(b) shows loose PPMC platelets.

The primary advantage of PPMC is that it maintains a high fiber volume fraction, between 50-60% depending upon the base prepreg, while still being applicable as a molding compound. SMC and BMC have random fiber distributions in their pre-molded material, however, PPMC has local fiber alignment corresponding to individual platelets. In addition to this, PPMC does not include extra filler or resin beyond the UD prepreg and fiber lengths can be longer than SMC. Usually, an increase
in performance means a decrease in processability [7], see Figure 1.2, but PPMC has similar processability to SMC with a noticeable increase in performance.

PPMC is not limited to a specific type of resin system, as it has been created and studied using both thermoplastics and thermosets [8, 9]. Additionally, experimental
studies on platelet aspect ratio, manufacturing conditions, and platelet orientation state [10–16] have shown that mechanical properties of PPMC components are dependent upon processing conditions.

Determining the final platelet, and thus fiber orientations of this material system is crucial for predicting performance of components made of PPMC. Nondestructive methods like CT scanning [17, 18] and modal tap tests [19] have been investigated as options for determining platelet orientation post manufacturing. CT scanning has proven effective at measuring fiber orientation, but the method is currently limited to parts less than 2” wide because of the need for a high scan resolution. Simulating the manufacturing process is another alternative to determining final fiber orientations. Favaloro and Sommer introduced methods to model the compression molding of prepreg platelet material systems in order to predict final fiber orientation distribution within parts [20–24].

1.2 Crashworthiness

Crashworthiness, crash avoidance, driver performance, and highway construction are the primary focuses of transportation safety in this day and age [25]. While car manufacturers can create additional features to keep driver’s alert and avoid crashes, new vehicle models are still required to meet government regulations to protect the passenger in the event of an impact [26]. As stated by Bois et al, “[…] in the automotive industry, crashworthiness connotes a measure of the vehicle’s structural ability to plastically deform and yet maintain a sufficient survival space for its occupants in crashes involving reasonable deceleration loads” [25].

The structural frames of modern vehicles are designed to withstand operating loads without plastic deformation while also having the ability to fail progressively and absorb energy in a crash [27]. These frames are usually made of carbonized steel or aluminum as their performance meets standards, produces repeatable results, and can be successfully modeled with computer simulations. Research completed
by automakers has concluded that carbon composites have potential as structural members on vehicles, but the slow cycle time and cost of production of these materials are preventing implementation in industry [28,29].

In addition to production roadblocks, simulation of composite materials under impact loading conditions are not at predictive levels yet. The failure of composites are more complicated than metals because of the heterogeneous and anisotropic aspects of the material system. A composite component made of PPMC may not fail in the same location and under the same loading condition consistently due to variation in manufacturing parameters and fiber orientation distribution. Additional model development is needed before automotive manufactures can fully predict crash performance of such composite materials.

1.3 Problem Statement

As previously mentioned, the auto industry has several challenges to overcome before carbon composites can be introduced as structural members in vehicles. This work aims to make progress on this front by experimentally evaluating the performance of PPMC components under loading conditions associated with impact and introducing a new modeling approach using manufacturing informed crash simulations.

Previous work regarding crush response of metal and composite tubes is discussed in Chapter 2. While a significant amount of research has investigated the progressive failure of continuous fiber tubes, little work has been completed in regards to the failure of discontinuous fiber tubes. The manufacturing method implemented and experimental results of PPMC crush tubes are presented in Chapter 3.

Current modeling practices for crash simulations are introduced in Chapter 4, including a review on common material models, contact definitions, and post processing techniques within LS Dyna. Chapter 5 covers the material characterization of the UD prepreg and its implementation into LS Dyna material models. In addition,
simple 3-point bend and crush tube models are verified with closed-form solution and validated with experiment. Additionally, a 1D finite element model was written and is discussed in order to address a common modeling issue associated with the failure and subsequent deletion of finite elements in compression. Chapter 6 goes on to introduce a method for creating manufacturing-informed performance simulations. Models of flexure samples and crush tubes are examined and validated with experimental results, and a discussion of the applicability for advanced geometries is included.
2. BACKGROUND ON EXPERIMENTAL CRUSH TUBES

Research regarding crash performance of carbon fiber composites is generally split into two categories: physical testing to evaluate material performance and crash simulation of components as part of the design product life cycle. This chapter will provide details regarding common test methodologies and experimental results of composite components tested in crush.

There are a number of structural and regulatory requirements for energy absorbing components within a vehicle experiencing a high velocity impact [30, 31]. These requirements provide a guide for testing structural components in a meaningful setting. A vehicular crash can subject components to both uniaxial and multiaxial loading depending on the angle of impact. This work investigates a limited scope of loading conditions by evaluating the performance of crush tubes in uniaxial compression and flexure samples in 3-point bend.

2.1 Variables of Interest

There are 3 primary variables of interest when testing structural components for crash response: failure morphology, load-displacement curve, and specific energy absorption.

2.1.1 Failure Morphology

Failure morphology qualitatively describes the failure mechanism of a material system. Ductile failures in compression are described with terms like local buckling, while brittle failures can be described as fragmentation.
Hull [32], Farley, and Jones [33] are well known for their thorough investigations of failure morphology in continuous fiber crush tubes. Both sets of authors chose to look at hollow tubes with circular cross sections comprised of continuous fiber layups. Their investigations produced four failure morphologies associated with the progressive failure of composite tubes. These are the splaying mode also known as the lamina bending mode, the fragmentation mode sometimes called the transverse shearing mode, the brittle fracture mode, and the local buckling mode. Figure 2.1 provides representative graphics for each of the failure modes.

![Figure 2.1. Continuous fiber failure modes presented in literature](image.png)

The splaying mode occurs where there is a high ratio of axial fibers within the laminate. This failure is dominated by delamination between plies. Cracks form
between plies, and a wedge of fiber/matrix debris sustains the delamination process. The energy absorption in these layups comes from the crack growth between plies.

The fragmentation mode is characterized by lamina bundles that break away from the circumference of the tube. This occurs when there is a high ratio of hoop fibers in the laminate. During the crushing process, cracks develop in the matrix and whole fiber bundles slough off of the tube at an angle. The remaining material fractures and forms debris as the cycle continues. In this case, the primary energy absorption mechanism is the fracturing of the matrix that results in fiber bundles detaching.

The brittle fracture mode was only described in Farley’s work and is a combination of the splaying and fragmentation modes. Cracks form within the laminate to promote delamination, but lamina bundles also break off of the layup, particularly in the center of the tube.

The local buckling mode is the last failure mode described, and it is most commonly associated with ductile materials like aramid-fiber composites. Yielding of a local section of fiber and matrix forms a buckle that collapses. As the tube continues to crush, more buckles develop and eventually stack on top of each other forming an accordion shape.

2.1.2 Load-Displacement Curve

The physical quantities measured during crush experiments are the load and the displacement of the platens on the test fixture. From these quantities the load-displacement curve can be created and provide information regarding the crush response of the component.

A typical load-displacement curve is presented in Figure 2.2. The curve is divided into five stages corresponding to common phases of loading during the crushing of composite tubes [34]. Once the test begins, the tube starts to take load, which is evident by a relatively linear slope. During this time, the trigger installed on the tube via bevel, notch, or ply drop off is crushing. The peak load, designated by a 1
on the figure, is attained when local fracture has occurred through the full thickness of the tube. A drop in load, at point 2, is observed afterwards and corresponds to the crash front forming. The size of the load drop is dependent upon the type and angle of the trigger installed on the tube. The load gradually saturates again (point 3) and eventually reaches the stable crushing load at point 4. Depending on the failure morphology active, stable crushing can have a cyclic response due to fiber bundles forming and breaking off or localized buckles forming. The splaying mode does not contain oscillations because failure propagation is dependent upon a constant load to continue delamination. The stable crush zone, point 5, is defined as the distance on the curve that maintains the stable crush load. The start and end points of this zone are defined by $S_1$ and $S_2$.

![Generic load-displacement curve of composite tube during progressive failure](image)

Figure 2.2. Generic load-displacement curve of composite tube during progressive failure

The failure morphology of a tube can often be determined by looking directly at the load-displacement curve. When the splaying failure mode is present, the load-displacement curve has a high initial peak followed by an immediate drop in load to
the stable crushing load. The initial peak can be anywhere from 2-3 times larger than that of the stable crushing load. This is because the force needed to initiate damage and incur delamination is high, but the force required to maintain delamination is much lower.

The load-displacement curve of a tube in fragmentation mode has an initial peak that is about the same force as the stable crushing load. In this case, during the stable crush zone, the force is not a constant value but oscillates around an average value. The oscillations in the force are erratic and due to individual fiber bundles taking on load, failing, and then sloughing off of the tube as the next set of fiber bundles is impacted.

The load-displacement curve of a tube in brittle fracture failure mode is similar to that of fragmentation mode. However, the stable crushing load will increase as the percentage of axial fibers increases in the layup.

Finally, the load-displacement curve of a tube experiencing local buckling will have a high initial peak followed by large predictable oscillations. The oscillations will correspond to each buckle forming. Once the tube has buckled completely, a large increase in force will appear as further crushing cannot be obtained without pulverizing the material.

### 2.1.3 Specific Energy Absorption

The specific energy absorption, or SEA, is the amount of kinetic energy absorbed per unit material during a failure event. The SEA is calculated with data corresponding to the stable crush duration of the test, and is primarily presented in units of kJ/kg.

The total work done on the system is defined as

\[
U = \int_{S_1}^{S_2} PdS
\]  

(2.1)
In this case, $U$ is the energy absorbed, $S_1$ and $S_2$ are the crush distances corresponding to the beginning and end of the stable crush zone, and $P$ is the load measured during the crushing process. From there, the specific energy absorption can be defined as the energy absorbed per unit material. Using the average stable crush load $\bar{P}$, allows for the removal of the integral and produces the following

$$SEA = \frac{U}{m} = \frac{P(S_2 - S_1)}{V\rho} = \frac{P(S_2 - S_1)}{AL\rho}$$ (2.2)

$V$ is the volume that breaks down to area, $A$, multiplied by length, $L$, of the cross section, and $\rho$ is the density of the material.

Assuming the specific energy absorption is independent of the current location in the stable crush zone, the SEA can be further reduced to

$$SEA = \frac{\bar{P}}{A\rho}$$ (2.3)

If the density of the material is not specifically known, and it is difficult to consistently measure the cross sectional area, the weight and height of the sample can be used to form an alternative equation for SEA seen below. This form of the equation was used throughout this work.

$$SEA = \frac{\bar{P}h}{W}$$ (2.4)

### 2.2 Metal Tube Studies

Metal tubes as energy absorbers in the automotive industry have been studied since the late 1970s, with S. R. Reid at University of Manchester and W. Johnson at Cambridge being two of the most prolific writers on the topic. These men studied different loading conditions and uses for metal tubes as energy absorbers in conjunction with vehicles. Reid focused primarily on lateral crushing of metal tubes and the use of these tubes as a deformable barrier in the event of a car crash [35, 36]. However,
Johnson investigated the axial collapse of metal tubes, and worked on quantitatively describing the geometry of collapses under compression [37].

Metals do not inherently share failure modes with composite circular tubes as they are more ductile and absorb energy through plastic deformation. Andrews et al. described 7 different collapse modes for metal circular tubes [38]. Of these, the concertina and diamond failure modes were the most distinct and also studied by Reid and Johnson [37, 39]. Figure 2.3 shows examples of these failure morphologies with aluminum and steel samples.

![Steel tube in concertina failure](image1) ![Aluminum tube in diamond failure](image2)

Figure 2.3. Example concertina and diamond collapse modes in metal tubes

The concertina collapse mode is described as a symmetric folding mode where individual rings of the tube bulge out and buckle. In this case, energy is absorbed via bending and stretching in the plasticity region of the material. Alexander first described the mean crushing load of a tube in concertina deformation as

\[ P_{av} = 6.08Yt(Dt)^{1/2} \]  \hspace{1cm} (2.5)

where \( D \) is the mean tube diameter, \( t \) is the tube thickness and \( Y \) is the yield strength [40]. From there, theoretically, the SEA of a metal tube in the concertina mode can be calculated with
\[ SEA = \frac{6.08Yt(Dt)^{1/2}}{\pi Dt\rho} = \frac{1.94Y}{\rho} \left( \frac{t}{D} \right)^{1/2} \] (2.6)

where \( \rho \) is the density of the material.

The diamond failure mode is similar to the concertina mode, but during the folding of the tube a number of lobes form. The cross section of the tube beneath the folding is reduced dependent upon the number of lobes. Energy is absorbed via the bending and shearing associated with the layers forming [41]. Jones and Abramowicz suggested a modification to Alexander’s equation for the mean crushing load associated with diamond failure modes [42,43].

\[ P_{av} = 18.15Yt^2(D/t)^{1/3} \] (2.7)

This leads the specific energy absorption of a metal tube in the diamond mode to be calculated using

\[ SEA = \frac{18.15Yt^2(D/t)^{1/3}}{\pi Dt\rho} = \frac{5.78Y}{\rho} \left( \frac{t}{D} \right)^{2/3} \] (2.8)

The specific energy absorption of both of these failure modes is dependent upon the thickness-to-diameter ratio, \( t/D \).

### 2.3 Continuous Fiber Tube Studies

The specific energy absorption for composites is not a material property; instead it is dependent upon a multitude of parameters from manufacturing process to testing conditions. Figure 2.4 shows a sample of the parameters that affect the specific energy absorption [34].

#### 2.3.1 Effect of Material System

The parameters within the material system category have the largest effect on SEA. Farley compared typical fiber systems used in composite materials and found
that generally graphite/epoxy composites absorbed more energy than glass/epoxy and Kevlar/epoxy composites for the same layup [44]. He also discovered that composite tubes have the potential to absorb more energy than metal tubes depending on the layup. Lukaszewicz from BMW provided an overview of reported SEA numbers from a variety of fiber and matrix combinations that had been tested in crush [29]. Carbon fiber/PEEK combinations had the highest SEA values followed by carbon fiber/epoxy, and glass fiber combinations. A further reporting of comparative SEA numbers by Hancox [45], Lu [46], and Ramakrishna [34] allowed for the creation of Figure 2.5, which provides a visual for SEA versus material type.

A research group at Oak Ridge National Labs determined that as fiber volume fraction decreased, the SEA increased [47]. They cited the reduction in interlaminar strength because of the increased matrix volume as a potential cause. Hamada et al. also noted that a high interlaminar toughness value would result in an increase in energy absorption, and thus, PEEK (a thermoplastic) was an ideal matrix choice for progressive crushing [48].

Schultz, Hyer, and Fuchs found that tubes made of large, low-cost tows did not absorb as much energy as tubes made from more expensive, low count tows. They
Figure 2.5. Reported values of SEA versus material type

suggested this might be due to the coarse microstructure associated with larger tows [49].

A study on the effect of fiber orientation using balanced layups revealed that energy absorption increased as the angle, $\theta$, approached $\pm 15$ deg and then decreased thereafter [50]. However, this trend does not necessarily hold for layups other than $\pm \theta$. Farley studied the effect of fiber orientation with graphite/epoxy, Kevlar/epoxy, and glass/epoxy tubes all with layups of $[0/\pm \theta]$ [44]. The layup of highest SEA was $[0/\pm 15]$ for graphite, $[0/\pm 90]$ for Kevlar, and $[0/\pm 75]$ for glass.

2.3.2 Effect of Processing Conditions

The effect of processing conditions on energy absorption has been studied to a lesser extent. Rapid cooling of a thermoplastic tube post cure has shown higher SEA results compared to gradual or slow cooling [51]. However, an alternative study of thermoset tubes manufactured via resin transfer molding (RTM) found that post curing the samples in an oven increased the SEA [52]. Generally, processing conditions
that affect the fracture toughness will influence the SEA. Benedetto et al. investigated how cure temperatures for thermoplastics affect the thermo-oxidative degradation of the matrix. They determined degradation in the matrix negatively affected the energy absorption in crash [53].

Bladder molding is the most common manufacturing method used for creation of composite crush tubes. However, Thornton [54] completed a study with Ford Motor Company that looked at the crush performance of pultruded tubes, and Turner looked at performance of resin transfer molded (RTM) tubes [52] as well as tubes made from the vacuum assisted resin transfer molded (VARTM) process [55].

2.3.3 Effect of Geometry

Studies on the cross sectional shape of continuous fiber tubes have shown that circular cross sections absorb the most energy followed by square and then rectangular shapes [56,57]. Square tubes commonly fail in the splaying mode because of the stress concentrations at the corners with small radii of curvature. Sine webs provide an alternative, self-supported geometry for crush that is easier to manufacture and does not have the hoop constraints native to tubes [58, 59]. While the sine webs display the main failure modes, the specific energy absorption does not directly compare with compression tubes made of the same material.

Sigalas, Kumosa, and Hull discovered that the angle of the trigger influenced the initial slope of the load-displacement diagram and affected the initial failure mechanism [60]. Hanagud et al. compared trigger methods between an end chamfer, ply drop off, and adding a notch to the specimen, determining that longer triggers actually reduced the energy absorption in sine webs [58].

Mamalis et al. completed studies finding that thinner carbon reinforced plastic tubes are more likely to buckle than progressively fail, and the peak load is dependent upon the thickness of the tubes [61, 62]. Farley and Jones presented a procedure capable of determining the effect geometry changes have on the stable crush load [63].
2.3.4 Effect of Testing Parameters

It is unclear whether the speed of the crush tests affect the SEA. Generally, experimentalists perform crush tests at quasi-static speeds at a range of 1-15 mm/min and then again at dynamic speeds of 5-7 m/s \[32, 44\]. However, authors have yet to come to a consensus on whether specific energy absorption is a function of crush speed \[64\]. Bannerman and Kindervater found an increase in SEA with crush speed for epoxy tubes \[65\]. Farley's work on graphite/epoxy, Kevlar/epoxy and glass/epoxy suggested that crush speed did not directly affect energy absorption \[44\], but his later work found the SEA of graphite/epoxy and Kevlar/epoxy were in fact a function of crush speed \[66\]. He came to the conclusion that if the constituent that controls the failure process is dependent upon strain rate, then the energy absorption would also be dependent upon crushing speed. Hamada and Ramakrishna had a similar notion and argued that a change in failure mode due to the change in speed affected specific energy absorption \[67\]. Mamalis et al. agreed with Farley and stated that the SEA will be dependent upon the crush speed if the failure mechanism is also a function of crush speed \[68\].

A research group at Oak Ridge National Labs (ORNL) created a test fixture that measures crashworthiness for a material system by forcing bars of material into the splaying failure mode \[69–71\]. However, results are dependent upon the constraints that force the bar into a specific radius of curvature. In addition, composite tubes do not always fail in the splaying mode. Forcing a bar of material into a specific failure morphology via constraints may not actually represent how the material system would perform in situ.

2.4 Discontinuous Fiber Tube Studies

There is a gap in literature with regards to crash performance of discontinuous carbon fiber material systems. The ORNL group performed tests on PPMC materials, but their modified fixture created results that can not be directly compared to test
results from tubes [72, 73]. In their studies, they found that fiber length was a critical parameter that affected SEA, but they could not assign a specific correlation between increase in length and SEA because the relationship was also dependent on tow size. Turner et al. completed the most comprehensive study of discontinuous fiber tubes found in literature [55]. This team used dry fiber preforms that were infused with resin via VARTM.
3. EXPERIMENTAL WORK ON CRUSH TUBES

3.1 Manufacturing the Tubes

The material supplied for this project was a thermoset PPMC in sheet form with VORAFUSE M6400\textsuperscript{TM} resin. The processing conditions provided by the manufacturer suggested a 12-15 min cure cycle at 130°C, or 3 min at 150°C, with 1600 psi. While this pressure is possible when compression molding with a commercial grade press, manufacturing crush tubes with a bladder molding process at this pressure level is impractical.

A modified bladder molding manufacturing process, similar to one presented by Whitney et al. [74] was created for manufacturing the PPMC tubes. The design includes a closed-end perforated steel mandrel with a NPT threaded pneumatic male fitting on one end, and a two part closed mold.

During the manufacturing process, a thin layer of Crisco vegetable shortening is applied to the mandrel, and a tight-fitting sleeve of high temperature silicone rubber is slid over the mandrel and secured using square cross section O-rings. PPMC sheets, cut into 10” squares, are made tacky with a portable heat gun and rolled onto the silicone tube/mandrel setup.

When manufacturing PPMC, the platelets drop onto a moving conveyor belt. The moving direction of the conveyor belt is noticeable in the PPMC sheets as the platelets have some alignment associated with the line direction. In addition, the sheet is thickest in the center of the conveyor line direction and tapers off on the edges. To counteract the effect the thickness distribution might have along the circumference of the tube, PPMC sheets are rolled onto the mandrel with the thinner edges acting as the longitudinal axis of the tubes. The thinner edges of the sheet overlap each other.
when rolled onto the mandrel, evening out the thickness distribution and preventing excessively thin ends on the tubes.

While the mandrel is being wrapped with material, the mold and the platens on the press are pre-heated to 180°C. Once the mold is removed from the oven, the wrapped mandrel is encased, and the entire assembly is placed on the press and loaded to 2 tons. A compressed air line is connected to the nozzle on the mandrel, and the bladder is inflated to an internal pressure of 100 psi. These loading conditions are held for 20 minutes. The bladder is then deflated and the mold is unloaded. The mandrel setup is immediately removed from the tool and allowed to cool independently. Figure 3.1 provides a diagram of the bladder mold and tool setup.

![Diagram of bladder mold and tool setup](image)

Figure 3.1. Schematic of bladder-mold manufacturing method

### 3.1.1 3D Printed Composite Tool

Molding tools made out of carbon fiber materials offer unique advantages to their metal counterparts. While the weight reduction can be significant, especially for larger tools, an additional benefit of having a tool made of composite materials is the low
coefficient of thermal expansion. When processing composite materials with a metal tool, the expansion of the metal during the cure cycle affects the final dimensions of the part. In addition, demolding can be particularly cumbersome because the tool shrinks during cooling while the composite part does not. This is not a problem with a composite printed tool. A molding tool manufactured with composite materials can be created with the dimensions of the final part shape, and demolding is not a strenuous activity. In addition, 3D printing a composite tool allows for a faster manufacturing time and cheaper tool because the amount of machining is reduced.

The Purdue Composites Manufacturing and Simulation Center, has an in-house composite 3D printer known as the CAMRI. This 3D printer has a single screw extruder with a max temperature of 800°F and an output of 5-10 lb/hr on a stationary frame. A 3-axis motion table with a max speed of 470 in/min allows for a printing size up to 20” cubed. Figure 3.2 provides a digital representation of the CAMRI.

Figure 3.2. CAD model of CAMRI at Purdue CMSC
A design for a 3D printed tool was created to manufacture 10” long hollow tubes with a constant 2.125” outer diameter. Figures 3.3 and 3.4 provide schematics of the composite mold design. The tool had alignment ledges along 3 sides, and a cutout for the nozzle located on the mandrel. The raw material used for printing the tool was Techmer Electrafil® XTI 3DP. This is a polyphenylene sulfide (PPS) with 50% by weight carbon fiber. In order to ensure the mold maintained a cylindrical shape when heated, the layers had to be printed in line with the longitudinal axis of the cylindrical hollow in the mold. An additional height constraint on the printer at the time required each half of the composite mold to be printed in 2 separate parts approximately 6.875” long.
Figure 3.3. Detailed schematic of top half of composite mold, all values in inches
Figure 3.4. Detailed schematic of bottom half of composite mold, all values in inches.
Once printed, the sides of each mold section were sanded down to create a flat surface for bonding. A room cure epoxy adhesive, Loctite EA E-40HT, was used to join the mold sections. The mold was printed larger than the final design, and the excess material was milled out of the printed part with a 5-axis DMS CNC router. Any holes or voids exposed by the milling process were filled with an aluminum paste and sanded down. A rudimentary metal brace was added to the exterior of the mold to ensure the adhesive did not expand and break the mold apart during manufacturing.

Prior to use in manufacturing, the mold was cleaned with Chemlease® Mold Cleaner EZ and sealed with Chemlease® 15 Sealer EZ. Finally, multiple coats of a semi-permanent release agent, Chemlease® R&B EZ, were applied to the mold to ensure resin did not adhere to the mold in between manufacturing cycles.

The 3D printed mold was used extensively, as a total of 46 continuous fiber and PPMC tubes were manufactured with this setup. Overtime, deformations were noticed in sections of the mold that had aluminum paste, usually corresponding to sharp corners. This is thought to be from high loading when the material was pinched along the seams of the mold. The aluminum paste that came into contact with the material eventually discolored from a gray to a brown. Even with a semi-permanent mold release, small amounts of resin adhered to the surface of the tool after a several cycles creating an unsatisfactory surface texture on some of the tubes manufactured later. In addition, during one of the first cure cycles, resin seeped into the bonded surface of the tool sections. This resin expanded with the increase in heat, and created a noticeable seam in the tool surfaces. The epoxy adhesive and the metal brace kept the tool intact, but tubes manufactured after this had noticeable flash located in the area. Figure 3.5 shows the mold and metal brace after being decommissioned.

In this design, the diameter of the hollow cavity in the mold was calculated to perfectly fit the steel mandrel, silicone tube, PPMC material, and O-rings. The goal was to provide a tight fit on the O-rings that would prevent air leaking out during inflation of the bladder. However, this tight fit led to pinching of PPMC on the cusp
Figure 3.5. Composite 3D printed tool with metal brace after 46 manufacturing cycles

of the cavity. This was more common in thicker tubes, e.g., tubes made out of PPMC or continuous fiber layups with 6 or more plies.

The straight cylinder design in combination with the O-rings did not provide a consistent seal. During bladder inflation some amount of air leakage was detected on the majority of tubes during manufacturing. With that being said, there were different levels of air leakage. Notes on the sound and qualitative amount of air leaking were recorded. The target air pressure was 100 psi for the bladder. Over 60% of the manufactured tubes reached the target air pressure, and over 90% maintained at least 80 psi during the cure cycle.
3.1.2 Aluminum Tool

A second iteration mold was designed to refine the manufacturing process and validate the results from samples manufactured with the composite mold. The mold was manufactured out of tooling aluminum, and it accommodates a constant 2.25” outer diameter for a 10” long tube. Reduced diameter shelves were added to the ends of the cavity to provide a better seal for the O-rings, and alignment ledges were added to all sides to ensure a better tool fit. The design can be seen in Figures 3.6 and 3.7. The mold was milled on a 3 axis Haas VF-2 CNC.

The same chemical preparation process for the composite mold was also completed for the aluminum mold, and the finished product can be seen in Figure 3.8. Although fewer tubes have been manufactured with the aluminum mold, 100% of the samples manufactured have achieved 100 psi with no air leaking. In addition, the larger outer diameter has eliminated pinching of PPMC material.
Figure 3.6. Detailed schematic of top half of aluminum mold, all values in inches.
Figure 3.7. Detailed schematic of bottom half of aluminum mold, all values in inches
3.1.3 Sample Preparation

Each manufactured tube, regardless of implemented method, was approximately 10” long. These tubes were cut with a tile saw to create 2 separate crush samples that were 4” long. The excess material from the ends of the tubes were kept and sectioned for microscopy studies. Any flash or pinched material was removed from the tube using a Dremel. A 45° bevel was milled onto one end of the tube to initiate progressive failure during testing. The 45° angle is not exact because there were instances of platelets breaking away or fibers fraying while being milled, however the lathe was set up to make a 45° cut.
3.2 Quasi-Static Test Method

The testing was completed on two MTS load frames with 55 kip and 22 kip load cells. Flat platens were loaded into the cross-heads to provide a planar crushing surface. The beveled edge of each sample was oriented upwards towards the stationary platen.

The crush rate was set to 240 mm/min or 4 mm/sec and was chosen to emulate quasi-static testing performed by D. Hull [32]. The cross-heads were directed to crush a distance of 40 mm. This ensured stable crushing could be achieved but stopped the test before the tube filled with compacted debris preventing further crushing.

A 2-camera setup was utilized to video record the crushing process at a rate of 20 frames per second. Load and displacement data were recorded from the MTS machine at a rate of 200 Hz.

3.3 Results

3.3.1 Metal Study

Seven crush tubes made of aluminum 6061 and 3 crush tubes made of low-carbon steel were tested and compared to results found in literature.

Two sets of beveled aluminum samples were crushed using the MTS machine and test conditions described previously. The first round of testing had 2 samples with an average inner diameter of 47.94 mm, average thickness of 1.2 mm, and a $t/D$ ratio of 0.025. These samples displayed the diamond failure mode. The average specific energy absorption was 43.6 kJ/kg, and the specific energy absorption calculated from Eq. 2.8 was 42.7 kJ/kg. This indicates good correlation with closed-form solution.

The second set of aluminum test samples had an average inner diameter of 46.5 mm, average thickness of 2.05 mm, and a $t/D$ ratio of 0.044. The increase in $t/D$ suggests an increase in SEA for the same failure mode. All 5 of these aluminum tubes showed the diamond failure mode and had an average SEA of 75.8 kJ/kg. Equation
2.8 under predicted the specific energy absorption and produced an average value of 61.86 kJ/kg. The second set of samples had close to identical load displacement curves, displaying consistency of failure in aluminum. Figure 3.9 provides the load displacement curves for all 7 aluminum samples.

Figure 3.9. Load-displacement curves for two test rounds of aluminum samples

Mamalis gathered SEA values for a variety of t/D ratios and material systems that were reported in literature from the late 1970s to the early 1990s [75]. Figure 3.10 plots the closed form solution of SEA vs t/D for AL 6061 from Equation 2.8 versus the experimental results obtained during this study and the values reported by Mamalis. From the figure, it is evident that the closed form solution does an acceptable job of providing an estimate of SEA for a given t/D ratio for aluminum 6061.
Figure 3.10. SEA of aluminum 6061 versus t/D ratio for diamond failure mode

Three beveled, low carbon steel crush tubes with an average internal diameter of 48.1 mm, average thickness of 1.34 mm, and t/D of 0.028 were also tested to failure. These samples displayed the concertina failure mode, and produced an average SEA of 16.58 kJ/kg. The closed form solution for SEA of tubes in concertina mode, provided by Equation 2.6, slightly under predicted this with an estimate of 12.87 kJ/kg. The load displacement curves of the steel tubes showed consistency among samples and can be seen in Figure 3.11.

3.3.2 Continuous Fiber Tests

A continuous fiber tube study was completed with the prepreg that serves as the base material for the PPMC investigated in this project. The SEA values obtained in this study allow for a direct comparison of the carbon fiber VORAFUSE material
Figure 3.11. Load-displacement curves for low carbon steel samples

to other carbon/epoxy systems reported in literature. PPMC performance can then be directly compared to that of continuous fiber.

**Confirming Failure Morphology**

Two continuous fiber tubes were manufactured in order to confirm that the splaying and fragmentation failure modes presented in literature occurred within the VO-RAFUSE prepreg material system. One tube was made with 100% axial fibers to capture the splaying mode, and the second tube was made with 87.5% hoop fibers to capture the fragmentation mode. The samples produced from each tube were crushed quasi-statically in lab and photographed. The best representative sample from each layup was potted for microscopy.
The literature versus experimental characterization for the two tubes can be seen in Figures 3.12 and 3.13. The axial fiber samples were dominated by delamination between plies. Individual fronds, not all of equal size, separated from the circumference of the tube and were pushed back by the compressing platens. Because there was no hoop support on the exterior of the tube, the individual fronds on the exterior were not forced into a small radius of curvature that caused permanent deformation. Instead, once the load was removed, the fronds sprang back into almost vertical position. The fronds on the interior of the tube where compressed and started overlapping each other forming a mat-like structure on the interior of the hollow tube. The stereoscope image showing a cross sectional view of one side of the tube revealed multiple delaminations between layers as well as debris wedges associated with the delaminations. It is also evident that the rate of delamination between layers varied, with the most external and internal layers delaminating the furthest compared to central layers.

The hoop fiber samples were dominated by fragmentation failure and produced significantly more debris than the axial fiber samples. During failure, rings of fiber bundles fractured and sloughed off of the main tube at various angles. This created a bird nest shape with older rings getting pushed further down the tube during compression. The top view reveals the pulverized material grouped together increasing the cross sectional area of the tube. There were axial fibers present on the internal side of the tube, and these are seen sticking out of the top of the tube from the side view. The axial layers did not bend and break but delaminated and detached from the interior surface. The cross sectional view shows the fiber bundles broken off from the tube. The angle of the bundles broken off of the tube indicate a shear failure in the matrix, which is consistent with previously cited research. Similar angles of failure were seen when testing 90° plies in compression using the Modified ASTM D695 fixture for material characterization.

The load-displacement curves for the axial and hoop dominated layups also followed trends seen in literature. Figure 3.14 provides a load displacement curve for a
Figure 3.12. Failure morphology comparison for crush tube that displayed the splaying mode
Figure 3.13. Failure morphology comparison for crush tube that displayed the fragmentation mode.
representative sample of each of the tested layups. The axial fiber sample produced a high initial peak around 55 kN and proceeded to have a massive drop off in load with a constant stable crushing load around 30 kN. The hoop fiber sample had a peak load at 25 kN, and this was the same value as the stable crushing load. The load was not constant and oscillated in an erratic manner as fiber bundles broke off of the tube.

![Load-displacement comparison of splaying and fragmentation failure modes](image)

**Figure 3.14.** Load-displacement comparison of splaying and fragmentation failure modes

The experimental results for the axial fiber and hoop fiber tubes are consistent with the splaying and fragmentation modes presented by Hull [32] and Farley’s work [33] in the early 1990s. This indicates the manufacturing method employed is sound enough to produce consistent results with literature, and the VORAFUSE prepreg is comparable to other carbon fiber epoxy material systems tested.
Axial Fiber Percentage Study

A series of continuous fiber tubes were manufactured with varying percentages of axial and hoop fibers to recreate a similar study completed by Hull in 1991 [32]. Samples were produced with laminates containing only 0° and 90° plies and were manufactured with the composite mold. Twenty-six continuous fiber tubes were manufactured, each producing 2 test samples, with 8 different percentages of axial fibers: 12.5%, 20%, 33%, 50%, 66%, 80%, 87.5%, and 100%.

It was impossible to create balanced, symmetric layups for each of the axial-to-hoop ratios tested because the composite mold could only accommodate tubes up to 8 plies thick. Hull found that tubes with hoop fibers on the external surface were more likely to fail in a stable manner and absorb more energy than tubes with axial fibers on the external surface [32]. To confirm Hull’s findings, 2 laminates (one with the hoop fibers on the external surface and one with the axial fibers on the external surface) were used for each of the 8 tested axial fiber percentages. A visual of the axial fiber placement can be seen in Figure 3.15.

![Axial fibers external vs. Hoop fibers external](Figure 3.15. Stability of tubes as related to placement of axial plies)
Table 3.1 lists the number of tubes and the unique laminates created for each of the axial percentages. In this notation, the first angle is associated with the ply on the internal surface of the tube, and the last angle is the ply on the external surface of the tube.

Table 3.1. Laminates and number of tubes manufactured for continuous fiber study

<table>
<thead>
<tr>
<th>Percentage Axial Fibers</th>
<th>Layup</th>
<th>Number of Test Samples</th>
</tr>
</thead>
<tbody>
<tr>
<td>12.5%</td>
<td>[0/90\text{\textdegree}] &amp; 4</td>
<td></td>
</tr>
<tr>
<td>12.5%</td>
<td>[90\text{\textdegree}/0]  &amp; 4</td>
<td></td>
</tr>
<tr>
<td>20%</td>
<td>[0/90\text{\textdegree}] &amp; 4</td>
<td></td>
</tr>
<tr>
<td>20%</td>
<td>[90\text{\textdegree}/0]  &amp; 4</td>
<td></td>
</tr>
<tr>
<td>33%</td>
<td>[0/90\text{\textdegree}] &amp; 4</td>
<td></td>
</tr>
<tr>
<td>33%</td>
<td>[90\text{\textdegree}/0]  &amp; 4</td>
<td></td>
</tr>
<tr>
<td>50%</td>
<td>[90\text{\textdegree}/0]  &amp; 4</td>
<td></td>
</tr>
<tr>
<td>50%</td>
<td>[90\text{\textdegree}/0]  &amp; 2</td>
<td></td>
</tr>
<tr>
<td>66%</td>
<td>[0\text{\textdegree}/90\text{\textdegree}] &amp; 4</td>
<td></td>
</tr>
<tr>
<td>66%</td>
<td>[90\text{\textdegree}/0\text{\textdegree}] &amp; 4</td>
<td></td>
</tr>
<tr>
<td>80%</td>
<td>[0\text{\textdegree}/90\text{\textdegree}] &amp; 4</td>
<td></td>
</tr>
<tr>
<td>80%</td>
<td>[90\text{\textdegree}/0\text{\textdegree}] &amp; 4</td>
<td></td>
</tr>
<tr>
<td>87.5%</td>
<td>[0\text{\textdegree}/90\text{\textdegree}] &amp; 4</td>
<td></td>
</tr>
<tr>
<td>87.5%</td>
<td>[90/0\text{\textdegree}] &amp; 2</td>
<td></td>
</tr>
<tr>
<td>100%</td>
<td>[0\text{\textdegree}] &amp; 4</td>
<td></td>
</tr>
</tbody>
</table>

Out of the 52 test samples, 9 failed catastrophically due to unstable layups and/or issues with tube preparation. The SEA values were calculated for the samples that displayed progressive failure, and the load-displacement curves and failure morphology were recorded.

It was immediately observed that Hull’s findings regarding stability of tube failure based on location of axial fibers were accurate. Axial fiber plies generally fail in
delamination, and when these plies have no external hoop support the delamination process progresses faster. Delamination still occurred with 0° plies on the interior of tubes, but the presence of 90° plies force the axial fibers into a smaller radius of curvature when splaying. An example of this is seen in Figure 3.16. The unstable layup on the left had sections of axial fibers completely break off and hit the blast shield during testing. The stable layup on the right shows the permanent deformation of the axial fibers as they were forced into a smaller radius of curvature during the test. This was caused by the presence of the hoop layers on the external surface of the tube.

![Image](image1.png)  ![Image](image2.png)

Layup: [90/0]  Layup: [0/90]

Figure 3.16. Example crushed tubes with unstable and stable layups

Plotting the specific energy absorption versus the percentage of axial fibers in the tube reveals that more energy is absorbed with an increasing amount of axial fibers in a given layup, see Figure 3.17. In fact, stable layups outperformed their unstable counterparts in every layup tested. While the presence of more axial fibers increases
SEA, having a single layer of hoop fibers providing a circumferential constraint will increase the energy absorption further.

![Graph showing experimental specific energy absorption versus percentage of axial fibers for continuous fiber tubes.]

Figure 3.17. Experimental specific energy absorption versus percentage of axial fibers for continuous fiber tubes

### 3.3.3 PPMC Tests

Twenty-three PPMC tubes, producing 45 test samples, were manufactured between the two molds; sixteen of the tubes were manufactured with the composite mold and 7 with the aluminum mold.

The exterior of all tubes were smooth and conformed to the tooling surface regardless of the mold used. However, the interior of PPMC tubes had significant variation in surface texture and thickness along the circumference of the tube. The wavy surface is caused by overlapping platelets in the initial PPMC sheets. The thickness
variation of the tubes is likely due to similar variation in the PPMC sheets. The pressure used for manufacturing the tubes is not enough to ensure an even thickness distribution.

In fact, the pressure used in manufacturing is not enough to cause platelets to flow. This was seen during manufacturing with either mold. However, the aluminum mold produced larger amounts of flash than the composite mold, and it reduced the amount of pinched material along the seams of the mold. In addition, tubes manufactured with the aluminum mold displayed resin rich exterior surfaces, indicating the resin flowed more to the surface than with the composite tool. This can all be seen in the unfinished tubes displayed in Figure 3.18.

![Composite Mold](CompositeMold.png) ![Aluminum Mold](AluminumMold.png)

Figure 3.18. Comparison of tubes made with composite and aluminum molds prior to sample preparation

The thickness of each manufactured tube was measured in 12 locations along the top and bottom edges. The average thicknesses for PPMC tubes manufactured using the composite mold and the aluminum mold were 2.00 mm and 1.93 mm respectively.
The average standard deviation of these thickness measurements were 0.23 mm and 0.1 mm. Table 3.2 lists the average thickness, inner diameter, and volume for samples made from both molds as well as their standard deviations. The large thickness variations along the circumference of a single tube encouraged the use of the weight and height of the samples to calculate SEA (Eq.2.4) instead of estimated area calculations for Eq. 2.3. The calculated volumes of tubes made with each mold were within 0.2 g/cc of each other. However, it is important to remember that these volume values used the average thickness and effectively smears the thickness variation along the circumference of the tube. In addition, the calculated volumes were not validated in the lab and therefore remain estimates.

Table 3.2. Geometry measurements of PPMC tubes

<table>
<thead>
<tr>
<th></th>
<th>Composite Mold</th>
<th>Aluminum Mold</th>
</tr>
</thead>
<tbody>
<tr>
<td>Average thickness [mm]</td>
<td>2.00</td>
<td>1.93</td>
</tr>
<tr>
<td>Std. Dev. [mm]</td>
<td>0.23</td>
<td>0.1</td>
</tr>
<tr>
<td>Average inner diameter [mm]</td>
<td>51.01</td>
<td>53.43</td>
</tr>
<tr>
<td>Std. Dev. [mm]</td>
<td>0.84</td>
<td>0.37</td>
</tr>
<tr>
<td>Average Volume [cm$^3$]</td>
<td>32.2</td>
<td>32.4</td>
</tr>
<tr>
<td>Std. Dev. [cm$^3$]</td>
<td>3.65</td>
<td>1.84</td>
</tr>
</tbody>
</table>

Microscopy

Detailed microscopy images of unconsolidated PPMC mat and PPMC tubes made with both molds were captured and studied. Prior to consolidation (see Figure 3.19), there is a significant amount of empty space between platelet groupings. The mounting epoxy filled the voids between platelets, and the gaps of air are visible without potting the sample and looking at it under magnification. The microscopy image can be misleading because it only shows the unconsolidated platelets on a single cross sectional plane. There are more platelets through the depth of the sample in different
orientation states that remain unseen. The platelets have primarily rectangular cross sections, but this can change during molding (see Figure 3.20). While most platelets retain their rectangular cross section, deformation of platelet-ends into triangular shapes is seen in multiple locations. This is due to the material being compressed and filling the space between neighboring platelets. Additional resin is not added during the bladder molding process, so the platelets and their resin must fill all of the voids in the part. The gaps between the platelets are diminished during molding, and resin rich pockets develop at the end of and in between platelets. There are variations in platelet packing seen in microscopy images, which are likely depend on the initial state of the PPMC sheet. Pre-consolidation is a possible route for improving platelet packing and void content for tube manufacturing with such a material. There was not a significant visual difference between microscopy images of tubes made from the composite mold versus the aluminum mold, although tubes made with the aluminum mold had a more resin rich exterior surface.

Figure 3.19. Microscopy image of an unconsolidated PPMC sheet
In order to understand the material consolidation with this manufacturing method, measurements of platelet thickness and number of platelets through the thickness were recorded for multiple samples of unconsolidated PPMC mat, and tubes made from both molds (see Table 3.3). There are a finite number of platelets associated with each tube because additional material is not added during the manufacturing process. The number of platelets through the thickness is not used to measure level of consolidation, but provide an indication of the platelet density in the PPMC sheet prior to manufacturing. As platelets are compressed they become thinner and wider, therefore measuring the average platelet thickness provides information on the quality of consolidation with these tubes. One hundred platelet thickness measurements were made across 4 unconsolidated PPMC samples, and 160 measurements were made for a single sample tube manufactured with the composite mold and another with the aluminum mold.

A PPMC plate pressed with 80% charge coverage at the manufacturer recommended 2500 psi has an average platelet thickness of 0.09 mm. This is a 60% reduction in thickness when compared to the 0.223 mm average thickness of unconsolidated
Table 3.3. Platelet measurements from microscopy

<table>
<thead>
<tr>
<th></th>
<th>PPMC Sheet</th>
<th>Composite Mold</th>
<th>Aluminum Mold</th>
</tr>
</thead>
<tbody>
<tr>
<td>Average platelet thickness [mm]</td>
<td>0.223</td>
<td>0.162</td>
<td>0.147</td>
</tr>
<tr>
<td>Std. Dev. [mm]</td>
<td>0.037</td>
<td>0.039</td>
<td>0.034</td>
</tr>
<tr>
<td>Average # of platelets through thickness</td>
<td>7.73</td>
<td>8.94</td>
<td>9.6</td>
</tr>
<tr>
<td>Std. Dev.</td>
<td>3.3</td>
<td>3.77</td>
<td>3.83</td>
</tr>
</tbody>
</table>

platelets. The bladder molding process definitely thins the platelets but not to the extent seen when pressing a plate. The differences in pressure provide an obvious explanation for the changes in average platelet thickness between the manufacturing methods. However, the differences in charge coverage between the manufacturing methods may also play a role. The tubes were manufactured with full charge coverage meaning there is no extra room for the platelets and resin to flow. Meanwhile, the flat plate was intentionally manufactured without full charge coverage so the platelets would flow and compress, resulting in thinner platelets. A better comparison would be to look at the average platelet thickness from a plate pressed with full charge coverage.

The composite and aluminum molds split the difference between the pressed level of consolidation and unconsolidated cases. The composite mold averaged a platelet thickness of 0.162 mm, and the aluminum mold averaged 0.147 mm. The platelet thickness results in conjunction with the number of platelets through the thickness indicate the aluminum tube consolidated the material slightly better than the composite tool. The lack of noticeable voids in microscopy results combined with the change in platelet thickness pre and post manufacturing indicate the tubes were well consolidated.
Failure Morphology

Of the 45 PPMC samples tested, 9 samples experienced catastrophic failure or failed incorrectly. Four of the failed samples were manufactured with the PPMC sheet rotated 90° from its usual wrapping orientation. This resulted in the tubes having excessively thin ends and being difficult to bevel. In one case, the beveling process resulted in the complete failure of the sample. Two of the remaining five samples that catastrophically failed did not consolidate well during manufacturing as an enormous amount of air was leaking from the bladder. The surface texture of the tubes revealed the resin did not distribute throughout the part, and there were a significant number of voids along the exterior surface. Both of these samples came from the same parent tube. The final 3 samples that failed incorrectly did not have well defined bevels, as all of these tubes failed at the bottom of the tube instead of failing at the trigger.

Figure 3.21 provides images of the failure morphology of a typical PPMC tube. Graphical representations of the PPMC failure mode, similar to those found in literature for continuous fiber failure modes, were created for ease of comparison in the future. In general, the progressive failure of PPMC tubes showed characteristics of the splaying failure mode found in continuous fiber laminates. Delamination occurs between individual platelets, this can be seen in the stereoscope image labeled cross section, and large sections of PPMC mat curl off from the tube in a splaying fashion. However, these sections of curled PPMC mat are larger than fronds seen in continuous fiber splaying mode, and they are commonly asymmetric. The location and size of the splayed fronds likely depends on local orientation of platelets. The majority of failed PPMC tubes had between 3 and 5 separate fronds that split off from the tube, compared to the 20-40 fronds produced with splaying failure mode in continuous fiber tubes.

The radius of curvature associated with the PPMC splayed fronds is smaller than that of a continuous fiber axial tube, producing permanent deformation. The PPMC
material has a smaller bending stiffness than continuous axial fibers, and the PPMC fronds do not spring back once the compressive load is removed. Debris wedges help the delamination process in the PPMC tubes and are located in between most platelets that have separated from their neighbors. In the continuous fiber axial tubes, delamination between plies extended the furthest at the internal and external edges of the tube, while the separation between plies in the middle of the laminate was limited. The PPMC tubes did not follow this trend, and there does not seem to be a relationship between longest delamination and location along the radial direction of the tube.

During testing, small platelet bundles broke off from the tubes and fell away. This is similar to the fragmentation mode seen with continuous fiber tubes. However, the continuous fiber bundles that broke off formed circular rings that encased the tube. These rings were produced from cracks in the matrix growing large enough to shear away from the primary tube structure. The PPMC platelet bundles that fell away from the tube resulted from overlapping platelets that form a mat that holds together. When the platelets are delaminated from their neighbors, they cannot support one another and fall apart.

Load-Displacement Curve

The load-displacement curves produced by PPMC tubes are similar to those seen with the fragmentation mode of continuous fiber tubes. Figure 3.22 provides 4 example load-displacement curves from PPMC samples tested. In general, there is an increase in force until a peak load is reached followed by a rapid drop in loading. In 3 of the 4 example curves, the peak load is definitively higher than the stable crushing load. This is traditionally seen with load-displacement curves of the splaying fiber failure mode. However, example PPMC 2 is more like a fragmentation mode curve with a peak load at a similar value of the stable crushing load. Inconsistent oscillations are seen in the stable crush zone, however they’re not as jagged and abrupt
Figure 3.21. Graphical representation versus experimental results of PPMC failure
as the oscillations seen in the load displacement curves for the fragmentation failure mode.

Figure 3.22. Example load-displacement curves for 4 PPMC test samples

An average load displacement curve for PPMC samples that progressively failed was produced and can be seen in Figure 3.23. Averaging the curves smoothed out the oscillations and revealed that the average stable crushing load for these PPMC samples was 22.4 kN with a standard deviation of 2.2 kN. The average peak load was 30.0 kN with a standard deviation of 3.9 kN. Figure 3.23 shows an average peak load much lower than 30.0 kN; however, this is artificially lowered because the location of the peak loads did not occur all at the same displacement.
SEA

The specific energy absorption was calculated for 27 of the 45 samples. As previously mentioned, 9 of the samples failed catastrophically, and therefore did not warrant a SEA calculation. The remaining 9 samples were not weighed before they were tested. While the specific energy absorption was calculated using both Eq. 2.3 and Eq. 2.4, Eq. 2.4 produced more consistent results that did not rely on estimates for density and cross sectional area. Unfortunately, this was not considered in early rounds of testing, resulting in the first 9 samples not being weighed prior to testing. Therefore, the specific energy absorption was calculated for 13 samples
manufactured with the composite mold and 14 samples manufactured with the aluminum mold. The averages and standard deviations can be seen in Table 3.4. The stable crushing load is almost identical between samples made from the two molds, however the tubes manufactured with the aluminum mold weighed more on average than those produced from the composite mold. This resulted in the tubes made with the aluminum mold having a lower SEA. While the tubes with the aluminum mold appeared to be consolidated better, it did not seem to make a difference in specific energy absorption.

<table>
<thead>
<tr>
<th></th>
<th>Composite Mold</th>
<th>Aluminum Mold</th>
</tr>
</thead>
<tbody>
<tr>
<td>Average SCL [kN]</td>
<td>22.7</td>
<td>21.98</td>
</tr>
<tr>
<td>St. Dev. [kN]</td>
<td>2.5</td>
<td>1.61</td>
</tr>
<tr>
<td>Average Weight [g]</td>
<td>40.77</td>
<td>42.22</td>
</tr>
<tr>
<td>St. Dev. [g]</td>
<td>2.43</td>
<td>2.24</td>
</tr>
<tr>
<td>Average SEA [kJ/kg]</td>
<td>56.03</td>
<td>50.25</td>
</tr>
<tr>
<td>St. Dev. [kJ/kg]</td>
<td>3.74</td>
<td>2.82</td>
</tr>
</tbody>
</table>

There was a spread of 16.12 kJ/kg in the SEA for the 27 PPMC tubes. Potential causes of the large variation in SEA include manufacturing parameter variability, final orientation state of the platelets, and geometry of the tubes. While all of the manufacturing parameters could not be completely controlled, the process was kept as consistent as possible for each manufacturing cycle. There were multiple instances of samples cut from the same parent tube performing similarly while samples from different tubes had a larger difference in SEA. An example of this can be seen in Figure 3.24 where the load-displacement curves for two sets of samples are compared. It is clear that the samples from the same parent tube perform similarly while there is a larger difference in stable crushing load between samples from different initial tubes. This would indicate that manufacturing parameters affect SEA of the tubes. However, this trend was not a consistent occurrence. It is also impossible to identify
which manufacturing parameter, like pressure, temperature, or time in the oven was the primary driver of SEA difference without further manufacturing studies.

![Graph showing load vs. displacement for test samples](image)

**Figure 3.24.** Comparison of load-displacement curves for test samples produced from the same parent tube

The continuous fiber study revealed that SEA increased with laminate stiffness, and therefore, the final orientation state of the platelets within the tubes can affect the specific energy absorption. A rudimentary orientation study was completed in order to determine if there was a large variability in platelet orientation distribution of the PPMC sheets prior to molding. As the platelets do not flow during the bladder molding process any large orientation distribution differences in the sheets would result in stiffness changes of the tubes.
Pictures of the front and back surfaces of the PPMC sheets were taken prior to the molding. The images were ran through a 2D orientation application created by Dr. Benjamin Denos at Purdue University [76]. This application takes images of a surface and estimates the in-plane platelet orientation distribution. Users are allowed to alter the image and adjust filter settings for the analysis method. The application draws representative vectors on the picture to show estimated orientations in the local area, and users can visually compare the vectors to the actual image and adjust parameters as necessary.

While the pictures were taken on both surfaces of the PPMC sheets, they are only able to capture surface orientation distributions. Therefore, these results do not speak to the orientation state in the middle of the material system. However, a CT scan study by Dr. Denos revealed that the through-thickness orientation state of the PPMC material was comparable to that predicted by 2D surface measurements. From these results, it became reasonable to assume the orientation distributions predicted by the 2D orientation application were representative of the material system as a whole.

Figure 3.25 provides example outputs from the orientation application for the front and back surfaces of 3 PPMC sheets. The x-axis represents the platelet orientation angle, while the y-axis provides the probability of seeing a platelet at that specific angle. In these images, an angle of 90° corresponds to an orientation along the longitudinal axis of the tube.

In general, the orientation distribution was found to be similar for all of the measured PPMC sheets. In all of the analysis images, there is a peak at 90° followed by a dip in the area around 150°, so this is not a completely random orientation distribution with equal probability for each angle. However, these results indicate that the global stiffness for the tubes are similar. The orientation analysis was completed on a 10”x10” PPMC sheet used for manufacturing a single tube (two samples). Orientation distribution for smaller areas corresponding to each sample did not show the same consistency, but it was seen previously that two samples from the same tube
generally performed similarly. Therefore, it was determined that there was not a large enough variability in platelet orientation distribution to further investigate this as a major cause of the spread in SEA for PPMC tubes.

The specific energy absorption of metal tubes is dependent upon the t/D ratio. As the PPMC tubes had noticeable variations in thickness along the circumference of the tube as well as from sample to sample, it would not be illogical to assume the t/D ratio affects the SEA of PPMC tubes. The SEA based on the weight and height measurements of the samples was compared to the average thickness and inner diameter of the PPMC tubes. In general, the PPMC tubes made with the aluminum mold were thinner but had a larger inner diameter (see Table 3.2), this means they have a smaller t/D ratio. In addition, it was seen that the aluminum mold produced samples that had a lower SEA. Figure 3.26 indicates there is a slight upward trend in SEA versus t/D ratio for PPMC tubes. A simulation study covered in Chapter 6 attempts to quantify how much the t/D ratio actually affects SEA for PPMC tubes. It is likely the spread in SEA is a result of a combination of the presented factors. Without further control studies, it is impossible to pinpoint the dominating factor.

3.4 Conclusions on Quasi-Static Testing

The primary question to be answered for OEMs with regards to the energy absorption capability of PPMC material is: how does the energy absorption of PPMC compare to materials already prevalent in industry? There is not a straightforward answer to this because the geometry and the stiffness of the laminate affect the specific energy absorption.

Specific energy absorption results of PPMC compared to the closed form solutions of metal can be seen in Figure 3.27. Low carbon steel was assumed to fail in the concertina failure mode matching what was seen in experiment, and aluminum 6061 was assumed to fail in the diamond failure mode. It is evident that the energy
Figure 3.25. Example orientation distributions from the 2D orientation analysis application
absorption of PPMC is comparable to that of aluminum 6061 with similar t/D ratios, and it is far superior to that of steel with the same t/D ratio.

Previously, the continuous fiber SEA data was plotted against the % of axial fibers in the layup, but the platelet orientations of the PPMC material are not limited to axial and hoop directions. A new metric was required to compare the SEA results of the PPMC to the continuous fiber results. If a tensile bar of aligned, staggered platelets is manufactured and tested, it would have about half the strength of a unidirectional continuous fiber sample [77]. However, the stiffness of the staggered platelet sample would be similar to that of the continuous fiber sample. With this in mind, the laminate stiffness was chosen as a comparison metric for SEA. Figure 3.28 plots the SEA
of PPMC and continuous fiber tubes based off of the estimated laminate stiffness of the tube. The longitudinal stiffness of the continuous fiber laminate was determined using the classical laminate plate theory tool on the cdmHUB [78] in conjunction with the UD tape properties characterized in the lab at the CMSC. The estimated longitudinal stiffness for the PPMC tubes was calculated using the equations for prediction of mechanical properties of short fiber systems provided in the appendix of a paper by Advani et al. [79]. The orientation distribution for the platelets was assumed to be 2D random along the longitudinal axis; and the mechanical properties for the UD tape used in the analysis are seen in Table 5.1.

Figure 3.28 provides confirmation that the SEA of PPMC tubes is comparable to the energy absorption of continuous fiber laminates with similar longitudinal stiffnesses. The spread of SEA for the PPMC tubes is larger, but more PPMC tubes

![Figure 3.27. Experimental SEA of PPMC tubes versus closed form solutions for metal crush tubes](image-url)
were tested and they had considerable variability in geometric measurements. It is not unreasonable to assume that the SEA would increase with increased longitudinal alignment of platelets. However, that is a study for another time.

![Graph showing Experimental SEA versus laminate stiffness for PPMC and continuous fiber tubes](image)

Figure 3.28. Experimental SEA versus laminate stiffness for PPMC and continuous fiber tubes

The crush failure of carbon fiber/epoxy PPMC has proven to be comparable to continuous fiber material systems with similar mechanical properties. In addition, the system performed better in crush than steel and on par with aluminum 6061 with similar geometries. These are positive signs for manufacturers interested in using PPMC to create load bearing structures with complex geometries. Additional research investigating the performance of thermoplastic PPMC compared to the VORAFUSE™ system and a platelet alignment study would be helpful for further understanding of energy absorption mechanisms with PPMC.
3.5 Dynamic Testing

After all rounds of quasi-static testing were completed, preparation for high speed testing began in order to compare the specific energy absorption for PPMC tubes crushed at high and low speeds. The goal was to increase the speed of the crush tests to a value that could be seen in vehicular crash.

There are three main requirements for a successful high speed crush test: speed, energy, and a suitable test setup. The MTS machines used for quasi-static testing do not have the capability of displacing faster than 0.1 m/s. Test specimens for a Split-Hopkinson pressure bar are usually shorter (from around 5 - 20 mm) and produce results on the order of microseconds [80, 81]. The small specimen size would not be able to incorporate more than a few platelets, and the speed of impact for the Split-Hopkinson test is faster than what is seen in vehicular crash.

Previous high speed work completed by a group at Oak Ridge National Labs used the Test Machine for Automotive Crashworthiness (TMAC) for crushing composite and metal tubes. This machine can obtain speeds of 8 m/s while performing 50 kJ of work on a system. Unfortunately, this system was not available for testing PPMC tubes, so a smaller drop tower was implemented as the next best option.

The quasi-static results showed that PPMC tubes could absorb about 50 kJ/kg, and tubes made from the aluminum mold had an average weight of 42 g. With these results, a single tube could absorb around 1200 J. An Instron 9250 HV drop tower capable of a 4.0 meter free fall producing 1200 J was selected for testing. Adjusting the free fall height changes the energy entering the system, so a series of drop heights and impact velocities could be tested.

3.5.1 Drop Tower Fixture Design

The drop tower used an un-instrumented steel tup with an outer diameter of 0.75” and a length of 24” as the impacting rod. As the diameter of the tup was considerably smaller than that of the crush tubes, a fixture was designed to distribute the energy
from the impacting tup to the composite crush tube. The high speed crush fixture (seen in Figure 3.29) is split into three steel plates aligned via a four rail system. The bottom and middle plates are stationary and hold the test specimen and fixture in place. The top plate moves vertically on the four alignment rods with the help of flanged bronze bushings.

Figure 3.29. Drawing of 4-rail drop tower test fixture with the top plate removed
The bottom steel plate is 6” x 6” x 0.75” with four 0.625” diameter post holes drilled into the corners centered 0.7” from each edge. Three 1” deep holes were drilled and threaded on each side of the bottom plate to attach THORLABS heavy-duty magnetic bases to hold the fixture during testing. The four steel rods used for alignment were welded to the bottom plate.

The middle plate had the same dimensions as the bottom plate as well as the same four post holes. A 2.5” diameter hole was bored through the center of the plate to hold a crush tube. Additionally, four 0.375” holes were drilled and threaded on the side of the plate for set screws to tighten the position of the crush tube. The middle plate was welded to the alignment rods approximately 0.75” away from the bottom plate. The placement of the middle plate allows 62 mm of displacement (for a 100 mm crush tube) before the top plate hits the middle plate.

The top plate is 6” x 6” x 1” and has 1.125” diameter post holes drilled into the top to accommodate bronze bushings with an inner diameter of 0.625”. The bronze bushings have a flange on either side of the plate to prevent the bushings from being knocked loose during testing. A small divot was machined into the center of the top plate about 0.5” in diameter and 0.25” deep to provide a landing spot for the impacting tup. Engineering drawings for the test fixture plates are provided in Appendix B.

Figure 3.30 shows the fixture in situ. Three THORLABS magnetic bases, each rated for 175 lbs, were attached to the test fixture and activated prior to testing. In addition, the fixture was centered to ensure the tup impacted the divot in the top plate as an eccentric load risked locking the top plate and damaging the fixture.

3.5.2 Dynamic Test Method

The drop tower was outfitted with additional weight to achieve the highest possible potential energy for the setup. The free fall weight was 28.66 kg, and the drop heights varied from 2.0 m to 4.0 m during testing. The impact velocity and potential energy
were calculated for each of the tests cases prior to weight release using basic physics. Impact velocity was calculated using

\[ V_f = \sqrt{V_0^2 + 2ad} \]  \hspace{1cm} (3.1)

In the case of the drop tower, the initial velocity was 0 m/s, the acceleration, \( a \), was provided by gravity and equal to 9.81 m/s\(^2\), and the distance \( d \), was the drop height in meters.

The potential energy of the system was calculated with the following equation:

\[ PE = mgh \]  \hspace{1cm} (3.2)
The drop height, impact velocity, and potential energy for each of the test cases is listed in Table 3.5.

<table>
<thead>
<tr>
<th>Drop height [m]</th>
<th>Impact Velocity [m/s]</th>
<th>Potential Energy [J]</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.0</td>
<td>6.263</td>
<td>562.22</td>
</tr>
<tr>
<td>2.5</td>
<td>7.002</td>
<td>702.776</td>
</tr>
<tr>
<td>3.0</td>
<td>7.670</td>
<td>843.33</td>
</tr>
<tr>
<td>3.5</td>
<td>8.285</td>
<td>983.89</td>
</tr>
<tr>
<td>4.0</td>
<td>8.857</td>
<td>1,124.4</td>
</tr>
</tbody>
</table>

The test specimen was placed in the fixture, and the set screws held it in place. Before each test, the displacement of the drop tower was zeroed at the location where the tup met the top plate. Two cameras were set to record the test at a frame rate of 240 Hz. The drop weight was raised to the designated height, recording started, and the weight was released and impact followed. The time history of the load was recorded at about 500 kHz with 8192 data points spanning roughly 15 milliseconds.

### 3.5.3 Results

Nine PPMC samples made with the aluminum mold were tested at 5 different drop heights. As load and time were the only variables recorded during testing, additional post processing was completed to calculate the velocity, displacement, acceleration, and energy. The acceleration was calculated using Newton’s second law of motion. As acceleration, velocity, and displacement are all related, the velocity and displacement could be calculated using the first and second integral of the acceleration with respect to time. The energy absorbed during testing was determined by integrating the load-displacement curve. Figure 3.31 provides the plots for a single test sample with a drop height of 2.0 meters.
Figure 3.31. Load, acceleration, velocity, displacement, and energy calculations for a PPMC tube crushed at a drop height of 2.0 meters
There is a clear oscillation in the load versus time data with a frequency of about 600 Hz. The cause of the oscillation is unknown as the frame rate for the video was not fast enough to see the oscillations. It is possible the tup and top plate made contact multiple times or the composite failed in an oscillatory fashion like ductile metals.

The failure of the PPMC crush tubes was different than what was seen in quasi-static testing. Images from testing revealed a large initial displacement immediately after contact followed by a large debris cloud with dust erupting from the tube. The tup bounced upwards after the initial impact and comes back to rest on the setup as the debris settles. It is unclear whether the bouncing of the tup is related to the oscillations in the load time history. After testing each sample, a cloud of dust remained in the air for a few minutes and there was an acrid odor possibly associated with the debris. This was unique to high speed tests.

Figure 3.32. Sequence of crash images from crush test with h = 3.0 meters showing cloud of debris and dust forming immediately after impact

The large failure fronds seen in quasi static testing were not present in the failure of the drop tower test samples. The majority of the debris was found in the hollow
of the tube after testing, and chunks of debris were usually no larger than 1 square inch. Figure 3.33 shows a side view of PPMC tubes tested with drop heights of \( h = 2.0 \) m to \( h = 4.0 \) m. As the drop height increased, the maximum displacement of the tup also increased. This is to be expected as more energy is being inserted into the system. Ideally, if the crush tubes presented a consistent specific energy absorption, the maximum displacement achieved for each tube would be directly related to the drop height and therefore, the energy absorbed.

![Figure 3.33. Remains of PPMC crush tubes (in order) after drop tests with \( h = 2.0, 2.5, 3.0, 3.5, \) and \( 4.0 \) m](image)

The maximum displacement was plotted against the energy absorbed for each of the 9 samples tested, and a positive trend was captured with limited scatter. This can be seen in Figure 3.34. With so few data points, additional testing would help solidify this trend for PPMC.

The maximum load, displacement, energy absorbed, and specific energy absorption for each sample are listed in Table 3.6. The maximum load and displacement did not always increase with the drop height, but a positive trend was captured, as was seen in the previous figure. The specific energy absorption ranged from 42.01 kJ/kg to 51.04 kJ/kg, with an average of 47.7 kJ/kg.

Table 3.7 reveals that the average SEA of the PPMC tubes made from the aluminum mold were within a standard deviation of each other for the quasi-static and dynamic tests. For this material system and speeds tested, it seems that test speed
Table 3.6. Max load, displacement, energy absorbed, and SEA calculated for drop tower test samples

<table>
<thead>
<tr>
<th>Sample</th>
<th>Height [m]</th>
<th>Max Load [kN]</th>
<th>Max Disp. [mm]</th>
<th>Energy Absorbed [J]</th>
<th>SEA [kJ/kg]</th>
</tr>
</thead>
<tbody>
<tr>
<td>13B</td>
<td>2.0</td>
<td>101.74</td>
<td>31.8</td>
<td>555.65</td>
<td>42.01</td>
</tr>
<tr>
<td>9A</td>
<td>2.5</td>
<td>89.33</td>
<td>29.4</td>
<td>702.54</td>
<td>50.48</td>
</tr>
<tr>
<td>9B</td>
<td>3.0</td>
<td>113.5</td>
<td>35.9</td>
<td>843.01</td>
<td>49.27</td>
</tr>
<tr>
<td>12A</td>
<td>3.5</td>
<td>92.12</td>
<td>50.2</td>
<td>983.22</td>
<td>49.65</td>
</tr>
<tr>
<td>10A</td>
<td>4.0</td>
<td>140.59</td>
<td>55.7</td>
<td>1,123.3</td>
<td>48.09</td>
</tr>
<tr>
<td>12B</td>
<td>3.5</td>
<td>97.60</td>
<td>51.4</td>
<td>983.38</td>
<td>45.11</td>
</tr>
<tr>
<td>10B</td>
<td>3.5</td>
<td>149.16</td>
<td>49.9</td>
<td>983.55</td>
<td>45.03</td>
</tr>
<tr>
<td>14B</td>
<td>2.0</td>
<td>96.15</td>
<td>28.0</td>
<td>556.24</td>
<td>51.04</td>
</tr>
<tr>
<td>14A</td>
<td>2.5</td>
<td>80.68</td>
<td>36.8</td>
<td>695.14</td>
<td>48.96</td>
</tr>
</tbody>
</table>

does not affect the energy absorption significantly. This is an important conclusion, as researchers have not been able to come to a definitive answer regarding the effect
of speed on SEA [47]. In fact, Hull [32] saw a 50% drop in SEA when jumping from a test speed of 4 mm/s to 4 m/s for a specific glass fiber layup. The SEA of quasi-static tubes made with the composite mold was still higher than that of tubes made with the aluminum mold, but that can be related to the difference in thickness-to-diameter ratio between the molds.

Table 3.7. Energy absorption comparison between dynamic and quasi-static tests

<table>
<thead>
<tr>
<th></th>
<th>Quasi-Static Composite Mold</th>
<th>Quasi-Static Aluminum Mold</th>
<th>Dynamic Aluminum Mold</th>
</tr>
</thead>
<tbody>
<tr>
<td>Average Weight [g]</td>
<td>40.77</td>
<td>42.22</td>
<td>40.9</td>
</tr>
<tr>
<td>St. Dev. [g]</td>
<td>2.43</td>
<td>2.24</td>
<td>3.26</td>
</tr>
<tr>
<td>Average SEA [kJ/kg]</td>
<td>56.03</td>
<td>50.25</td>
<td>47.7</td>
</tr>
<tr>
<td>St. Dev. [kJ/kg]</td>
<td>3.74</td>
<td>2.82</td>
<td>3.02</td>
</tr>
</tbody>
</table>

During quasi-static testing, it was noticed that samples from the same parent tube performed similarly during testing. With this in mind, the time histories of load from dynamic tests were compared for samples from the same parent tube. Figure 3.35 presents load histories of the samples from 3 parent tubes, Tube 9, Tube 10, and Tube 12. These names do not correspond to the number of the sample during dynamic testing but the number of the parent tube made with the aluminum mold. For Tube 9 and Tube 10, the samples were crushed with different drop heights, however the load versus time curves are almost identical. This can be somewhat misleading though, because the max displacements and energy absorbed were different between samples from the same parent tube. The samples from Tube 12 were tested at the same height and produced slightly different magnitudes in the initial oscillations. However, the shape of the load versus time curves are still similar.
Figure 3.35. Time history of dynamic load data for PPMC samples from the same parent tube
3.5.4 Conclusions from Dynamic Testing of PPMC Tubes

There are two primary conclusions regarding the dynamic results of PPMC tubes: the SEA is similar at both high- and low- speed tests and samples from the same parent tube perform similarly regardless of test speed. As previously stated, researchers have not found consistency in results regarding the energy absorption of tubes tested at different speeds. Work by Farley produced contradictions regarding the dependence of SEA on test speed [44, 66]. It is likely that the dependence on test speed is material system dependent. However, that was not proven in this work. In addition, the trend of samples from the same parent tube performing similarly was reaffirmed in high speed testing. It is still unclear why this happens, but it is most likely related to the inconsistency in the manufacturing cycle. Adding further control to the cure cycle might reduce the variability between tubes.

Additional high speed testing is needed to better understand performance variability as several drop heights only had one data point. It was unclear from video and load data whether the top plate of the test fixture crushed the tube all the way down to the middle plate when a drop height of 4.0 meters was used. Longer test samples could be manufactured to prevent this from happening when using the maximum drop height. On top of this, a high speed camera is needed to identify the cause of the oscillations in the load data. Future work should try to calculate the amount of energy that is being dissipated by the friction of the bronze bushings, the magnets, and the test fixture itself.
4. SIMULATING CRASH BEHAVIOR OF COMPOSITES

Crash simulation of composite materials is a difficult task even for the most astute modeler due to the anisotropy of the material system and the complexity of composites in failure. Introducing PPMC as the material system complicates things further because the effects of manufacturing and the final fiber orientation state must be accounted for in the model. While it is common for researchers to introduce new material models or adaptations to finite element software to capture composite failure, this work will not approach that topic. Instead, the goal of this research is to take existing material models, methods, and software and apply them to a new material system. This allows for a study of the potential limitations of existing capabilities in order to determine next steps regarding best modeling methods for PPMC in crash conditions.

In general, crash simulations allow designers to run components or systems through a variety of loading conditions to predict performance in high speed impacts. Modelers look at how the component or system fails, the load curve during the event, and the energy absorbed during the impact. These can all be directly compared to the variables measured during crush tests. If enough of the simulation load cases can be validated with experiments, the material card or modeling method can be used to extrapolate performance from load cases that are not physically tested.

However, predicting crash performance of composite materials using finite element analysis has proven difficult as failure location and strength are not consistent from sample to sample, even in continuous fiber components. In fact, it is quite common to use experimental results of component tests to “perfect” the simulation so it will match the experiment and can then be used to extrapolate further test cases. If a perfect performance prediction was necessary for a new material system to be introduced into production, composite materials would have a difficult time finding
traction in the transportation industry. This brings the following question to the forefront: what is the criteria for simulation results to be considered representative of real components under testing? As industry requirements and tolerances are not publicly available, this work did not focus on meeting an accuracy goal. Instead, this research tried to determine how close simulation results of PPMC components could get to their experimental counterparts without manipulating characterized material properties or adjusting model parameters to obtain a more favorable numerical result.

4.1 Finite Element Software

A variety of finite element software has the capability to investigate failure due to high speed impacts, and many times, the choice is ultimately dependent upon software availability and previous experience of the modeler. This work will focus specifically on crash modeling with the software LS Dyna created by Livermore Software Technology Corp, an offshoot of Lawrence Livermore National Labs. This choice was made because the company funding the project works primarily with LS Dyna for crash simulation.

Although LS Dyna is the selected software for this work, strides have been made using alternative programs such as Dassault Systemes’ Abaqus, ESI’s PAM-CRASH, and Altair’s RADIOSS. In fact, an original equipment manufacturer (OEM) presented a conference paper in 2005 detailing their transition to Abaqus as their crash simulation software of choice [82]. The user material (UMAT) feature in Abaqus has been used to incorporate 3D continuum damage mechanics (CDM) models for continuous fiber thermoplastic and braided tubes with initial success [83–86]. These researchers aimed to create predictive crash simulations without requiring non-physical material parameters and a comparison to experimental component tests. However, there are computational disadvantages to using solid elements with a 3D material model. A single sinusoidal component simulation took up to 40 hrs to complete with 16 CPUs on a high performance computing cluster (HPCC). Using the same modeling method
and material definition for a full system simulation, a car for example, could potentially take weeks. This makes such a modeling method impractical for engineers working in industry.

Greve et al. [87] shared these concerns, and presented a phenomenological model for crush tubes using the software PAM-CRASH that was deemed “CPU-efficient” for industrial applications.

4.2 Material Models for Crash Simulation of Composites in LS Dyna

LS Dyna offers a multitude of material models for orthotropic and composite materials, and this work required a material model that could capture progressive failure in a variety of loading conditions. Two of the most common composite damage models used for crash simulations are MAT 54, a progressive failure model called *MAT_ENHANCED_COMPOSITE_DAMAGE, and MAT 58, a continuum damage mechanics model also referred to as *MAT_LAMINATED_COMPOSITE_FABRIC. Both of these material models were investigated for use with PPMC.

4.2.1 MAT 54

MAT 54 is a progressive failure model that is compatible with shell and thick shell elements [88]. There are 4 separate definitions for failure built into MAT 54: 1) the stress based Chang-Chang failure criterion, 2) ultimate strain values for each direction, 3) a general effective strain that applies to all directions, and 4) element failure due to time step reduction.

The Chang-Chang failure criterion was initially created as a damage model capable of predicting ultimate strength of laminates with stress concentrations [89]. The model assumes plane stress conditions [90], and is therefore best suited for shell elements. The model defines 4 strength based failure modes for fiber tension, fiber compression, matrix tension, and matrix compression. When a failure is reached in a lamina, associated parameters for the ply are set to zero and the analysis continues.
Assuming the 1-direction represents the axis of fiber orientation and the 2-direction represents an axis transverse to the fibers, the failure mode for fibers in tension, when $\sigma_{11} > 0$, can be defined as

$$e_f^2 = \left( \frac{\sigma_{11}}{X_t} \right)^2 + \beta \left( \frac{\tau_{12}}{S_c} \right)^2 \begin{cases} \geq 1 & \text{failed} \\ < 1 & \text{elastic} \end{cases}$$  \hspace{1cm} (4.1)$$

Upon failure, $E_1 = E_2 = G_{12} = \nu_{21} = \nu_{12} = 0$

where $X_t$ is the tensile strength of the lamina in the 1-direction, $S_c$ is the shear strength of the lamina, and $\beta$ is a shear stress weighting factor chosen by the user. When $\beta$ is set to 1, the fiber tensile failure is Hashin’s failure criteria [91]. When $\beta$ is set to 0, the maximum stress criteria is used [90]. For this work, $\beta = 1$.

The failure mode for fibers in compression, when $\sigma_{11} \leq 0$, can be defined as

$$e_c^2 = \left( \frac{\sigma_{11}}{X_c} \right)^2 \begin{cases} \geq 1 & \text{failed} \\ < 1 & \text{elastic} \end{cases}$$  \hspace{1cm} (4.2)$$

Upon failure, $E_1 = \nu_{21} = 0 = \nu_{12} = 0$

where $X_c$ is the compressive strength of the lamina in the 1-direction.

The failure mode for the matrix in tension, when $\sigma_{22} > 0$, is defined as

$$e_m^2 = \left( \frac{\sigma_{22}}{Y_t} \right)^2 + \left( \frac{\tau_{12}}{S_c} \right)^2 \begin{cases} \geq 1 & \text{failed} \\ < 1 & \text{elastic} \end{cases}$$  \hspace{1cm} (4.3)$$

Upon failure, $E_2 = \nu_{21} = G_{12} = 0$

where $Y_t$ is the tensile strength of the lamina in the 2-direction.
The final failure mode is for compression of the matrix, when $\sigma_{22} \leq 0$, and is defined as

$$e_d^2 = \left( \frac{\sigma_{22}}{2S_c} \right)^2 + \left[ \left( \frac{Y_c}{2S_c} \right)^2 - 1 \right] \frac{\sigma_{22}^2}{Y_c} + \left( \frac{\tau_{12}}{S_c} \right)^2 \begin{cases} \geq 1 \rightarrow \text{failed} \\ < 1 \rightarrow \text{elastic} \end{cases}$$

Upon failure, $E_2 = \nu_{21} = \nu_{12} = G_{12} = 0$

The second failure definition for MAT 54 defines ultimate strain values for different loading conditions. The parameters DFAILC, DFAILT, DFAILM, and DFAILS allow the user to specify a maximum strain for an element in fiber compression, fiber tension, matrix tension/compression, and shear respectively. If these variables are greater than zero, failure will occur in an element after one of the strain levels has been reached. Once failed, an element is deleted from the simulation. These strain parameters provide load carrying capabilities to an element past its defined strength. This failure definition works in conjunction with the stress based failure definition of the Chang-Chang model.

The third failure definition is through a parameter called EFS, which stands for effective element strain. This failure mechanism is similar to the DFAIL parameters, but it sets the ultimate strain (for all directions) in an element to a single value designated by EFS. If EFS is greater than zero, all strains will be checked against this parameter throughout the simulation. EFS cannot be used at the same time as the DFAIL parameters because they both define ultimate failure strains.

The final failure definition for MAT 54 is through the TFAIL parameter. TFAIL limits the time step for individual elements. If an element has decreased in size enough to have a time step smaller than the value specified by TFAIL, the element is deleted. This can be used as a tool to speed up explicit simulations dominated by the time steps of a few small elements.

This work employs the Chang-Chang failure criteria and the DFAIL parameters but not the effective element strain. TFAIL must have a non-zero value assigned in order for the crash front feature to activate in LS Dyna, but a small enough value
(TFAIL = 1E-09) was provided to ensure elements were not deleted via that failure definition.

MAT 54 has over 45 different inputs, and these vary from material properties to numerical constructs that determine the element response after initial failure. The input variables for MAT 54 are split into two categories: the dynamic parameters that are changed frequently throughout this work (Table 4.1) and the static parameters that are rarely or never altered in the course of this work (Table 4.2). The dynamic parameters are primarily found through material characterization, however, a few of the numerical constructs like the SLIM and SOFT parameters are given values based on trial and error.

The static parameters are either unused or always kept constant within this work and include variables to activate strain rate dependent properties as well as alternative ultimate strain definitions.

It is difficult to understand the importance of individual parameters without visually seeing how they relate to the performance of the material. Figure 4.1 provides stress-strain curves for the 1 and 2 directions of a lamina defined with MAT 54 under uniaxial loading. In the elastic region of the material, the stiffness is defined by the given Young’s modulus. Once a strength value is achieved, in either tension or compression, the stress immediately reduces to a value determined by a SLIM parameter multiplied by its related strength. This reduced stress is held until the ultimate strain defined by a DFAIL parameter is attained.

For example, if an element is loaded in compression along the 2 direction, the stress will increase at a rate dependent upon the Young’s modulus, EB. Once the stress reaches the matrix compressive strength, YC, the stress will immediately reduce to a value of SLIMC2*YC. The elemental stress will remain constant until the ultimate strain -DFAILM is reached. The strengths, Young’s moduli, and DFAIL parameters can all be calculated with experimental data, but the SLIM parameters are not representative of a physical property. Instead, the SLIM parameters are usually determined via trial and error.
Table 4.1. LS Dyna MAT 54 inputs - Dynamic Parameters

<table>
<thead>
<tr>
<th>Variable</th>
<th>Definition</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>MID</td>
<td>Material identification number</td>
<td></td>
</tr>
<tr>
<td>RO</td>
<td>Mass density</td>
<td></td>
</tr>
<tr>
<td>EA EB EC</td>
<td>Young’s moduli</td>
<td>Fiber, transverse, and normal</td>
</tr>
<tr>
<td>PRBA PRCA PRCB</td>
<td>Poisson’s ratios</td>
<td>2-1, 3-1, and 3-2 directions</td>
</tr>
<tr>
<td>GAB GBC GCA</td>
<td>Shear moduli</td>
<td>1-2, 2-3, and 3-1 directions</td>
</tr>
<tr>
<td>AOPT</td>
<td>Material axes option</td>
<td>Suggest AOPT=2</td>
</tr>
<tr>
<td>A1 A2 A3</td>
<td>Define vector $\mathbf{a}$ for AOPT=2</td>
<td>Defines fiber direction</td>
</tr>
<tr>
<td>D1 D2 D3</td>
<td>Define vector $\mathbf{d}$ for AOPT=2</td>
<td>Defines in-plane transverse direction</td>
</tr>
<tr>
<td>DFAILM</td>
<td>Max strain for matrix tension/compression</td>
<td>Only active if DFAILT&gt;0</td>
</tr>
<tr>
<td>DFAILS</td>
<td>Max shear strain when in tension</td>
<td>Only active if DFAILT&gt;0</td>
</tr>
<tr>
<td>TFAIL</td>
<td>Time step for element deletion</td>
<td>TFAIL&gt;0 for crash front to work</td>
</tr>
<tr>
<td>SOFT</td>
<td>Strength softening reduction factor</td>
<td>Applies to elements in crashfront</td>
</tr>
<tr>
<td>DFAILT</td>
<td>Max strain for fiber tension</td>
<td></td>
</tr>
<tr>
<td>DFAILC</td>
<td>Max strain for fiber compression</td>
<td>Input value should be negative</td>
</tr>
<tr>
<td>XC</td>
<td>Longitudinal compressive strength</td>
<td>Fiber direction</td>
</tr>
<tr>
<td>XT</td>
<td>Longitudinal tensile strength</td>
<td>Fiber direction</td>
</tr>
<tr>
<td>YC</td>
<td>Transverse compressive strength</td>
<td></td>
</tr>
<tr>
<td>YT</td>
<td>Transverse tensile strength</td>
<td></td>
</tr>
<tr>
<td>SC</td>
<td>Shear strength</td>
<td></td>
</tr>
<tr>
<td>PFL</td>
<td>% of failed layers to start crashfront</td>
<td>Default 80%</td>
</tr>
<tr>
<td>SOFT2</td>
<td>Orthogonal strength reduction factor</td>
<td></td>
</tr>
<tr>
<td>SLIMT1</td>
<td>Factor for min stress limit after stress max</td>
<td>Fiber tension</td>
</tr>
<tr>
<td>SLIMC1</td>
<td>Factor for min stress limit after stress max</td>
<td>Fiber compression</td>
</tr>
<tr>
<td>SLIMT2</td>
<td>Factor for min stress limit after stress max</td>
<td>Matrix tension</td>
</tr>
<tr>
<td>SLIMC2</td>
<td>Factor for min stress limit after stress max</td>
<td>Matrix compression</td>
</tr>
<tr>
<td>SLIMS</td>
<td>Factor for min stress limit after stress max</td>
<td>Shear</td>
</tr>
<tr>
<td>SOFTG</td>
<td>Shear strength reduction factor</td>
<td></td>
</tr>
</tbody>
</table>

MAT 54 does not lend itself well to materials with nonlinear responses, and the immediate drop in load carrying capability is aggressive considering some resin systems can soften prior to ultimate failure. MAT 58 addresses that concern by creating a softening response pre and post failure.
Table 4.2. LS Dyna MAT 54 inputs - Static Parameters

<table>
<thead>
<tr>
<th>Variable</th>
<th>Definition</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>KF</td>
<td>Bulk modulus</td>
<td>Not used, is inactive</td>
</tr>
<tr>
<td>2WAY</td>
<td>Flag for 2-way fiber action</td>
<td>Keep at 0.0 for standard behavior</td>
</tr>
<tr>
<td>V1 V2 V3</td>
<td>Define vector v for AOPT=3</td>
<td></td>
</tr>
<tr>
<td>MANGLE</td>
<td>Material angle</td>
<td>For AOPT=0 and AOPT=3</td>
</tr>
<tr>
<td>ALPH</td>
<td>Shear stress parameter for nonlinear term</td>
<td>Keep at 0.0 for linear behavior</td>
</tr>
<tr>
<td>FBRT</td>
<td>Softening for fiber tensile strength</td>
<td></td>
</tr>
<tr>
<td>YCFAC</td>
<td>Compressive fiber strength reduction factor</td>
<td>Only after matrix failure</td>
</tr>
<tr>
<td>EFS</td>
<td>Effective failure strain</td>
<td></td>
</tr>
<tr>
<td>CRIT</td>
<td>Failure criterion</td>
<td>Select CRIT=54</td>
</tr>
<tr>
<td>BETA</td>
<td>Weight factor for shear term</td>
<td>Keep at 0.0 for standard behavior</td>
</tr>
<tr>
<td>EPSF</td>
<td>Damage initiation transverse shear strain</td>
<td></td>
</tr>
<tr>
<td>EPSR</td>
<td>Final rupture transverse shear strain</td>
<td></td>
</tr>
<tr>
<td>TSMD</td>
<td>Transverse shear max damage</td>
<td></td>
</tr>
<tr>
<td>LCXC</td>
<td>Load curve ID for XC vs strain rate</td>
<td></td>
</tr>
<tr>
<td>LCXT</td>
<td>Load curve ID for XT vs strain rate</td>
<td></td>
</tr>
<tr>
<td>LCYC</td>
<td>Load curve ID for YC vs strain rate</td>
<td></td>
</tr>
<tr>
<td>LCYT</td>
<td>Load curve ID for YT vs strain rate</td>
<td></td>
</tr>
<tr>
<td>LCSC</td>
<td>Load curve ID for SC vs strain rate</td>
<td></td>
</tr>
<tr>
<td>DT</td>
<td>Strain rate averaging option</td>
<td></td>
</tr>
</tbody>
</table>

4.2.2 MAT 58

MAT 58 is a CDM model in LS Dyna based on the Matzenmiller, Lubliner, Taylor (MLT) model for anisotropic damage in composites that utilizes the Hashin failure criteria for a lamina [92,93]. MAT 58 is restricted to shell elements because the failure model assumes plane stress conditions.

MAT 58 has 3 separate failure definitions within its material model: 1) the Hashin failure criterion, 2) a general effective strain that applies to all directions, and 3) element failure due to time step reduction.

Hashin described 4 failure criteria for a lamina: the tensile fiber mode, compressive fiber mode, tensile matrix mode, and compressive matrix mode [91]. When plane stress is assumed, these equations simplify to forms similar to those seen for MAT 54. In fiber tension, when $\sigma_{11} \geq 0$, the failure mode is defined as
Figure 4.1. MAT 54 Behavior in fiber and matrix directions

\[ e_f^2 = \left( \frac{\sigma_{11}}{X_t} \right)^2 \begin{cases} \geq 1 & \text{failed} \\ < 1 & \text{elastic} \end{cases} \] (4.5)

In fiber compression, when \( \sigma_{11} < 0 \), the failure mode is described as

\[ e_f^2 = \left( \frac{\sigma_{11}}{X_v} \right)^2 \begin{cases} \geq 1 & \text{failed} \\ < 1 & \text{elastic} \end{cases} \] (4.6)

The tensile matrix mode, when \( \sigma_{22} \geq 0 \), incorporates a shearing term and is defined as
\( e_m^2 = \left( \frac{\sigma_{22}}{Y_t} \right)^2 + \left( \frac{\tau}{S_c} \right)^2 \begin{cases} \geq 1 & \text{failed} \\ < 1 & \text{elastic} \end{cases} \) (4.7)

Finally, the compressive matrix mode, when \( \sigma_{22} < 0 \), is defined as

\( e_d^2 = \left( \frac{\sigma_{22}}{Y_c} \right)^2 + \left( \frac{\tau_{12}}{S_c} \right)^2 \begin{cases} \geq 1 & \text{failed} \\ < 1 & \text{elastic} \end{cases} \) (4.8)

The Hashin failure modes described above are used to calculate threshold values, that once surpassed, initiate damage within the material model. Damage parameters are used in the plastic region of the material to soften the response of each orthogonal direction independently, representing increasing amounts of damage. The stress-strain relation for the MLT model is defined as

\[
\begin{bmatrix}
\sigma_{11} \\
\sigma_{22} \\
\tau_{12}
\end{bmatrix} = \frac{1}{D} \begin{bmatrix}
(1 - \omega_{11})E_1 & (1 - \omega_{11})(1 - \omega_{22})\nu_{21}E_2 & 0 \\
(1 - \omega_{11})(1 - \omega_{22})\nu_{12}E_1 & (1 - \omega_{22})E_2 & 0 \\
0 & 0 & D(1 - \omega_{12})G_{12}
\end{bmatrix} \begin{bmatrix}
\varepsilon_{11} \\
\varepsilon_{22} \\
\gamma_{12}
\end{bmatrix}
\]

Where \( D = 1 - (1 - \omega_{11})(1 - \omega_{22})\nu_{12}\nu_{21} > 0 \) (4.9)

where \( \omega_{11}, \omega_{22}, \text{and} \omega_{12} \) are the damage parameters for the fiber direction, in-plane transverse direction, and shear respectively. These damage parameters are dependent upon the strain in the element and are effectively inactive during the elastic region of the material [94,95]. The damage parameters are defined as

\[
\omega_i = 1 - \exp \left[ -\frac{1}{m_i \exp(1)} \left( \frac{\epsilon_i}{\epsilon_f} \right)^{m_i} \right] \quad (4.10)
\]

where \( \epsilon_f \) is the nominal failure strain for that loading condition (tension or compression in the fiber or matrix), \( \epsilon_i \) is the current strain in the damage direction, and \( m_i \) is the damage exponent calculated in LS Dyna as [96]

\[
m_i = \frac{1}{\ln \left( \frac{\epsilon_q}{\epsilon_f} \right)} \quad (4.11)
\]
where $\epsilon_q$ is the failure strain associated with the maximum stress (assuming a linear slope to failure).

The second failure definition for MAT 58 is the maximum effective strain for element failure. The parameter ERODS specifies the ultimate strain applicable to all directions of an element. Once this strain is attained, the element is deleted from the simulation. This variable cooperates with the Hashin failure criterion and can be used to incorporate post failure load carrying capability.

The third and final failure definition for MAT 58 is through the TSIZE parameter. An element is deleted if its associated time step is smaller than the value provided by TSIZE. This prevents time steps in explicit analyses from being dominated by a few erroneously small elements. TSIZE serves the same function as TFAIL within MAT 54.

MAT 58 has over 50 material parameters, and these include variables determined by material characterization and numerical constructs. Tables 4.3 and 4.4 split the inputs into two categories: dynamic properties and static properties for MAT 58.

The dynamic parameters for MAT 58 are similar to those for MAT 54, this is seen with the inclusion of common physical properties and the SLIM variables. However, there is a slight difference in the strain definitions for the 2 material cards. As previously mentioned, MAT 54 utilizes the DFAIL parameters to describe ultimate strain for each direction. MAT 58, on the other hand, uses the E variables (E11C, E11T, E22C, and E22T) to define the strain at each associated strength value, and then implements a parameter, ERODS, to represent the maximum strain for the element that leads to ultimate failure and deletion. The E strains are user-defined because MAT 58 does not incorporate a linear slope to initial strength failure. Instead, the model softens the material prior to failure because of increasing damage, and the E variables define the strain associated with strength after the stress-strain curve has gone nonlinear.

The static parameters associated with MAT 58 are primarily related to strain rate dependencies for physical properties. MAT 54 only provides options for strength
Table 4.3. LS Dyna MAT 58 inputs - Dynamic Parameters

<table>
<thead>
<tr>
<th>Variable</th>
<th>Definition</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>MID</td>
<td>Material identification number</td>
<td></td>
</tr>
<tr>
<td>RO</td>
<td>Mass density</td>
<td></td>
</tr>
<tr>
<td>EA EB EC</td>
<td>Young’s moduli</td>
<td>Fiber, transverse, and normal</td>
</tr>
<tr>
<td>PRBA PRCA PRCB</td>
<td>Poisson’s ratios</td>
<td>2-1, 3-1, and 3-2 directions</td>
</tr>
<tr>
<td>GAB GBC GCA</td>
<td>Shear moduli</td>
<td>1-2, 2-3, and 3-1 directions</td>
</tr>
<tr>
<td>SLIMT1</td>
<td>Factor for min stress limit after stress max</td>
<td>Fiber tension</td>
</tr>
<tr>
<td>SLIMC1</td>
<td>Factor for min stress limit after stress max</td>
<td>Fiber compression</td>
</tr>
<tr>
<td>SLIMT2</td>
<td>Factor for min stress limit after stress max</td>
<td>Matrix tension</td>
</tr>
<tr>
<td>SLIMC2</td>
<td>Factor for min stress limit after stress max</td>
<td>Matrix compression</td>
</tr>
<tr>
<td>SLIMS</td>
<td>Factor for min stress limit after stress max</td>
<td>Shear</td>
</tr>
<tr>
<td>AOPT</td>
<td>Material axes option</td>
<td>Suggest AOPT=2</td>
</tr>
<tr>
<td>TSIZE</td>
<td>Time step for element deletion</td>
<td>TSIZE&gt;0 for crash front to work</td>
</tr>
<tr>
<td>ERODS</td>
<td>Max effective strain for element layer failure</td>
<td></td>
</tr>
<tr>
<td>SOFT</td>
<td>Strength softening reduction factor</td>
<td>Applies to elements in crashfront</td>
</tr>
<tr>
<td>A1 A2 A3</td>
<td>Define vector a for AOPT=2</td>
<td>Defines fiber direction</td>
</tr>
<tr>
<td>D1 D2 D3</td>
<td>Define vector d for AOPT=2</td>
<td>Defines in-plane transverse direction</td>
</tr>
<tr>
<td>E11C</td>
<td>Strain at longitudinal compressive strength</td>
<td>Fiber direction</td>
</tr>
<tr>
<td>E11T</td>
<td>Strain at longitudinal tensile strength</td>
<td>Fiber direction</td>
</tr>
<tr>
<td>E22C</td>
<td>Strain at transverse compressive strength</td>
<td></td>
</tr>
<tr>
<td>E22T</td>
<td>Strain at transverse tensile strength</td>
<td></td>
</tr>
<tr>
<td>GMS</td>
<td>Shear strain at shear strength</td>
<td></td>
</tr>
<tr>
<td>XC</td>
<td>Longitudinal compressive strength</td>
<td>Fiber direction</td>
</tr>
<tr>
<td>XT</td>
<td>Longitudinal tensile strength</td>
<td>Fiber direction</td>
</tr>
<tr>
<td>YC</td>
<td>Transverse compressive strength</td>
<td></td>
</tr>
<tr>
<td>YT</td>
<td>Transverse tensile strength</td>
<td></td>
</tr>
<tr>
<td>SC</td>
<td>Shear strength</td>
<td></td>
</tr>
</tbody>
</table>

versus strain rate data, while MAT 58 includes variables for strength and failure strain versus strain rate data. However, the incorporation of the failure strain dependence on strain rate is solely because the model is designed to soften the material response prior to initial failure.

Figure 4.2 provides stress-strain curves of the 1 and 2 directions of a composite material under uniaxial loading represented by LS Dyna’s MAT 58. The slope of the curve is initially defined by the associated Young’s modulus. If the material model followed a linear path to failure, the failure strain would be at e1C-Y, e1T-Y, e2C-
Table 4.4. LS Dyna MAT 58 inputs - Static Parameters

<table>
<thead>
<tr>
<th>Variable</th>
<th>Definition</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>TAU1</td>
<td>Stress limit of nonlinear part of shear stress-strain curve</td>
<td>Only for FS=-1, not used</td>
</tr>
<tr>
<td>GAMMA1</td>
<td>Strain limit of nonlinear part of shear stress-strain curve</td>
<td>Only for FS=-1, not used</td>
</tr>
<tr>
<td>FS</td>
<td>Failure surface type</td>
<td>Suggest FS=1.0</td>
</tr>
<tr>
<td>EPSF</td>
<td>Damage initiation transverse shear strain</td>
<td></td>
</tr>
<tr>
<td>EPSR</td>
<td>Final rupture transverse shear strain</td>
<td></td>
</tr>
<tr>
<td>TSMD</td>
<td>Transverse shear max damage</td>
<td>Keep at default</td>
</tr>
<tr>
<td>XP YP ZP</td>
<td>Define point ( p ) for ( AOPT=1 )</td>
<td></td>
</tr>
<tr>
<td>V1 V2 V3</td>
<td>Define vector ( v ) for ( AOPT=3 )</td>
<td></td>
</tr>
<tr>
<td>BETA</td>
<td>Material angle for ( AOPT=0 ) and ( AOPT=3 )</td>
<td></td>
</tr>
<tr>
<td>LCXC</td>
<td>Load curve ID for ( XC ) vs strain rate</td>
<td></td>
</tr>
<tr>
<td>LCXT</td>
<td>Load curve ID for ( XT ) vs strain rate</td>
<td></td>
</tr>
<tr>
<td>LCYC</td>
<td>Load curve ID for ( YC ) vs strain rate</td>
<td></td>
</tr>
<tr>
<td>LCYT</td>
<td>Load curve ID for ( YT ) vs strain rate</td>
<td></td>
</tr>
<tr>
<td>LCSC</td>
<td>Load curve ID for ( SC ) vs strain rate</td>
<td></td>
</tr>
<tr>
<td>LCSC</td>
<td>Load curve ID for ( SC ) vs strain rate</td>
<td></td>
</tr>
<tr>
<td>LCTAU</td>
<td>Load curve ID for ( TAU1 ) vs strain rate</td>
<td></td>
</tr>
<tr>
<td>LCGAM</td>
<td>Load curve ID for GAMMA1 vs strain rate</td>
<td></td>
</tr>
<tr>
<td>DT</td>
<td>Strain rate averaging option</td>
<td></td>
</tr>
<tr>
<td>LCE11C</td>
<td>Load curve ID for ( E11C ) vs strain rate</td>
<td></td>
</tr>
<tr>
<td>LCE11T</td>
<td>Load curve ID for ( E11T ) vs strain rate</td>
<td></td>
</tr>
<tr>
<td>LCE22C</td>
<td>Load curve ID for ( E22C ) vs strain rate</td>
<td></td>
</tr>
<tr>
<td>LCE22T</td>
<td>Load curve ID for ( E22T ) vs strain rate</td>
<td></td>
</tr>
<tr>
<td>LCGMS</td>
<td>Load curve ID for ( GMS ) vs strain rate</td>
<td></td>
</tr>
</tbody>
</table>

Y, or \( e2T-Y \). Instead, the model lends itself to non-linear behavior associated with increasing damage in the material, and the failure strain is defined by \( E11C \), \( E11T \), \( E22C \), or \( E22T \). In addition, MAT 58 provides a gradual unloading post failure, followed by a constant stress determined by a SLIM parameter multiplied by the related strength. Ultimate failure is then defined by the ERODS variable.

While MAT 58 incorporates a better alternative to MAT 54’s unloading behavior post initial failure, it is not perfectly representative of what physically happens. Work by Xiao et al. [97–100] found that when using MAT 58 for the simulation of a crush tube, the stress unloading resulted in an under prediction of energy absorption. The group modified the CODAM model [101], a continuum damage mechanics model created at the University of British Columbia, in order to capture the stress unloading
behavior seen experimentally. This new model captures the unloading behavior of the material at high strains and produces energy absorption results closer to experiment [102].
Figure 4.2. MAT 58 Behavior in fiber and matrix directions
4.3 Modeling Methods for Composites in Crash Using LS Dyna

Researchers cannot agree on general best practices to model composites in crash. This is largely due to the complexity of the problem, and the fact that no modeling method can perfectly capture the load displacement curve and failure morphology while maintaining a reasonable analysis time. Typical differences in modeling techniques are seen in element formulations, contact definitions, and post processing of the data. One thing researchers can agree on is the use of material characterization and experiment to inform the material models. However, researchers will often make concessions on material properties to get the simulation results to match closer to experiment.

4.3.1 Element Formulations

Most composite damage material models within LS Dyna are only applicable for shell elements because of plane stress assumptions. While shell elements have a faster run time than solid elements, they cannot provide through thickness information. This makes capturing delamination failure morphology with a single layer of shell elements impossible. As shell elements have no real thickness, through thickness stresses are not calculated.

The two most common shell elements used in crash simulations for LS Dyna are the Belytschko-Lin-Tsay shell formulation, ELFORM=2, and the fully-integrated shell formulation, ELFORM=16. Both of these elements are based on the Reissner-Mindlin kinematic assumption [103], which incorporates through thickness shear deformations into the element formulations. The Belytschko-Lin-Tsay element is computationally efficient because it is based on a co-rotational and velocity-strain formulation that reduces the number of calculations necessary for each element [90]. This element is the default for shells in explicit analyses. The fully-integrated shell uses a similar co-rotational formulation to that of the Belytschko-Lin-Tsay element but is 2-3 times as expensive. This element is recommended for implicit solutions in LS Dyna [103].
LS Dyna introduced thick shell element types, TSHELLs, as a computationally efficient alternative to solid elements for bending load cases. These elements take thickness changes from loading conditions into account and have neutral planes that can move depending on the deformation. TSHELL elements are categorized by thin-thick shells and thick-thick shells. Thin-thick shells are extruded shell elements, and they still use the plane-stress assumption. Any thickness changes in the element are based solely on Poisson’s ratio, and bending can be captured with a single layer of elements. Thick-thick shells are 8-node brick elements that use a 3D based constitutive law. At least 2 thick-thick elements are required to capture bending to avoid a response that is too soft [88].

Shi proposed a new element type for modeling crush tubes consisting of layers of thin-thick shells connected by beam elements [104]. This method was able to capture through thickness stiffness values, capture delamination, and reduce bifurcation instabilities common with shell models in uniaxial compression. However, this proposal added 5 new elements to every shell element initially modeled. This increased the computational time significantly.

Boria et al. chose to model with solid elements because of the presence of large out of plane shear stresses [105]. They also modeled a layer of cohesive elements in their laminate to capture the splaying mode if it occurred. In the end, their model with solid elements gave similar results to their model with shell elements and increased the computation time drastically.

Component simulations can take several hours to run, and the ultimate goal is to incorporate the components into full scale assembly models. For that reason, this work utilized the Belytschko-Lin-Tsay element type because of its computational efficiency. While this element type cannot capture delamination with a single layer, it is still capable of obtaining load displacement results comparable to those seen in experiment. In addition, these elements are adequate in alternative loading conditions like bending and tension, which will be described in detail further on.
4.3.2 Time Step Calculations

In an explicit analysis, the time step of the simulation is determined by the size of the elements in the mesh and properties of the media through which the load is traveling. The smaller the time step, the longer the analysis takes to get to a specific end time. However, a smaller time step also increases fidelity in the model results. A simulation will generally become unstable if the information passing between elements is moving fast enough to leap elements in a single time step. LS Dyna calculates the time step for shell elements using the following equation

\[ \Delta t_c = \frac{L_s}{c} = \frac{L_s}{\sqrt{\frac{E}{\rho(1-\nu^2)}}} \]  \hspace{1cm} (4.12)

In this case, \( L_s \) is the longest side of the element, and \( c \) is the speed of sound through the medium that is in itself dependent upon the stiffness, density, and Poisson’s ratio of the material. Increases in the stiffness of the material or decreases in the size of the elements in the mesh will result in a reduced time step for the simulation.

4.3.3 Contact Definitions

The method in which subsections of the finite element model interact with each other determines the associated forces in the model. Contact methods in LS Dyna are split into 3 categories: the kinematic constraint method, the distributed parameter method, and the penalty method. The kinematic constraint method is primarily used to tie two meshes together, while the distributed parameter method is only used for sliding between surfaces. The penalty method works for general contact, sliding, separation, and friction [90,106]. As this work models individual components in crash loading conditions, it was not necessary to use any kinematic constraints or distributed parameter methods.
Penalty Stiffness Formulations

The penalty method places springs between two meshes when penetration has occurred. The spring stiffness is calculated using 1 of 3 available penalty algorithms: the standard penalty formulation (SOFT=0), the soft constraint penalty formulation (SOFT=1), and the segment-based formulation (SOFT=2). The standard penalty formulation and the soft constraint penalty formulation are both standard contacts that activate penalty forces when nodes of one mesh have penetrated segments of another mesh. However, the segment-based formulation activates when one segment of a mesh penetrates a segment of another mesh.

In the standard penalty formulation, the spring stiffness is calculated as the minimum stiffness of either the master or slave segment in the contact definition.

\[ k_{\text{soft}=0} = \min\{k_m, k_s\} \quad (4.13) \]

For shell elements these stiffnesses are calculated as

\[ k_m = SLSFAC \cdot SFM \frac{K_m A_m}{d_{\text{max},m}} \quad (4.14) \]
\[ k_s = SLSFAC \cdot SFS \frac{K_s A_s}{d_{\text{max},s}} \quad (4.15) \]

where SLSFAC is a user-defined scale factor for sliding interface penalties, SFM and SFS are user-defined scale factors for the default master and slave penalty stiffnesses, \( K_m \) and \( K_s \) are the bulk moduli of the elements, \( A_m \) and \( A_s \) are the face areas of the master and slave elements, and \( d_{\text{max},m} \) and \( d_{\text{max},s} \) are the max length of the element diagonals.

The soft constraint penalty formulation is used for soft materials that will drastically reduce the interface stiffness calculated with the standard penalty formulation. In this case, a stability contact stiffness is calculated with

\[ k_{cs} = 0.5 \cdot SOFSCL \cdot \max(m_s, \frac{1}{2} m_1 + m_2 + m_3 + m_4) \left( \frac{1}{\Delta t_c} \right)^2 \quad (4.16) \]
where SOFSCL is a user-defined scale factor for the soft constraint penalty formulation, \( m_s \) is the mass of the slave node penetrating the master, the additional \( m_i \) variables are the masses of the master nodes, and \( \Delta t_c \) is the current time step.

The final interface spring stiffness is defined as the max stiffness of either the stability contact stiffness or the standard penalty stiffness.

\[
k_{soft=1} = \max\{k_{cs}, k_{soft=0}\}
\] (4.17)

The segment based penalty formulation is considered the most stable penalty definition, but it is also the most computationally expensive. It is calculated in a similar manner to the soft constraint penalty formulation but the stability contact stiffness is defined as

\[
k_{cs} = 0.5 \cdot SLSFAC \cdot \begin{cases} SFS \\ or \\ SFM \end{cases} \left( \frac{m_1 m_2}{m_1 + m_2} \right) \left( \frac{1}{\Delta t_c} \right)^2
\] (4.18)

where the masses, \( m_i \) are the segment masses (equal to the element masses for shells).

Adjusting scaling factors for penalty stiffness definitions can be risky because a stiffness that is too high can result in force oscillations and instability in the model. A penalty stiffness that is too low can cause large penetrations to occur and then the model can fail to simulate the correct contact.

Feraboli et al. found [107, 108] that the *CONTACT_ENTITY keyword allows the user to define a load-penetration curve. Instead of the provided penalty stiffness formulations, the user can specify the amount of load applied to the contacting surfaces based on the amount of penetration that has occurred. However, this contact method defines a master rigid geometric entity to contact a slave deformable body. The master rigid body is limited to a few predefined geometric shapes. Therefore, this contact definition might be difficult to utilize in a assembly model, unless said model incorporates a car getting hit by a perfect cylinder or similar shape.
The node based contact formulations are faster than the segment based formulation but are more likely to completely miss contacts and have instabilities. In this work, the segment based penalty formulation (SOFT=2) was used, and the scaling factors including SFS, SFM, and SLSFAC were all kept at their default values.

### 4.3.4 Post Processing Methods

Results in LS Dyna are not output at every time step. Instead, the user defines an individual time increment for each output requested. A small output time increment helps prevent aliasing of results, which is a concern in explicit analyses that contain oscillating results.

As there are so many different contact definitions available in LS Dyna, the output variable for force results depends on the contact type used. RCFORC, RWFORC, and GCEOUT are all force output variables that were tested in this work. RCFORC provides the resultant interface forces between two contact surfaces and is the variable of choice when using any of the *AUTOMATIC contact definitions like *AUTOMATIC_SINGLE_SURFACE. The RCFORC output file produces force components in the global x, y, and z directions for both the slave and the master surfaces. The RWFORC parameter appears when the *RIGIDWALL contact is used. *RIGIDWALL creates a master rigid surface that contacts a user-defined deformable body. The RWFORC variable supplies the rigid wall force, and the output file contains the global x, y, and z force components as well as the calculated normal force on the wall. The GCEOUT output is produced when *CONTACT_ENTITY is used, and it produces the x, y, and z force components between the geometric entity and the defined deformable body.

Once the output is obtained, the data is usually filtered to smooth out oscillations and noise in the results, but there are not any standard filter specifications for crash simulations among researchers. Wang et al. [109] used a 100 Hz 3dB filter because of project guidelines, while Feraboli et al [107] tested out various frequencies of SAE
filters. Individual companies may have recommendations or guidelines, but they were not found in literature.

4.3.5 Material Characterization

As previously mentioned, all researchers use material characterization results to inform their material models within LS Dyna. Recommended test methods for common mechanical properties of composites can be found in the book by Carlsson et al. [110] or in ASTM standards. However, there are no pre-defined test methods for determining softening parameters and post-failure performance properties. DeTeresa et al. [111] and O'Higgins et al. [112] produced papers relating to parameter identification in support of simulating failure in composites. However, these papers do not provide an framework for future studies so much as report their own experimental results for a given material system.

Due to the difficulties associated with modeling progressive failure of composites, researchers will sometimes alter material properties to obtain simulation results that better match experiment. An example of this can be seen in work by Mamalis et al. [113] when the parameter TFAIL was set to a value of 0.8 seconds causing element deletion before material failure occurs. The author justified this by stating the TFAIL parameter could be used to keep the material behavior brittle. Typical values of TFAIL are on the order of 1e-8 seconds [107] for a comparison.

Reuter et al. [114] provided another example of manipulating material properties when they set their ultimate compression strain for fibers to 65%. This would result in a ductile behavior of the elements and allow them to carry load well past failure strain seen experimentally. A similar situation was seen when Stalin et al. used an ultimate strain of 50% [115]. Unfortunately, it is difficult to provide all material, contact, and post-processing parameters in a journal paper. Therefore, it is often challenging to recreate reported results from LS Dyna.
4.3.6 Modeling PPMC in Crash

As PPMC is a relatively new material system, nothing has been published regarding the crash simulation of the material. The closest work found was published by DorMohammadi et al. in 2018, and focused on a multi-scale modeling method for crush simulation of chopped fiber composites [116]. However, they utilized a user-defined material model without providing details of the model or the material system itself.

This work aims to fill gaps left in literature by creating a modeling method for PPMC in crash simulations, as well as providing a procedure for material characterization, model creation, and post-processing for PPMC components in crash. Ideally, this modeling methodology could be applied to single components as well as assemblies made with PPMC or any composite material with flow induced variabilities in orientation.
5. UNIDIRECTIONAL MODELS IN LS DYNAX

The simulation methodology for capturing crash performance of PPMC presented in this work takes a building block approach, where confidence in simulations is established at each block and model complexity increases with block level. First, the unidirectional carbon fiber prepreg with VORAFUSE M6400$^{TM}$ is characterized and incorporated into LS Dyna material cards. Then, the behavior of the material is confirmed with single-element models and compared to the closed-form definitions of the material models. The progressive failure of composites in LS Dyna is studied with 4-element models. Once understood, unidirectional models are created in LS Dyna for flexure and crush loading conditions, and the results are then verified against analytical solutions and validated against experiment. These working simulations serve as baselines for modeling PPMC components. Ensuring functionality of the baseline model provides confidence in future analyses that add further complexity, like platelet orientations for modeling PPMC. This chapter will review the creation of the LS Dyna material cards and the baseline component models that are utilized for PPMC modeling later on.

5.1 Creation of the MAT 54 and MAT 58 Unidirectional Material Models

5.1.1 Experimental Material Characterization

Dow Chemical provided a roll of unidirectional prepreg that serves as the base for the PPMC sheets. Several flat plaques were molded to be used for material characterization of the UD material. Table 5.1 provides the material properties measured in lab along with the ASTM standards used for testing.
### Table 5.1. VORAFUSE M6400 TM material characterization

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Std. Dev</th>
<th>Test Standard</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Stiffness</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E1 [GPa] (Tension)</td>
<td>141.7</td>
<td>1.54</td>
<td>ASTM D3039</td>
</tr>
<tr>
<td>E2 [GPa] (Tension)</td>
<td>7.44</td>
<td>0.25</td>
<td>ASTM D3039</td>
</tr>
<tr>
<td>E1 [GPa] (Compression)</td>
<td>96.2</td>
<td>1.09</td>
<td>Modified ASTM D695</td>
</tr>
<tr>
<td>E2 [GPa] (Compression)</td>
<td>12.5</td>
<td>0.92</td>
<td>Modified ASTM D695</td>
</tr>
<tr>
<td>G12 [GPa]</td>
<td>5.73</td>
<td>0.53</td>
<td>ASTM D3518</td>
</tr>
<tr>
<td><strong>Strength</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>XT [MPa]</td>
<td>1654</td>
<td>17.8</td>
<td>ASTM D3039</td>
</tr>
<tr>
<td>YT [MPa]</td>
<td>58.6</td>
<td>2.7</td>
<td>ASTM D3039</td>
</tr>
<tr>
<td>XC [MPa]</td>
<td>1113.5</td>
<td>82.61</td>
<td>Modified ASTM D695</td>
</tr>
<tr>
<td>YC [MPa]</td>
<td>234.6</td>
<td>9.07</td>
<td>Modified ASTM D695</td>
</tr>
<tr>
<td>S6 [MPa]</td>
<td>91.7</td>
<td>5.6</td>
<td>ASTM D3518</td>
</tr>
<tr>
<td><strong>Ultimate Strain</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>e1T-U</td>
<td>1.35%</td>
<td>0.04%</td>
<td>ASTM D3039</td>
</tr>
<tr>
<td>e2T-U</td>
<td>0.90%</td>
<td>0.0002%</td>
<td>ASTM D3039</td>
</tr>
<tr>
<td>e1C-U</td>
<td>-1.75%</td>
<td>0.15%</td>
<td>Modified ASTM D695</td>
</tr>
<tr>
<td>e2C-U</td>
<td>-1.88%</td>
<td>0.16%</td>
<td>Modified ASTM D695</td>
</tr>
<tr>
<td>eS-U</td>
<td>23.0%</td>
<td>2.00%</td>
<td>ASTM D3518</td>
</tr>
<tr>
<td><strong>Other</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>v12</td>
<td>0.32</td>
<td>0.01</td>
<td>ASTM D3039</td>
</tr>
<tr>
<td>ro [kg/m(^3)]</td>
<td>1800</td>
<td></td>
<td>From Product Data Sheet</td>
</tr>
</tbody>
</table>

It was immediately evident that the Young’s modulus was not the same in tension and compression for both the 1 and 2 directions. The E1 calculated from compression of 0° samples was around 96 GPa, which is considerably softer than the 141.7 GPa calculated in tension. In addition, the compressive response from the 90° samples was stiffer than the tensile response (12.5 GPa compared to 7.4 GPa) and was highly nonlinear. However, the material models in LS Dyna only allow for a single modulus in both tension and compression. The experimentally determined tensile moduli for 0° and 90° samples were used as the E1 and E2 values within the LS Dyna material cards.
The difference between the tensile and compressive moduli in conjunction with the other experimentally measured mechanical properties results in a singularity within the LS Dyna material models, seen in Figure 5.1.

![Stress-Strain Curves for MAT 54 and MAT 58](image)

**Figure 5.1.** MAT 54 versus MAT 58 material behavior with measured properties

As previously mentioned, the compressive modulus for the 2-direction was stiffer than the tensile modulus, but the tensile modulus was applied in the FEA simulations. Therefore, the compressive strength was achieved at a larger strain in the FEA simulations than what was seen experimentally. This caused a singularity in MAT 58 and a cross over phenomena in the stress-strain curve of MAT 54. In order to correct this, the ultimate compressive failure strain in the 2-direction was increased just enough to prevent the singularity. This increase in strain was from the lab measured -1.88% to -3.5%, which will be used in all future simulations. The finalized material behavior for MAT 54 and MAT 58 can be seen in Figure 5.2 and does not allow for load reduction after failure in compression of the 2-direction.
The failure strain in the 1-direction is larger for MAT 58 compared to MAT 54. This is because MAT 58 only has a single ultimate strain value defined for all loading conditions. Meanwhile, MAT 54 has 3 ultimate failure strains defined for fiber tension, fiber compression, and matrix tension/compression. The ultimate failure strain for MAT 58 is set to 3.5% to prevent singularities in the material model. This means MAT 58 will absorb more energy in the 1-direction than MAT 54 solely based on the material model definition.

![Graphs showing 1-Direction Behavior (Fiber) and 2-Direction Behavior (Matrix) for MAT 54 and MAT 58.](image)

**Figure 5.2.** Finalized MAT 54 and MAT 58 material models

The finalized material cards can be see in their respective LS Dyna formats in Figures 5.3 and 5.4. MAT 54 and 58 both have the capability to incorporate strain rate dependent strengths for the material system. However, the material system used in this research was not characterized at dynamic speeds. Therefore, any high speed simulations to represent crash behavior do not include experimentally determined speed-dependent properties. In addition, it should be noted that the $a$ and $d$ vectors
within the material cards are subject to change depending on the orientation of the geometry in the model.

Figure 5.3. MAT 54 in LS Dyna format

Figure 5.4. MAT 58 in LS Dyna format
5.1.2 Material Characterization Compared to Manufacturer Reported Properties

Late in the project, simulations with a high amount of axial fibers were proving to be excessively stiff when compared to experimental results. The characterized material properties were compared to a product data sheet for a similar material system, carbon fiber prepreg with VORAFUSE P6300\textsuperscript{TM}. Table 5.1.2 provides a direct comparison of the characterized properties and the reported properties from Dow. In general, the stiffnesses and strengths line up well for the different loading conditions. However, there was a noticeable discrepancy in the 0° tensile modulus as Dow reported 119 GPa, and the experimental results produced 141.7 GPa.

<table>
<thead>
<tr>
<th>Property</th>
<th>Product Data Sheet [117]</th>
<th>Experiment</th>
</tr>
</thead>
<tbody>
<tr>
<td>E1 (Tension) [GPa]</td>
<td>119</td>
<td>141.7</td>
</tr>
<tr>
<td>E2 (Tension) [GPa]</td>
<td>7.9</td>
<td>7.44</td>
</tr>
<tr>
<td>E1 (Compression) [GPa]</td>
<td>107</td>
<td>96.2</td>
</tr>
<tr>
<td>E2 (Compression) [GPa]</td>
<td>8.6</td>
<td>13</td>
</tr>
<tr>
<td>G12 [GPa]</td>
<td>3.5</td>
<td>5.73</td>
</tr>
<tr>
<td>XT [MPa]</td>
<td>1625</td>
<td>1654</td>
</tr>
<tr>
<td>YT [MPa]</td>
<td>57.2</td>
<td>58.6</td>
</tr>
<tr>
<td>XC [MPa]</td>
<td>1110</td>
<td>1113.5</td>
</tr>
<tr>
<td>YC [MPa]</td>
<td>200</td>
<td>234.6</td>
</tr>
<tr>
<td>S6 [MPa]</td>
<td>66.8</td>
<td>91.7</td>
</tr>
</tbody>
</table>

The large difference between the published E1 and the characterized modulus affects the simulation results and could be responsible for the simulations not comparing well with experiment in cases with a high percentage of axial fibers. As this problem was discovered late in the project, it was not practical to rerun all of the
simulations with the lowered tensile modulus. When convenient, two models were run to compare to experiment: one with the experimentally characterized properties and one with the E1 adjusted down to 119 GPa. The other material properties were kept the same for all simulations. The results of simulations with the adjusted tensile modulus will be discussed in future sections.

5.1.3 Single-Element and Four-Element Models

Single element simulations were created in LS Dyna to confirm the material models behaved as expected when compared to the closed-form solutions. A single shell element was created and constrained to in-plane motion. In addition, a pin constraint was placed in one corner and a roller constraint was placed in an adjacent corner. This allowed the element to expand and contract in both planar directions while preventing translation of the element as a whole. Figure 5.5 provides a visual of the boundary conditions for the single element model.

![Figure 5.5. Single-element model boundary conditions](image)

A constant-velocity load was applied to the positive X-face of the element until element deletion was achieved. The simulation end time was 0.5 seconds, and the
load and displacement values were output every 5E-4 seconds. This simulation was repeated for the element in tension and compression for both 0° and 90° orientations. The stress-strain plots of the element were calculated, compiled, and compared to the closed form solution. Figures 5.6 and 5.7 plot the closed-form solution versus a sample of output data points from LS Dyna. It is evident that the LS Dyna single element models successfully captured the MAT 54 and MAT 58 material models for the supplied material parameters.

Figure 5.6. MAT 54 single-element stress-strain results compared to closed-form solution

After the material behavior was confirmed, a four-element model was created to investigate failure propagation in compression loading within LS Dyna. Progressive failure is achieved when a single element or section of the model is deleted due to any failure mechanism. If the crashfront feature is activated (the TFAIL parameter is greater than 0), all elements previously touching the deleted element(s) will have their strength reduced to a value of $SOFT \cdot STRENGTH$. The reduction in strength is used to simulate the progression of damage to neighboring elements. Under a constant
loading condition, the elements with reduced strength will fail before the unaffected elements, and the cycle will continue.

The four-element model, seen in Figure 5.8, was created with shell elements extruded along the X-axis. The first element was pinned in one corner and a roller was applied to an adjacent corner to prevent translation of the model. In addition, all elements were constrained to in-plane motion. The MAT 58 model with the previously defined material parameters was applied to the system. Unidirectional continuous fibers were aligned along the X-axis for all elements. The elements had dimensions of 1 mm x 1 mm x 0.1 mm, however the fourth element was defined with half the thickness (0.05 mm) of the other elements.

The compressive load was applied via a rigid wall moving with a constant velocity of 12.6 mm/s along the negative X-axis. With this setup, the thinner, fourth element will fail first initiating the damage propagation. A SOFT parameter of 0.5 will be applied to the adjacent (third) element causing it to fail at half the defined compressive strength, and the cycle will continue until no elements remain. A global
system damping coefficient of 0.1 was applied to the model, and a Rayleigh damping coefficient for stiffness-weighted damping was set to 0.1. The reaction force from the rigid wall was output every 1E-5 seconds, and the simulation termination time was 0.3 seconds.

Figure 5.9 provides the strain-stress curve of the four element model during progressive failure. The stress was calculated using the rigid wall force along with the cross sectional area of the full thickness elements. The plotted stress is the absolute value of stress as the model was in compression and produced a negative number. The strain was calculated using the initial length of the entire model and the total displacement as the change in length.

There are 4 peaks in stress, each followed by an immediate drop to a zero stress state. Zooming in on a single peak reveals the shape of the MAT 58 material model curve. The peak stress reached 550 MPa before unloading 20%. After maintaining the lower stress for a finite amount of strain, the element was deleted dropping the stress to 0 MPa. The peak stress achieved corresponds to half of the characterized compressive strength in the fiber direction. This makes sense because a 0.5 softening factor was applied to elements 1-3. The cross sectional area used to calculate the stress was from a full thickness element, however the fourth element (first element to
fail) had half the defined thickness of the others. Therefore the stress in the fourth element would have been the full 1113 MPa when it failed.
5.2 Element Deletion in FEA with Brittle Materials in Compression

The four element model reveals a major issue engineers face when simulating compressive failure of brittle materials: element deletion and the subsequent release of stored energy. When metal crush tubes are simulated, the ductile nature of the material prevents rows of elements from being deleted simultaneously. There is no point in the simulation when the rigid wall loses contact with elements, and therefore, the compressed energy is never released. However, when brittle materials are modeled in crush, full rows of elements will delete at the same time. This results in the rigid wall no longer being in contact with the mesh, and the stored energy is released resulting in a stress wave propagating throughout the structure. Depending on the tensile strength of the material, this can cause the structure to tear itself apart. Once the row of elements is deleted, there is no force on the structure until the rigid wall, moving at a constant velocity, reaches the next row of elements. Figure 5.10 provides a visual of the described phenomena. The element deletion results in intermittent peaks in force followed by periods of no loading in the response of the structure, and this was seen in the four-element model.

This phenomena is a direct result of discretization within the model. Using a finer mesh will not solve the problem, but it will shorten the amount of time spent at zero force because there is a smaller distance for the rigid wall to travel between elements. Use of an unstructured mesh prevents simultaneous element deletion, but it results in an uneven loading surface for the rigid wall. Therefore, the full cross sectional area of the structure will never be in contact with the rigid wall at once, resulting in a consistently lower reaction force.

The oscillations of the load displacement curve produced from the element deletion provide an underestimation of the energy absorption for the component. Greve et al. [87] identified the same problem, and therefore developed a new contact method to avoid the issue entirely. In this work, a mesh density study was performed to confirm that discretization is the origin of the saw-tooth load-displacement curve,
and a finite element script for a 1D model in compression was developed in order to test out alternative boundary conditions.

5.2.1 Mesh Density Study

A mesh density study was performed to see how reducing the size of elements affected the frequency of the peaks in the load-displacement curve. Three crush tube models were created with varying sizes of elements. The coarsest mesh had in-plane element dimensions of 2.4 mm x 2.4 mm, and the finest mesh had element dimensions of 0.6 mm x 0.6 mm. The crush tube had an inner diameter of 50.8 mm, and a thickness of 2.645 mm. The rest of the model settings for the tube are discussed in Section 5.4.1

Figure 5.11 plots the load-displacement curves for each of the models. As the mesh density increased, the frequency of peaks also increased. In fact, the distance between peaks was equal to the length of the elements. This is because the increase in load occurred when the rigid wall was in contact with a row of elements. Once
the row of elements failed and was deleted, the rigid wall continued advancing to
the location of the next row of elements. Ideally, as the limit of the size of the
elements approaches zero, the peaks in the load-displacement curve will cease to be
individual peaks and will merge together. When post-processing data with the saw-
tooth load-displacement curve, the peaks are identified and connected with a line to
better represent what would happen with a continuous system.

![Graph showing load-displacement results of mesh density study](image)

**Figure 5.11.** Load-displacement results of mesh density study showing
that frequency of peaks in load is related to element size

It is also clear in Figure 5.11 that the magnitude of the peaks in the load dis-
placement curve decrease as the mesh density increases. This is a problem unique to
explicit analyses. As the mesh density was increased, the model was more susceptible
to numerical instabilities from stress waves. A stress wave is produced when a row of
elements is deleted, and the rigid wall is no longer in contact with the tube. The tube
releases all of the energy it obtained during compression, and a stress wave propagates
through the tube. Depending on the amount of energy released, the size of the ele-
ments, and the speed of the wave, individual elements away from the crushing surface
can fail due to constructive interference of the waves. Remember, when an element fails, its neighbor is given a reduced strength equal to $SOFT \cdot STRENGTH$. This can cause a string of elements in the middle of the tube to delete over time leaving a hole in the side of tube. This issue propagates until complete tube failure is achieved. While this is happening, the rigid wall is still slowly impacting the tube and deleting rows of elements. The stable crushing load is reduced because the tube does not have its full cross sectional area due to missing elements. Figure 5.12 provides an example of failure propagation due to stress waves.

Figure 5.12. Numerical instability in model with increased mesh density

Generally this problem can be avoided with the right combination of mesh density, rigid wall speed, and SOFT parameter. This is why the development of a fully
functional baseline model is crucial when attempting to model a new material system using LS Dyna.

5.2.2 1D Model

A 1D finite element model framework was developed in Matlab to investigate alternatives to a constant-velocity boundary condition that results in the saw-tooth load-displacement curve. This framework does not incorporate dynamic effects like wave propagation as it is an implicit solver. Each step that adds further displacement is solved independently from the others, resulting in an over-simplified model from reality. However, the purpose of the model is not to capture all of the physical phenomena but to recreate the saw-tooth load-displacement curve seen in LS Dyna and find a viable alternative for the constant-velocity rigid wall.

The inputs for the user are split into mesh settings, material parameters, geometric definitions, and load definitions. The mesh settings include the number of elements for the model, the number of points for Gaussian integration, and the number of spatial dimensions and corresponding degrees of freedom (DOFs) for each element. Other than the number of elements, these settings remained constant throughout the investigation because they define the model to be one-dimensional.

The material parameters include the Young’s modulus along the axial direction and the strength in compression. If additional loading conditions were being tested, the associated material properties could easily be added to the Matlab code.

The geometric definitions include the inner and outer diameters of the hollow tube as well as the starting length of the elements. The cross sectional area is calculated from the inputs and is used for stress calculation later. Finally, the user applies the loading conditions by either defining a displacement or a point load to a specific node.

The framework was developed for static problems and therefore solves individual steps implicitly without accounting for accelerations in the model. However, plotting the results from a series of steps with increasing amounts of applied displacement...
gives the appearance of a time-dependent simulation even though no information is shared between the steps.

In order to achieve progressive failure, similar to what is seen in LS Dyna, the SOFT parameter was programmed into the finite element framework. The stress in each element is calculated after every step, and if an element has reached or surpassed its strength, it is deleted. Once an element is deleted, the neighboring element has its strength reduced by an amount designated by $STRENGTH \cdot SOFT$.

For this investigation, a hollow cylinder was modeled with 1D bar elements, only capable of resisting axial loads. Each element was modeled with 2 nodes, and all elements were defined with an initial length of 0.01 m. The tube was fixed on one end, corresponding with node 1 in the 1D model. Figure 5.13 provides an example of the node and element numbering for the 1D model. The inner diameter of the tube was set to 50.8 mm, and the outer diameter of the tube was 54.8 mm. The last element was defined with a reduced area to initiate progressive failure.

**Case 1: Constant-Velocity Rigid Wall**

The first model recreated the constant-velocity rigid wall that is captured in LS Dyna. The displacement was increased at a rate of 0.01 mm per step and was applied to the last element (element 5 in Figure 5.13). The subsequent displacement of the other elements was used to calculate the associated stresses. Once element 5 reached its strength value, it was deleted. The neighboring element’s (element 4) strength was then reduced by 50%. However, the neighboring element does not come into contact with the rigid wall until the total displacement of the wall equals the initial length of the element that just failed (element 5). Therefore, a considerable amount of time is spent without the model in contact with the rigid wall.

Figure 5.14 provides the resulting load displacement curve from the rigid wall as well as the displacement of the rigid wall associated with each step number. As the global displacement of the rigid wall increases, the reaction force increases until
the element with the reduced cross sectional area reaches its strength and is deleted. The reaction force immediately reduces to 0 kN as the wall is no longer in contact with any of the remaining elements. The process repeats itself once the rigid wall makes contact with the next element, which has a reduced strength from the SOFT parameter. This case results in a smooth curve for the displacement versus step number of the boundary condition, but a saw-tooth shape in the load-displacement curve.

**Case 2: Adjustable Boundary Condition**

The second case tested with the 1D framework incorporated a boundary condition that moves at a constant velocity until element deletion occurs. Once an element is
Figure 5.14. Load-displacement curve for 1D model with constant-velocity rigid wall

deleted, the rigid wall immediately jumps to make contact with the next element. This prevents the remaining elements from releasing the energy stored during compression. After the jump, the rigid wall continues at its previous velocity until the next element deletion occurs. This method drastically reduces the number of steps needed for analysis. However, in an explicit analysis there are sure to be inertial effects from the instantaneous hop by the rigid wall that are not captured with this rudimentary model.

Figure 5.15 provides a visual of the load-displacement curve as well as the displacement of the rigid wall versus the step number. The peaks of the load displacement curve for the constant-velocity rigid wall match the data points of the adjusted boundary condition. However, the adjusted boundary condition completes the process in under 30 steps moving while the constant-velocity rigid wall requires 4000 steps. The first test case had a linear relationship between displacement and step number, but the second test case does not.
Conclusions From 1D Model

The work on the 1D model provides further confirmation that the saw-tooth shaped load-displacement curve is solely a function of discretization within the finite element model. An alternative boundary condition where the rigid wall immediately moves to the next row of elements following element deletion was presented with promising results. In fact, if the peaks of the constant-velocity rigid wall test case were connected, the load displacement curve of the second test case would be created.

However, further study needs to be completed before this boundary condition should be implemented in large scale models. Inertial effects must be investigated to determine if the abrupt motion of the rigid wall causes problems.
5.3 Flexure Coupons

A unidirectional flexure model was created to investigate an additional loading condition also experienced in vehicular crash. The estimated flexural modulus and strength were compared to closed form estimates from classical laminate plate theory (CLPT) as well as experimental results.

5.3.1 Model Setup

Geometry

A baseline flexure model was created in LS Dyna to recreate the setup presented in ASTM D7264. Once the baseline model was functioning, a series of small investigations were completed to develop confidence in the model. Within this baseline model, the flexure sample was modeled to be 100 mm long and 13 mm wide. The pin supports were spaced 76.8 mm apart and had diameters of 6.3 mm. Figure 5.16 provides a schematic of the model setup with dimensions.

Figure 5.16. Dimensions for 3-point bend simulation

The flexure sample was meshed with type 2 shell elements with each element having in-plane dimensions of 0.7 mm x 0.7 mm. The bottom pins were meshed with
rigid, type 2 shell elements with dimensions of 1 mm x 1 mm. The top pin was modeled using the *RIGIDWALL,GEOMETRIC_CYLINDER_MOTION_DISPLAY keyword, which defines starting location and geometric dimensions of the cylinder, as well as the rate and direction of movement. Figure 5.17 provides a rendered image of the finite element model within LS Dyna. Note the beam and bottom pins are constructed of shell elements, while the top pin is comprised of solid elements. The top pin was not manually meshed but meshed as a product of the *RIGIDWALL keyword.

Figure 5.17. Graphical representation of LS Dyna finite element model for flexure

The flexure beam was unconstrained, but the two bottom pins were fixed in place. Two instances of *BOUNDARY_SPC_NODE were used to fix a single node on each of the bottom pins. This fixed the entire pin because each is comprised of rigid elements.

Control Keywords

There are several keywords within LS Dyna that control the specifics of the analysis, mesh definition, and solving method. The following settings listed all differ from the defaults within LS Dyna. Within the *CONTROL_SHELL keyword, the variable LAMSHT was set to 1 in order to activate laminate shell theory, and the variable ISTUPD was set to 1 to ensure elements in high-strain situations have their
thickness reduced. In order to avoid an hourglass effect on elements, the *CONTROL\_HOURGLASS keyword was activated with the hourglass viscosity type set to 4 and the coefficient set to 0.1. This provides a stiffness-based hourglass control for shell elements. Orthotropic shells can be sensitive to shear and hourglass deformation depending on how the element coordinate system is defined [106]. Invariant node number for elements can minimize these effects and was activated for the mesh by setting the INN variable within *CONTROL\_ACCURACY to 2. Under the *CONTROL\_PARALLEL keyword, the CONST variable was set to 1. This consistency flag ensures components of global vectors are summed in the same order regardless of the number of processors used with the solver [88].

**Contact Definition**

The *CONTACT\_AUTOMATIC\_SINGLE\_SURFACE method was the contact definition of choice for the flexure simulation. This contact method is a two-sided contact that allows for contact between several parts as well as the possibility of self contact. For this type of contact, there is no defined master surface. Instead, all of the potential contact surfaces form a single, slave contact group. In this case, the flexure beam and two bottom pins formed the slave contact group. The contact keyword has variables that defines the contact between surfaces including friction, damping, and penetration.

The frictional coefficient is dependent upon the relative velocity between the surfaces and is defined as

\[
\mu_c = FD + (FS - FD)e^{DC|v_{rel}|}
\]  

(5.1)

where FD is the dynamic coefficient of friction, FS is the static coefficient of friction, DC is the exponential decay coefficient, and \(v_{rel}\) is the relative velocity between the surfaces. The static and dynamic coefficients of friction were both set to 0.08
using example LS Dyna models as a guideline. The exponential decay was set to 0 for these simulations.

The viscous damping coefficient, VDC, within the contact keyword is an integer from 0 to 100 that defines the damping of the contact as a percentage of the critical damping. This helps prevent severe oscillations within the contact itself and can be represented in the following form

\[
\xi = \frac{VDC}{100} \xi_{crit}
\]  

(5.2)

where \(\xi_{crit}\) is dependent upon the masses of the contacting parts as well as the interface stiffness between the two. Generally this value was left at or below 1% because damping was applied with a separate keyword.

The segment based contact method SOFT=2 was used, and all of the scaling factors were left at default values. The *CONTROL_CONTACT keyword was activated, but the default values were kept for these simulations.

**Damping**

Damping was applied to the setup in two forms. First mass weighted nodal damping was applied to all parts, both deformable and rigid, using the *DAMPING_GLOBAL keyword. The system damping constant VALDMP was set to 0.3. The *DAMPING_PART_STIFFNESS keyword was used to apply additional damping to the flexure beam itself. This provided stiffness weighted Rayleigh damping for the part, and the coefficient, COEF, was set to 0.1. The mass proportional damping supplied by the global damping card is better for low frequency oscillations while the stiffness proportional damping is better for higher frequencies [88]. The use of both types of damping was to cover the entire frequency range of the part.
Data Output

The primary history output requested for the flexure baseline model was the rigid wall reaction force, RWFORC. This variable records the reaction force on the top pin which is defined using a RIGIDWALL card. Within the *DATABASE_OPTION keyword, RWFORC was checked. The DT variable was set to 1e-08 sec, and that determined the time interval between the outputs. The output frequency for the stress and displacement results for the entire model was set in *DATABASE_BINARY_D3PLOT. For the baseline model this was set to 1.0E-5 seconds. The variable CMPFLG in the *DATABASE_EXTENT_BINARY keyword should be set to 1 in order to have stress and strain data outputted in the material coordinate system.

Section Definition

The section definition for the flexure beam defines the element type, thickness, and layup for the composite. The ICOMP property in *SECTION_SHELL was set to 1, which defines the in-plane fiber angle for a ply as an integration point. Therefore, the number of plies in a layup corresponds to the number of integration points through the shell. This setting only works for orthotropic material keywords like MAT 54 and MAT 58. Figure 5.18 provides a diagram of how the ply orientations are applied to a shell element. The NIP variable defines the number of integration points through the thickness of the shell. This is obviously dependent upon the layup defined for orthotropic materials. However, once the number of integration points exceeds 10 for NIP, a user-defined rule must be created. The user-defined rule is referenced with the QR/IRID variable in the *SECTION_SHELL keyword. A negative value of QR/IRID specifies the rule number for an *INTEGRATION_SHELL keyword. The IRID number within the integration keyword matches the absolute value of the QR/IRID variable from the section keyword. The NIP was set to the number of plies within the integration keyword, and the ESOP value was set to 1 to have the integration points equally spaced through the thickness of the shell element.
The keywords mentioned in this section provided the backbone for the baseline flexural model. The total beam thickness, layup, simulation end time, and speed for the top pin were not presented because they were all subject to change depending on the circumstances. All flexure simulations presented moving forward were produced from this baseline model.
Post Processing Flexure Models

Flexure simulations have 3 points of contact on the test specimen. Upon initial contact, energy is transferred from the moving pin to the stationary beam. The contact method defined in these simulations does not allow the surfaces to stick together after first contact. This can result in a bouncing effect, depending on the speed of the top pin. In addition, the energy imparted by the pin produces stress waves that travel along the beam. Even with damping incorporated into the model, these oscillations are clearly evident in the reaction force on the rigid pin. This is a problem inherent to explicit analyses, where accelerations are accounted for from step to step. An example rigidwall reaction force from a flexure simulation can be seen in Figure 5.19.

![Figure 5.19. Example load versus displacement curve for flexure model](image)

The unfiltered force has oscillations as soon as contact occurs, and the noise increases in magnitude as the displacement increases. A moving average filter was applied to the data in order to reduce the noise, and the result looks like a typical load-displacement curve for a 3-point bend test. Because of this, future data from
flexure simulations will be post processed with a moving average filter. The number of data points used in the filter is dependent upon the time step between outputs.

5.3.2 Impact Speed Study

The recommended test rate for flexure experiments is between 1 and 5 mm/min, or 1.6E-05 to 8.33E-05 m/s. While testing at quasi-static speeds is relatively simple, performing explicit finite element simulations at these speeds can take an excessive amount of time. Simulating faster test speeds can save computational time and is an easy adjustment within the model, but the results need to be representative of those from a quasi-static experiment.

Three models were created to investigate the effect impact speed has on the simulated flexural response of a crossply laminate. All of the models were created from the baseline detailed in the previous section, but the impact speed of the top pin was adjusted for each. The layup was set to [0/90]_4s, and 16 integration points were defined through the thickness of the shell elements. MAT 54 was selected as the material model, and the thickness of the beam was set to 2.5 mm creating a 0.16 mm ply thickness. The constant velocity defined for the top pins in the models was 4 m/s, 0.4 m/s, and 0.04 m/s. The total displacement of the top pin was defined as 10 mm for each simulation, and the end time in the analysis was 0.0025 s, 0.025 s, and 0.25 s respectively.

Figure 5.20 provides the unfiltered and filtered stress strain curves for the 3 models, and Table 5.3 shows the calculated flexural moduli, strengths, and computational times. The fastest simulation in processing time and impact speed had a bouncing interaction between the top pin and the beam. This is evident in the unfiltered response of the V = 4 m/s simulation when the stress initially increases and then drops to 0 MPa for a time. The time spent at zero stress was when the beam deflected a farther distance than the pin had reached. The filtered response for this simulation is not a smooth line, as the oscillations in the unfiltered data are severe.
Decreasing the speed of the top pin by an order of magnitude results in a similar flexural modulus, but an increase in strength. In addition, the magnitude of the peaks in the raw simulation data have lowered, but the oscillations have a higher frequency.

The slowest model produced the lowest flexural modulus and strength, however, all of the estimated moduli were within 5 GPa of each other. The simulation with an initial impact speed of 0.04 m/s took 10 times as long to run as the model with V=0.4 m/s while producing similar results. A 4,000 element model taking 11 hrs to run is impractical for industry when the component will be inserted into additional assembly models down the road. Decreasing the run time or simplifying the model are both options to improve the issue. In this case, the model is already relatively coarse, and for this reason, the remaining flexure models in this work were modeled with a top pin speed of 0.4 m/s.

<table>
<thead>
<tr>
<th>Simulation</th>
<th>$E_f$ [GPa]</th>
<th>$\sigma_f$ [MPa]</th>
<th>Computational Time [min]</th>
</tr>
</thead>
<tbody>
<tr>
<td>V = 4 m/s</td>
<td>87.5</td>
<td>706.6</td>
<td>5</td>
</tr>
<tr>
<td>V = 0.4 m/s</td>
<td>87.8</td>
<td>817.1</td>
<td>60</td>
</tr>
<tr>
<td>V = 0.04 m/s</td>
<td>83.1</td>
<td>668.6</td>
<td>660</td>
</tr>
</tbody>
</table>

### 5.3.3 Verification of Flexure Model

Verifying a simulation is necessary in order to develop confidence in the model. For the 3 point bend flexure model, verification was performed using analytical solutions presented in literature. Pagano [118] and Whitney [119] both wrote about the difficulty associated with analyzing composite laminates in flexure. Symmetric layups with 0° and 90° plies can be analyzed with classical beam theory, but the modulus weighted moment of inertia must be used.
The weighted moment of inertia is calculated using parallel axis theorem, and a reference elastic modulus. For a symmetric laminate, the modulus weighted centroid is in the center of the stacking sequence. The moment of inertia is calculated for each ply in relation to the laminate centroid using parallel axis theorem. The ply moment of inertia is then multiplied by a ratio of the axial modulus of the ply (in this case either $E_1$ or $E_2$) and a predefined reference modulus $E_R$. The modulus weighted moment of inertia for the entire layup is the sum of the weighted inertias for each ply and can be described with the equation below

$$I^* = \int_A \frac{E}{E_R} y^2 dA = \sum_{k=1}^n \frac{E_k}{E_R} I_k$$  \hspace{1cm} (5.3)
The level of anisotropy within the material system (the ratio of $E_1/E_2$) determines how much the moment of inertia will change once the layup is accounted for. The effect of the modulus ratio for the $[0/90]_{4s}$ layup on the moment of inertia can be seen in Figure 5.21. As the $E_1/E_2$ ratio increases, the modulus weighted moment of inertia increases up to 18% before leveling off. The material system in this research has a $E_1/E_2$ ratio around 19, which would increase moment of inertia by over 15% for this layup.

![Figure 5.21. The effect of modulus ratio on the moment of inertia for a $[0/90]_{4s}$ laminate](image)

The displacement of a beam in 3-point bending is calculated with the following equation for an isotropic material

$$\delta = \frac{PL^3}{48EI} \quad (5.4)$$

Where $P$ is the load from the top pin, $L$ is the span, $E$ is the Young’s modulus of the layup, and $I$ is the moment of inertia. In the case of a cross ply laminate
beam, the bending stiffness $EI$ should be replaced with the equivalent stiffness of the beam [119]. Using the new equation

$$\delta = \frac{PL^3}{48E_R I^*}$$

results in the closed-form load-displacement curve seen in Figure 5.22. As is evident from the plot, the LS Dyna flexure simulation using MAT 54 and a pin speed of $V = 0.4$ m/s matches the closed form solution using weighted moment of inertia.

![Figure 5.22. Load-displacement results of cross ply laminate in flexure versus analytical solution](image)

Pagano [118] produced additional equations for the calculation of flexural modulus for cross ply laminates that are dependent upon the Young’s moduli of the material and the stacking sequence of the laminate. In this case, the flexural modulus is defined as
\[ E' = \frac{F}{N^3} \quad (5.6) \]

Where \( F = \begin{cases} 
8 \sum_{i=1}^{N} E_i(3i^2 - 3i + 1), & \text{N even} \\
E_1 + 2 \sum_{i=2}^{N+1} E_i(12i^2 - 24i + 13), & \text{N odd}
\end{cases} \quad (5.7) \]

In these equations, \( N \) is the number of plies in the symmetric layup and \( E_i \) is the longitudinal modulus for the ply. In the case of a 0° ply, the \( E_i \) would be 140 GPa, while the \( E_i \) for a 90° ply would be 7.44 GPa.

The calculated flexural moduli using Equation 5.5, Pagano’s Equation 5.6, and the LS Dyna simulation can be seen in Table 5.4. The various methods predicted similar flexural moduli, providing further confidence in the LS Dyna flexure model.

Table 5.4. Estimated flexural modulus for [0/90]s layup from different analysis methods

<table>
<thead>
<tr>
<th>Method</th>
<th>( E_f ) [GPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Equation 5.5</td>
<td>86.13</td>
</tr>
<tr>
<td>Pagano’s [118] Equation</td>
<td>86.15</td>
</tr>
<tr>
<td>LS Dyna Simulation</td>
<td>85.97</td>
</tr>
</tbody>
</table>

Material Model Comparison

All of the simulation results presented thus far have used MAT 54. However, the results from MAT 58 also need to be compared to develop confidence in that material model. A flexure model with the layup [0/90]s, 2 mm thickness, and pin speed of \( V = 0.4 \text{ m/s} \) was created using MAT 58. The results were compared to those from a similar simulation using MAT 54 and can be seen in Figure 5.23. It is immediately evident that the model using MAT 58 fails at a higher strain than the model using MAT 54. This is to be expected because the MAT 58 material model has a higher failure strain in fiber tension and compression than MAT 54. The slopes of the models
are the same until about 0.6% strain when the slope of the MAT 58 filtered curve softens slightly. These results provide confidence that the two material cards perform similarly in terms of stiffness. Unfortunately, the differences in strength between the models will not disappear without adjusting the failure strain values of one of the material models.

![Stress-strain results of crossply laminate in flexure for MAT 54 and MAT 58](image)

Figure 5.23. Stress-strain results of crossply laminate in flexure for MAT 54 and MAT 58

### 5.3.4 Validation with Experiment

Now that the 3-point bend simulations have been verified with mathematical models, they need to be validated against experiment. If the material models properly represent the behavior of the material system, and the simulations capture the physical characteristics of the experiment, the results of the simulations should match the experimental measurements. In this case, the flexural modulus and strength of the simulations will be compared to their experimental counterparts.
A 10" x 10" UD plate with a [0/90]_{4s} layup was compression molded at 130° C with 200 psi. This plate was cut into sixteen 13 mm x 254 mm flexure samples. The samples were loaded at a rate of 2 mm/min until failure. Three different span to thickness ratios were tested to ensure shear effects were not lowering the estimated flexural modulus significantly. The span to thickness ratios tested were 25, 40, and 50 with spans of 70 mm, 110 mm, and 162 mm respectively. The load and displacement results were recorded at a frequency of 5 Hz, and a 5 kip MTS machine was used for the experiment. The apparent flexural modulus was calculated using the equation provided by Carlsson et al. [110]

\[
E_f = \frac{L^3 \Delta P}{4wh^3 \Delta \delta}
\]  \hspace{1cm} (5.8)

where \( L \) is the span (distance between the bottom pins), \( w \) is the sample width, and \( h \) is the height of the sample. The strength of the experimental samples was determined using the following equation

\[
\sigma_{max} = \frac{3PL}{2wh^2}
\]  \hspace{1cm} (5.9)

Table 5.5 lists the modulus, strength, and failure strain calculated for the experiments in relation to each of the span to thickness ratios. The highest values for modulus, strength, and ultimate strain occurred in the samples with a span to thickness ratio of 40. However, these values were not significantly far from the others.

<table>
<thead>
<tr>
<th>L/h</th>
<th>( E_f ) [GPa]</th>
<th>Std. Dev [GPa]</th>
<th>( \sigma_{max} ) [MPa]</th>
<th>Std. Dev [MPa]</th>
<th>Strain [%]</th>
<th>Std. Dev [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>50</td>
<td>63.93</td>
<td>5.52</td>
<td>847.04</td>
<td>103.34</td>
<td>1.33</td>
<td>0.19</td>
</tr>
<tr>
<td>40</td>
<td>65.81</td>
<td>3.49</td>
<td>937.81</td>
<td>33.89</td>
<td>1.43</td>
<td>0.06</td>
</tr>
<tr>
<td>25</td>
<td>60.74</td>
<td>2.11</td>
<td>872.03</td>
<td>154.45</td>
<td>1.43</td>
<td>0.22</td>
</tr>
</tbody>
</table>

Table 5.6 compares the measured experimental flexure properties against those calculated in simulations using MAT 54 and MAT 58. The average flexural modulus
for the experimental samples of all spans was 63.32 GPa with a standard deviation of 4.42 GPa. This is significantly lower than the LS Dyna predictions of 86 GPa. However, the strength of the LS Dyna MAT 54 simulation was only 15 MPa away from the experiment average. The MAT 58 simulation had a higher strength, but the estimated failure strain was almost identical to that measured in experiment.

The large difference in flexural modulus raises questions in regards to both the simulation and experiment. While the simulations have been verified with analytical models, verification only proves the simulation was solving the given problem correctly. If the simulation and analytical model were given incorrect inputs, such as material properties or stacking sequence, they would still produce similar results. The question then becomes: how well does the simulation represent the physical beam tested?

It is possible the fibers were not aligned well during manufacturing, and the samples do not actually represent a $[0/90]_{4s}$ laminate. However, after testing, a few samples were sectioned and polished for microscopy to confirm the orientation state. Figure 5.24 provides a microscopy image of a flexure sample and confirms the orientation of fibers within the plies. It is evident the plies were not uniform in thickness across the layup, and the simulation did not take this into account.

With the stacking sequence confirmed, the question regarding the quality of the material characterization comes to the forefront. As was discussed previously, the characterized tensile modulus for $0^\circ$ plies is 21 GPa higher than the modulus reported

<table>
<thead>
<tr>
<th></th>
<th>$E_f$ [GPa]</th>
<th>$\sigma_{max}$ [MPa]</th>
<th>Strain [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>63.32</td>
<td>807.9</td>
<td>1.4</td>
</tr>
<tr>
<td>Standard Deviation</td>
<td>4.42</td>
<td>118.4</td>
<td>0.19</td>
</tr>
<tr>
<td>LS Dyna MAT 54</td>
<td>85.97</td>
<td>822.0</td>
<td>1.02</td>
</tr>
<tr>
<td>LS Dyna MAT 58</td>
<td>85.18</td>
<td>984.3</td>
<td>1.395</td>
</tr>
</tbody>
</table>
by Dow. The flexural modulus of the simulation using the characterized E1 is off by 34% from the experiment. Figure 5.25 provides a visual of the simulation results for MAT 54 compared to the test results. The strength of the model using MAT 54 is on par with the experimental results, but the stiffness is obviously too high.

When the Dow-published tensile modulus is used in Pagano’s equation, Eq. 5.6, the flexural modulus is estimated to be 73.7 GPa instead of the previously reported 86.15 GPa. A second LS Dyna model was created that reduced the the Young’s modulus in the fiber direction down to 119 GPa. The resulting flexural modulus from the simulation was reduced to 72.0 GPa from the previous 85.97 GPa. The simulation results were once again comparable with the analytical model, verifying that the model is solving the appropriate problem. The simulated flexural modulus was still higher than the experiment, but now the difference is closer to 10% instead of 34%. Figure 5.26 shows the updated stress-strain curve for the flexure model using MAT 54. The strength decreased, but the ultimate strain was consistent. This indicates the model failed due to the strain limit before the strength value was reached. A lower stiffness in conjunction with the same strength values means the failure strains should
be larger. Therefore, the strength of the simulation could be fixed by increasing the failure strain in the material model.

5.3.5 Conclusions on Unidirectional Flexure Simulations

The LS Dyna flexure model was verified with analytical solutions and compared with experiment. Because the 0° plies were on the outside of the [0/90]₄s laminate, the E1 stiffness dominated the bending response. As previously mentioned, the material characterization performed on the UD prepreg gave a significantly higher 0° tensile modulus than the material properties provided by the manufacturer for a similar material system. Once the tensile modulus was adjusted and the model was run again, the simulation results were comparable to those of experiment. At this point,
Figure 5.26. Experimental flexure results versus simulation results with E1 adjusted to 119 GPa

the baseline flexure model is fully functional and only takes about an hour to run with 16 integration points.

5.4 Crush Tubes

The next loading condition investigated was compression. In this case, a baseline model was created to represent the experimental crush tubes tested and discussed in Chapter 3. The load-displacement curve and the specific energy absorption were the primary quantitative variables of interest. Matching experimental failure morphology was originally a priority but was dropped in favor of a faster analysis time.
5.4.1 Model Setup

Geometry

The crush tube was modeled as a hollow cylinder with a 100 mm length and a 50.8 mm inner diameter. The baseline model had a 54.8 mm outer diameter, so the thickness in the model was defined as 2 mm. The thickness is prone to change depending on the experimental sample the model is representing. However, the inner diameter remained constant for all crush tube models. A 45° bevel was machined into the experimental samples and represented using thickness variations within the LS Dyna model. A constant-velocity infinitely large rigid wall was used to apply the axial compressive force to the tube, and the top row of nodes on the tube were fixed. When the rigid wall makes contact with the tube, the fixed nodes prevent rigid body movement, and the tube is crushed. Figure 5.27 provides a schematic of the crush tube with boundary conditions.

![Figure 5.27. Geometry and boundary conditions for baseline crush tube simulation](image)
The tube was meshed with type 2 shell elements with in-plane dimensions of 2.4 mm x 2.4 mm. The mesh was uniform down the length of the tube, but the bottom two rows of elements were separated and assigned their own part definitions. The bottom row is assigned to the part named the “lower bevel”, and the next row up is assigned to the part named “upper bevel”. The rest of the mesh belongs to the part labeled “cylinder”. Figure 5.28 shows the full LS Dyna model for the crush tube. The lower bevel is yellow, the upper bevel is blue, and the cylinder is red.

The rigid wall is defined with the *RIGIDWALL,GEOMETRIC,FLAT,MOTION keyword. The user defines the initial starting coordinates, the size, and the displacement versus time curve for the flat plate. For the majority of crush tube models, the rigid wall was set to move at a speed of 0.32 m/s, and the plate was defined to be infinite in width and length.

Figure 5.28. Finite element model of crush tube with bevel built into bottom two rows of elements
A 45° bevel was machined into the experimental samples, but it was not possible to capture this with discrete shell elements. The lower bevel was given a 0.5 mm thickness, and the upper bevel had a 0.75 mm thickness defined. Even though there are separate thicknesses defined for the two layers of the bevel, they are connected by a shared node. This results in an unusual shape which can be seen in Figure 5.29. The simulated bevel does not create a 45° angle, but it serves its designated purpose as a failure initiator. The reduced cross sectional area of the bevel causes the bottom row of elements to fail first. This reduces the strength in the second row of elements by a designated value, and the process continues. It is not necessary to have two designated rows of elements for the bevel because the crash front forms once a single element has failed. The second row of elements for the bevel was included out of preference by the author.

![Machined Bevel vs Simulated Bevel](image)

**Figure 5.29.** Machined bevel for experimental samples versus simulated bevel shape within LS Dyna

While the geometry of the model is visually simple, the unique challenges associated with crushing a perfect cylinder made of a brittle material make the simulation quite difficult.
Control Keywords

Many of the control settings that were used in the baseline flexure model were also utilized in the crush tube model. This includes activating invariant node numbering, hourglass energy settings, and laminate shell theory.

In explicit analyses, the step solution builds off of the results of the previous step. Caution must be taken when solving these analyses because numerical errors can compound going from step to step. Double precision solvers should be used when possible to reduce the likelihood of roundoff errors. However, even when using a double precision solver, it is possible to run the same LS Dyna simulation multiple times and get drastically different results. A quote from the LS Dyna manual reveals this is a known problem for loading conditions that are being used in this research:

“The accuracy issues are not new and are inherent in numerical simulations of automotive crash and impact problems where structure bifurcations under compressive loads are common.” [90]

The parallel consistency flag within the keyword *CONTROL_PARALLEL is designed to reduce these effects by forcing global vectors to be calculated consistently (regardless of the number of processors used) when using the shared memory parallel solver (SMP). However, even with the consistency flag set to CONST=1, the results may not be identical.

A consistency test was performed on the baseline crush tube model by running a single file 3 times and comparing the results. Figure 5.30 shows the load-displacement curve from the models and focuses on two separate peaks. It is clear from the global load-displacement curve that the peaks occur in roughly the same location for every run of the model. However, when a peak is singled out, there are differences in peak load and peak location from run to run. Comparing the two results of the zoomed-in sections reveals that there is not a consistency in which run had the highest peak, and the peak magnitudes varied by as much as 2 kN from run to run.
Figure 5.30. Comparison of load-displacement results from crush tube model that was solved 3 separate times.
As the consistency flag was already set to 1 and an SMP, double precision solver was used, there are not any further settings to address this problem. A variability study could be performed to see how the results compare over a large number of runs, but that would quickly become computationally expensive and time consuming. The user must be aware that variability is a possibility with these simulations and this is a current limitation of LS Dyna.

In addition to the consistency issues, large errant spikes were noticed in the rigid wall reaction force data during baseline model setup. An example of these spikes can be seen in Figure 5.31. After contacting Livermore Software Technology Corp (the creators of LS Dyna) about the issue, they suggested setting the scale factor for rigid wall penalties, RWPNAL, to -1 in the *CONTROL_CONTACT keyword. This defines a penalty method for the wall and contacting surface and helps reduce the spikes in load. It should be noted however, that the manual does not list RWPNAL = -1 as an option for the keyword, so documentation is not available for this setting.

![Errant Spikes](image)

Figure 5.31. Errant spikes seen in rigid wall reaction force during crush tube baseline setup
Contact Definition

The ERODING contact class within LS Dyna is used in models where element deletion is common. The ERODING contact keywords update the contact surface when an element fails. The *CONTACT_ERODING_SINGLE_SURFACE keyword was used in the crush tubes. This is a segment based contact method that uses the SOFT=2 penalty definition. The contact method can control the time step definition for the model by setting the ECDT variable within the *CONTROL_CONTACT card to 0. The static and dynamic coefficients of friction were set to 0.08, and the viscous damping coefficient was set to 0.2%. Scale factors for the penalty method were left at default values, but Dr. Tabiei suggests increasing the factors for high velocity impacts [106]. The tube and bevel were both treated as slaves in the contact definition because a single surface contact method was used.

Damping

Global damping was applied to the crush tubes with the *DAMPING_GLOBAL keyword. The system damping constant was set to 0.3. Additional part stiffness damping was applied to the tube and bevel parts using the *DAMPING_PART_STIFFNESS keyword. The Rayleigh damping coefficient for the stiffness weighted damping was set to 0.1.

Data Output

The rigid wall reaction force was the primary output of interest for the crush tube model. This was activated on the *DATABASE_OPTION keyword, and the RWFORC parameter was checked. The output was requested for every 1E-08 seconds. The stress and strain outputs were requested every 1.0E-04 seconds using the *DATABASE_BINARY_D3PLOT keyword. The CMPFLG variable was set to 1 in
the *DATABASE_EXTENT_BINARY in order to have composite results output in the material coordinate system.

Section Definition

The shell elements were defined in the *SECTION_SHELL keyword as ELFORM = 2. The number of integration points varied from tube to tube depending on the layup, but the QR/IRID option was always activated. The number of integration points, NIP, were defined within the *INTEGRATION_SHELL keyword that was called by the section keyword through the IRID number. The ICOMP flag in the shell definition was activated so fiber orientations could be defined for in-plane angles of the element. The NLOC variable controls the location of the surface reference for shell elements with a defined thickness. The simulated crush tubes always have an inner diameter of 50.8 mm, therefore any thickness applied to the cylinder needs to be applied to the exterior of the tube. This is accomplished by setting the NLOC variable in the *SECTION_SHELL keyword to -1.0. The reference surface then becomes the interior surface of the tube.

Post-Processing Crush Tube Models

The rigid wall reaction force is the primary output from the crush tube models. Because the rigid wall is set to move at a constant velocity, the displacement of the rigid wall can be calculated using the basic equation

\[ d = r \cdot t \] (5.10)

where \( d \) is the distance, \( r \) is the rate, and \( t \) is the time. The rigid wall reaction force is output in a rwforc file along with an associated time vector. A Matlab script was written to read the rwforc file and calculate the raw load-displacement curve. As described in previous sections, a brittle material system in compression creates a saw tooth shape in the load-displacement plot. This is a result of element deletion in
conjunction with the constant-velocity rigid wall. As the mesh density study showed, the distance between the peaks in the load-displacement curve is directly related to the length of the element. Ideally, as the size of the element approaches zero, the distance between peaks would not be discernible. The load-displacement curve from experiments do not show a saw-tooth shape because the flat platens remain in constant contact with the part. For these reasons, it was decided to identify the peaks in the load-displacement simulation results and connect them with a line. This assumes that a continuous sample would maintain the load as the rigid wall moves to the next region of the tube, which is what happens in experiment. Figure 5.32 provides an example of the raw simulation results, and the results once the peaks are connected.

Figure 5.32. Post-processing example of the load-displacement curve from a crush tube model
The Matlab script breaks the load-displacement results into a number of bins related to the spacing of the peaks and identifies the peak load in each of these regions. The second peak in Figure 5.32 is not connected because the first two row of elements associated with the bevel do not fail at the same spacing as the other elements. This is not a problem because the stable crushing load (SCL), which is used to calculate the specific energy absorption, is the average load for the peaks associated with the full thickness of the tube. Therefore, the SCL was calculated using peak number 3 onward.

This is not a perfect solution for the discretization problem, and unfortunately, the Matlab script can be fooled by errant spikes in the load displacement response seen in Figure 5.31. In addition, the raw simulation results underestimate the amount of energy absorbed because of the element deletion. This can make an assembly model complicated if the user needs the energy absorption to be correct without post-processing.

The failure morphology of the tube simulations is not useful to compare to experiment simply because the failure of the simulated tubes is rather uninteresting. In general, when then tube is compressed a row of elements bend slightly inwards and are then deleted. The failure progression just shows the tube getting shorter as time passes.

The baseline model described in the above sections was used for all future analyses of composite tubes in crush. While some settings like tube thickness and stacking sequence may have varied slightly from model to model, the basics like the mesh and contact algorithms stayed the same. Using a baseline model provides a needed level of consistency between models. Once confidence in the baseline model has been developed, an additional layer of complexity can be added and validated. This building block approach works for any model development with this material system.
5.4.2 Validation with Experiment

There are not closed-form solutions available for the crushing of a composite tube at high speeds. While many researchers have published work related to the advancement of simulations with crush, none have found an analytical solution. This is largely because of the complexity of a failure problem with composites. In general, failure theories with composites are not 100% predictive with continuous fiber laminates because of the variability in strength. Add in the fact that these crush tubes are combined structural, material system, and failure propagation problems, and the increased complexity reduces the likelihood of finding a closed-form solution drastically.

Without a closed-form solution, it is difficult to verify the crush tube simulation. However, a comparison of experimental results and a simulation recreating said experiment can provide confidence in the model through validation.

Simulating the Axial Fiber Percentage Study

A series of crush tube simulations were developed to recreate the axial fiber percentage study described in Chapter 3. The baseline model was copied, and the thickness, number of integration points, and stacking sequence were all altered for each simulation. As previously mentioned, the thickness to diameter ratio of the tube affects the specific energy absorption results. In order to avoid including this effect in the simulations, the average thickness of the samples manufactured with a set percentage of axial fibers was applied as the simulation thickness. Table 5.7 provides the layup, number of integration points, and thickness for each of the simulations representing different axial fiber percentages in the study. A single stacking sequence was used for each of the axial fiber percentages, and the stable layup with $0^\circ$ plies on the interior of the tube was selected.

The load-displacement curves were recorded for each of the simulations and compared to the experimental results from the same axial fiber percentage grouping (See Figures 5.33 and 5.34). Going through the plots one by one reveals that the simulated
Table 5.7. Model specifications for each simulation in the axial fiber percentage study

<table>
<thead>
<tr>
<th>Percentage Axial Fibers</th>
<th>Layup</th>
<th>Number of Integration Points</th>
<th>Thickness [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>12.5%</td>
<td>[0/90_7]</td>
<td>8</td>
<td>2.65</td>
</tr>
<tr>
<td>20%</td>
<td>[0/90_4]</td>
<td>5</td>
<td>1.60</td>
</tr>
<tr>
<td>33%</td>
<td>[0/90_2][2]</td>
<td>6</td>
<td>1.93</td>
</tr>
<tr>
<td>50%</td>
<td>[90/0][4]</td>
<td>8</td>
<td>1.90</td>
</tr>
<tr>
<td>66%</td>
<td>[0_2/90][2]</td>
<td>6</td>
<td>2.0</td>
</tr>
<tr>
<td>80%</td>
<td>[0_4/90]</td>
<td>5</td>
<td>1.65</td>
</tr>
</tbody>
</table>

Load-displacement curve (with peaks connected) is lower than experimental results for 12.5% and 20% axial fibers. The simulation results are comparable to experiment for axial fiber percentage of 33% and 80%, and the simulation results are high compared to experiment for 50% and 66%. There is not a clear trend related to the percent of axial fibers and the simulation comparison to experimental results, but that may be because the results have not been normalized for the size of the tube. An alternative for comparing the simulation and experimental results is by using the specific energy absorption. This will normalize the response based off of the weight of the tube.

The specific energy absorption was calculated for each of the simulation results and plotted against the experimental SEA results for continuous fiber tubes using equation 2.4. When calculating the SEA from the simulation results, it became apparent that using a density of 1.8 g/cc was not correct when comparing simulation to experiment. Using the weight measurements from experiments in conjunction with the estimated cross sectional area allowed for an estimate of density. The estimated density averaged out to 1.424 g/cc and was used when calculating the SEA from crush tube simulations. Once this was completed, the simulation results were comparable to the experimentally determined values. Figure 5.35 shows that the simulated specific energy absorption follows an upward trend as the percent of axial fibers increases.
Figure 5.33. Experimental and simulated load-displacement curves for axial fiber percentages of 12.5%, 20%, and 33%
Figure 5.34. Experimental and simulated load-displacement curves for axial fiber percentages of 50%, 66%, and 80%
However, once 80% axial fibers was reached, the simulated specific energy absorption decreased.

![Figure 5.35. Specific energy absorption of continuous fiber tube simulations versus experimental results with the same layup](image)

As the percentage of axial fibers increased significantly (80% and beyond), the simulations had a tendency of going unstable. An example of instability in the model was presented in the section describing the mesh density study, and a similar phenomena happened in models with 80% or more axial fibers. It is interesting to note that the mesh density and the stiffness properties of the model both affect the time step calculations for the simulations.

Figure 5.36 provides an example of the catastrophic failure that occurs in tubes with a higher stiffness. Unlike the failure presented in Figure 5.12, the failure associated with increased stiffness is generally catastrophic. The tube stops taking any load as whole sections of the tube fall away. In 1.0E-05 seconds, over two dozen elements simply fail. The failure of so many elements at once results in the damage spreading faster throughout the tube, and the saw-tooth shape in the load-displacement curve
disappears. Figure 5.37 provides a load-displacement curve for a crush tube that had catastrophic failure. In this example, the percentage of axial fibers was 87.5%. As is evident, after the initial peak, the tube failed completely, and the reaction force on the rigid wall was reduced down to 0 kN.

This problem is a result of using an explicit analysis, the material properties, and the model setup. However, altering some settings could potentially mitigate the issue. These include altering the *CONTROL_TIMESTEP keyword and reducing the scale factor for the time step.

**Altering Stable Time Increment within LS Dyna**

As previously discussed, LS Dyna determines the stable time increment for a simulation based on material properties, element size in a mesh, the contact definition. However, this calculation merely provides an estimate for the software. The variable TSSFAC within the *CONTROL_TIMESTEP keyword is a scale factor for the calculated time step. The maximum time step available in the simulation will be the calculated time step multiplied by TSSFAC. The default is 0.9 for standard simulations, but a value of 0.67 is recommended for simulations with explosions.

Because of the instabilities associated with crush tube models that have higher stiffnesses, a brief study was completed to determine if altering the TSSFAC parameter reduced model instability. Three crush tube models were created with an axial fiber percentage of 87.5%. The models were identical with one exception, the TSSFAC value. The models were run with TSSFAC values of 0.9, 0.75, and 0.67. Figure 5.38 shows the load-displacement curves produced from the simulations. Once the TSSFAC value was reduced from the default of 0.9, the model stabilized and produced consistent results. In fact, the results using TSSFAC = 0.67 produced a constant load during the stable crushing of the tube.

This indicates that the stable time increment calculated by LS Dyna should not be wholly trusted. Models with increased mesh density or effective stiffness should have
Figure 5.36. Numerical instability causing catastrophic failure of crush tube
the time step scale factor reduced to improve stability. Unfortunately, the effect of this control variable was not discovered until late in this project. Many of the models had not been reran with a lowered TSSFAC variable by the time of this writing. However, knowing a solution to the instability problem exists is still valuable.

**Integration Point Study**

The continuous fiber simulations all had 8 or fewer integration points through the thickness of the shell elements. Each integration point represented a single angled ply, so the number of plies in the laminate determined the number of integration points. A simulation study was completed to see if the number of integration points defined for a specific layup affected the SEA. Increasing the number of integration points in the model drastically increased the run time and the size of the output files, so this study was only completed with two of the continuous fiber models: 20% axial fibers (which had a low simulated SEA compared to experiment) and 50% axial fibers (which had a high simulated SEA compared to experiment).
The 20% axial fiber model initially had 5 integration points. Three additional models were made with 10 integration points, 20 integration points, and 40 integration points respectively. The layup stayed the same, but the number of plies increased. In this case, instead of a single ply of 0°, there were now 2, 4, and 8. Figure 5.39 provides the load-displacement curves for the simulations with 20% axial fibers. Looking at the full graph shows that the peaks generally occur in the same location. However, zooming in on individual peaks reveals that the magnitude is not consistent for each model. The model with 5 integration points had a lower magnitude peak than the other models. This difference exceeded 5 kN at times, which is a 25% error and not insignificant. It is possible that small numerical differences compounded as time went on in the explicit analysis, but this was not confirmed. In addition, there is some
slight variability when running models because of how LS Dyna solves simulations with multiple processes, so that could play a part as well.
Figure 5.39. Comparison of load displacement curves for simulations with increasing number of integration points and 20% axial fibers
The model with 50% axial fibers produced slightly different results. The original model had 8 integration points, and the additional models had 16 and 24 integration points respectively. Figure 5.40 compares the load displacement curves for those models. The full load displacement curve reveals that the peaks occur in the same location for all simulations. Zooming in on individual peaks shows that responses primarily stayed consistent between the models. The results are not identical, but that can be chalked up to using an explicit analysis with multiple cores. This layup appears to be more insensitive to an increase in integration point. However, the fewest amount of integration points used for 50% axial fibers was 8, while the 20% axial fiber model used 5 integration points. Therefore, the models could be more sensitive when fewer than 8 integration points are used. Future crush tube models representing the PPMC all have over 8 integration points. Therefore, if the reduced number of integration points is the cause of the discrepancy between peak magnitudes, this will not be an issue in PPMC models.
Figure 5.40. Comparison of load displacement curves for simulations with increasing number of integration points and 50% axial fibers.
Once there was reason to believe an increased number of integration points (beyond 8) would provide different results, an additional model was created for each of the axial fiber percentages increasing the initial number of integration points. The updated stacking sequences and number of integration points for the models are listed in Table 5.8. In general, the plies in the layup were increased by either a factor of 2 or 4. The smallest number of integration points for any of the models was now 16.

Table 5.8. Model specifications for each simulation with an increased number of integration points

<table>
<thead>
<tr>
<th>Percentage Axial Fibers</th>
<th>Layup</th>
<th>Number of Integration Points</th>
<th>Thickness [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>12.5%</td>
<td>[0/90]_2</td>
<td>32</td>
<td>2.65</td>
</tr>
<tr>
<td>20%</td>
<td>[0/90]_4</td>
<td>20</td>
<td>1.60</td>
</tr>
<tr>
<td>33%</td>
<td>[0/90]_8</td>
<td>24</td>
<td>1.93</td>
</tr>
<tr>
<td>50%</td>
<td>[90/0]_2</td>
<td>16</td>
<td>1.90</td>
</tr>
<tr>
<td>66%</td>
<td>[0/90]_8</td>
<td>24</td>
<td>2.0</td>
</tr>
<tr>
<td>80%</td>
<td>[0/90]_4</td>
<td>20</td>
<td>1.65</td>
</tr>
</tbody>
</table>

Once the increased integration point models were run, the specific energy absorption was calculated for each of the layups. Figure 5.41 plots the specific energy absorption of experimental results, the original simulation results, and the new simulation results with an increased number of integration points.

Increasing the number of integration points did not have a consistent effect on the results. For axial fiber percentages of 12.5% and 20%, increasing the integration points resulted in a higher specific energy absorption. In the case of 20%, the SEA jumped almost 10 kJ/kg. The SEA stayed exactly the same for 50% axial fibers, and decreased for 33%, 66%, and 80%. The specific energy absorption dropped over 40 kJ/kg for the 80% case. However, that is because the model went unstable.

These results improved the comparison between the lower percent of axial fibers, but did nothing to improve the large difference of SEA seen at 50% and 66%.
As the percentage of axial fibers increases in the laminate, the laminate stiffness becomes higher because the fiber modulus becomes more dominant. If the material characterization produced an $E_1$ value that was too high, this could cause the simulation results to be inaccurate. It is a fair assessment that reducing the $E_1$ modulus in the crush tube simulations will lower the SEA of tubes with a higher percentage of axial fibers. This could get the simulation predictions to match better to experiment. With this in mind, the Young’s modulus in the fiber direction was reduced down to the value listed on the product data sheet (119 GPa) for a similar material system. The reduced number of integration points was used for these models.

These results were compared to experiment as well as the previous simulation results and can be seen in Figure 5.42. The SEA remained relatively unchanged at the lower percentage of axial fibers but reduced at 50% and 66% of axial fibers. In an unusual result, the SEA actually increased for the 80% case. It is possible the value
was lowered previously due to damage development. Overall, these results are more comparable to experiment, but there are obvious limitations in the modeling method.

![Graph showing specific energy absorption (SEA) vs. percent axial fibers]

**Figure 5.42.** Specific energy absorption of continuous fiber tubes compared to simulation results with reduced $E_1$

### 5.5 Conclusions Regarding Unidirectional Modeling in LS Dyna

A series of unidirectional models were developed as part of a building block approach to gain confidence in LS Dyna crash modeling. Baseline models were created and manipulated to understand the effect of model settings and to compare to experimental results. When possible the models were verified against closed form solution and validated with experiment.

LS Dyna, and all software for that matter, has limitations regarding crash simulation of composites. The composite material models utilized in this work are only available for shell elements, and the loading conditions applied to the shells can pro-
duce instabilities in the models. It is necessary to use explicit analysis but doing so can result in different solutions each time a model is run.

In addition, a brittle material modeled with shell elements under compressive loading presents unique challenges with regards to the stability of the solution. Element deletion related to the compression of the material results in a saw-tooth shaped load displacement curve that underestimates the energy absorption of the component. On top of this, the element deletion can cause instability in the model through the propagation of stress waves. The right combination of mesh density, material properties, and model settings can reduce the occurrence of instability but it may not be able to completely prevent it.

The flexure models were verified against analytical solutions, but when compared to the experimental results were much too stiff. Reducing the E1 modulus to 119 GPa and increasing the failure strain would result in better matching results. With that being said, the baseline model is fully functional, and it is safe to have confidence in the model before moving forward and adding further complexity.

In general, the SEA calculated from crush tube simulations was comparable to that of experiment. However, as the layup increased in longitudinal stiffness for the models, the results diverged. Decreasing the Young’s modulus for the fiber direction of the material reduces the divergence but does not completely fix it. Along with this, instability was common in models with an increased longitudinal stiffness, and models with a reduced mesh density.

As long as the limitations of the model are known, users can try and work around them. The baseline models created for both flexure and compression loading cases are not perfect, but the behavior is understood and the limitations and issues are known. The next chapter will add further complexity by incorporating platelet orientation states from the PPMC material system.
6. MANUFACTURING-INFORMED PERFORMANCE MODELS

Now that confidence has been established in the baseline UD models, an additional layer of complexity will be added to the models in the form of platelet orientations from the PPMC material. As previously stated, the prepreg platelet molding compound is comprised of layers of platelets made from unidirectional prepreg. The final orientation state of platelets determines the stiffness and strength of the part. Therefore, it is vital to incorporate orientation information into simulations in order to predict the mechanical performance of components.

6.1 Orientation Distribution Function

Advani and Tucker [79] introduced the use of second and fourth order tensors to describe the fiber orientation state for short fiber systems in 1987. A considerable amount of research has built off of their work, including flow simulation work for PPMC material [120]. For PPMC material, the fibers are assumed to be collimated and aligned along the longitudinal axis of the platelet. If the orientation of a platelet is known, then the orientation of the fibers within the platelet is assumed to be consistent. Say a platelet has the orientation, $\vec{p}$, in Figure 6.1.

The orientation of the platelet can be represented using spherical coordinates with

$$\{\vec{p}\} = \begin{pmatrix} \sin \theta \cos \phi \\ \sin \theta \sin \phi \\ \cos \theta \end{pmatrix}$$ (6.1)
A second order orientation tensor can be defined by using a series of platelet orientation measurements from micrographs or stereographs with the following equation [121]

$$A_{ij} = \frac{1}{N} \sum_{n=1}^{N} p_i p_j$$ \hspace{1cm} (6.2)

where $N$ is the number of orientation measurements and $p$ is the orientation vector for a platelet.

Sample bias is a concern when using microscopy measurements to determine orientation states, and weighting factors are often implemented to mitigate the issue. However, the method used in this work does not rely on fiber measurements but platelet orientation measurements. This means that a large quantity of fibers with a similar orientation are reduced down to a single platelet orientation value. This reduces bias so a weighting factor is unnecessary.

For this work, the orientation distribution of platelets is assumed to be planar, and all components associated with the 3-axis of the $A_{ij}$ tensor should be zero. The $A_{11}$ component is referenced the most in this work and is a measure of the degree of
alignment along the 1-axis (whatever its definition). The value ranges from 0 to 1, with 1 indicating the fibers are perfectly aligned along the 1-axis.

### 6.2 Modeling Approach for PPMC

There are two options when it comes to representing the PPMC material system in finite element analyses. The first option is to characterize the PPMC as a homogenous material and use those mechanical properties as inputs into an LS Dyna composite material card (see Figure 6.2). The second option is to characterize the unidirectional prepreg that is the base for PPMC and incorporate platelets and their orientations into the model (see Figure 6.3). A case can be made for either options, especially if computational time or accuracy is a concern.

![Characterize PPMC as homogeneous material](image1.png)

**Figure 6.2.** Modeling approach 1: Treat PPMC as homogeneous material

The first modeling approach requires a considerable amount of experimental work to develop a database of material performance, but it also allows for a relatively simple finite element analysis technique. The PPMC would be experimentally characterized as a homogenous material system. This is not as straightforward as it seems because the material properties are dependent on the platelet orientation distribution. Characterization would be necessary for various levels of platelet alignment in order to quantify the effect of alignment on mechanical properties. In addition, the material displays considerable strength variability even when samples were produced from the
same manufactured plate [15]. This is because of the stochastic nature of the material system. The variability in mechanical behavior for a set of orientation distributions would have to be quantified in order to understand the limits of performance.

After characterization, the material properties could be applied to a finite element analysis using a single material and orientation definition. However, if the material properties change significantly with the level of platelet alignment, multiple material definitions might need to be created in order to run the stiffest and softest cases. This would allow engineers to develop a performance envelope for the component.

The simplified modeling approach allows for reduced analysis times, but it also increases the necessary experimental work. In addition, this approach only provides a performance envelope for the component and may not be able to produce precise results if comparing directly to an experimental sample. It also neglects the effects of manufacturing.

![Characterize unidirectional tape](image1)

![Material orientations assigned to each element in FEM to capture platelets](image2)

**Figure 6.3. Modeling approach 2: Characterize UD prepreg and insert orientations into model**

The second modeling option for PPMC entails characterizing the unidirectional tape that serves as the base material for the PPMC. This traditional characterization approach requires less testing than the first modeling option.
Once the material properties are known, these can be input into LS Dyna as a single material keyword. However, the simplicity of the model stops there, as each platelet needs to have a separate orientation definition. As mentioned previously, when modeling laminates with shell elements in LS Dyna, the ply orientations are provided as in-plane angles for each integration point through the thickness. If a component has 33 platelets through the thickness, then 33 integration points, each with their own orientation angle, need to be defined for every shell element. Platelets can span across multiple elements, so the orientations associated with a platelet need to be carried over to each element in its footprint. Manually assigning each platelet orientation on shell elements is impractical for large parts. It then becomes necessary to create a process to assign platelet orientations to LS Dyna models. This method ensures platelet orientations affected by manufacturing parameters are captured, making this modeling method manufacturing sensitive.

This modeling option requires a considerable amount of software development work and increases the computational time of models drastically because of the increased number of integration points. However, this process would allow for the creation of digital twins, where the measured orientation state of an experimental sample could be mapped onto an LS Dyna mesh. The versatility available with this modeling option led to its selection in this work. The next section will describe how 3D platelet models are generated and mapped onto 2D meshes.

6.3 Orientation Mapping Process

The orientation mapping process needs to take orientation information from a 3D digital source and apply it to a 2D mesh. The source of the orientation data can be CT scan results, flow simulations, or digitally generated models. Anything that creates a 3D point cloud of orientation data can be used as an input to the mapping process.
Dr. Denos at Purdue University created an application, MapOr, in 2017 to map CT scan information onto a Matlab generated mesh and input file for a 3D Abaqus simulation [122]. For this project, the MapOr script was adjusted to accept orientation vectors related to the fiber and transverse directions and map them onto an Abaqus 3D, Abaqus 2D, or LS Dyna 2D mesh. The MapOr application uses the scattered interpolant Matlab class with the nearest neighbor option to map platelet angles to the element centroids of the 3D or 2D meshes.

When mapping to an LS Dyna mesh, MapOr reads in a .csv file containing the coordinates and associated orientation vectors for each of the data points. A LS Dyna k-file is imported into the application to receive the mapped information. If the geometries are oriented differently or not of an identical size, the user has a chance to manipulate the original data by either translation, rotation, or scaling to ensure proper alignment. Once this is done, the user specifies the number of plies and ply thickness to be applied to the new mesh. This creates an equally spaced number of interpolant points to be identified in the original point cloud. If the user defines 8 plies with a 0.1 mm ply thickness, a grid of points spaced 0.1 mm apart in the shell element normal direction would be created. Matlab’s scattered interpolant class then determines the components for the orientation vectors at each point. The orientation vectors are organized into a unique stacking sequence for each element within the mesh. The information is then output into the correct format for the LS Dyna keyword file.

In order to assign unique stacking sequences to each element in the mesh, a new section definition must be created for every element. This in turn requires a new part definition for all elements. This is not an inherently negative effect, but it makes the keyword file considerably longer and produces an interesting color distribution for the model in LS Pre-Post (see Figures 6.4 and 6.5). The local material orientations can be viewed for each integration point through the thickness of the shell, and examples of this can also be seen in Figures 6.4 and 6.5. The A-axis corresponds to the fiber direction of the platelet, and the B-axis is the in-plane transverse direction. The
C-axis is the out of plane direction normal to the shell element. Platelet groupings can be seen with local material orientations lining up across several elements.

6.3.1 Generating Digital PPMC Components

The orientation information for mapping can come from a variety of sources like CT scanning and flow simulations. However, when studying the performance vari-
ability of PPMC, it is faster to digitally create models with various orientation distributions than to run a series of flow simulations or CT scans.

Dr. Favaloro at Purdue University wrote a Python application, referred to as PlateletGen, that places platelets into a predefined global volume following a given orientation distribution. This application was originally written to create virtual tensile bars for simulation studies, but was adjusted to include crush tubes as well.

The user defines the dimensions of the virtual sample, the number of platelet layers, the global level of collimation (A11 in the second-order orientation tensor), and any global off axis angle. In addition, the element dimensions for the mesh and the platelet dimensions are defined. Figure 6.6 provides a visual of the graphical user interface (GUI) for the PlateletGen application created by Dr. Favaloro.

Once the user has defined all of the required variables, the PlateletGen code runs through an algorithm to fill the volume with platelets. The code starts with the lowest layer of elements that is not completely filled with platelets. An unfilled element
within this layer is selected as the centroid of a new platelet. An in-plane angle is produced from the prescribed orientation distribution and applied to form a platelet of the given dimensions. The platelet’s footprint is noted, and the code identifies the lowest unfilled element within each column of the footprint. If every column within the platelet’s domain has an unfilled element available and the maximum thickness of the part has not been exceed, the code assigns the unfilled elements to the platelet. However, if there are elements that are unavailable, the code repeats the process by picking another element as a centroid. If the code is still unsuccessful after a large
number of attempts, the process is repeated again, but the filled columns are simply ignored.

Out-of-plane angles are assigned to the orientation of the elements if the footprint of the platelet spans across elements in different layers. If there is a limited number of available space left in a virtual sample, the code tends to bias towards generating aligned platelets. This is because the code produces a new platelet angle after a denied attempt. If there is only 1 layer of unfilled elements left, overlapping platelets will not be likely with the algorithm specified. Figure 6.7 compares the first 100 platelets defined in an 8-layer sample (where overlapping can occur) and 1-layer sample (where overlapping is less likely) both with an \( A_{11} = 0.5 \). Overlapping is much more common in the 8-layer sample because there are available unfilled elements in the next layer preventing rejection.

First 100 platelets of 8-layer sample

First 100 platelets of 1-layer sample

Figure 6.7. First 100 generated platelets in 8-layer model versus 1-layer model with \( A_{11} = 0.5 \) showing inherent bias in the 1-layer model

Making a virtual tube sample is similar to a flat plate, but a periodic boundary is applied to the free edges along the longitudinal axis of the tube to prevent a discontinuity. An example of the discontinuity can be seen in Figure 6.8, where a noticeable seam runs along the length of the tube. Incorporating the periodic boundary condition prevents the seam which is not seen in real manufactured tubes.
6.4 Mapped Flexure Samples

Once the mapping process was established, several simulation studies were created to determine if the modeling approach for PPMC could be verified against analytical solutions and validated against experiment.

6.4.1 Platelet Collimation Sensitivity Study

Forty separate PPMC flexure simulations were created using the baseline model. The sample thickness was changed to 2.7 mm creating a span to thickness ratio of 28.4, and 30 platelets were defined through the thickness. Individual platelets had dimensions of 25.4 mm x 4.75 mm x 0.09 mm. The platelet generation code produced 10 models for each of the following levels of global collimation: \( A11 = \{0.16, 0.33, 0.66, 0.83\} \). Figure 6.9 shows the distribution of platelets on the top surface of a

Figure 6.8. Digitally created crush tube with a seam because the periodic boundary conditions are not applied
sample from each of the collimation groups. As $A_{11}$ increased, the platelets became
more aligned with the longitudinal axis of the flexure sample. It is important to note
that even though 10 models were made with the same global level of collimation,
the PlateletGen application does not produce models with the exact same $A_{11}$ value
every time. There is variation in the final orientation state from what was initially
requested. This means performance results will vary.

$A_{11} = 0.16$

$A_{11} = 0.33$

$A_{11} = 0.66$

$A_{11} = 0.83$

Figure 6.9. Flexure models with varying levels of platelet collimation
$A_{11} = \{0.16, 0.33, 0.66, 0.83\}$

Each model was run 2 times, once with MAT 54 and once using MAT 58. Figures
6.10 and 6.11 show the stress strain plots for the models.
Figure 6.10. Stress-strain curve for flexure samples with varying A11 components using MAT 54

Figure 6.11. Stress-strain curve for flexure samples with varying A11 components using MAT 58
From the figures, it is clear that the flexural modulus and strength increase with the level of platelet collimation. This is to be expected as it is more likely that platelets (and therefore fibers) will be oriented along the longitudinal axis when $A_{11}$ is higher. When fibers are oriented along the longitudinal axis, the bending stiffness of the part increases, resulting in a higher flexural modulus.

The results from the models using MAT 54 generally had a linear slope to failure while the MAT 58 models had a peak in strength followed by an extended amount of strain holding a constant stress. This is because of the higher failure strain associated with the MAT 58 material model. In addition, models with the same global collimation level performed similarly to each other. While there are slight differences in modulus and strength between the models with the same $A_{11}$, there is a clear distinction between the collimation levels. A single model from the $A_{11} = 0.33$ group and the $A_{11} = 0.83$ group had significantly lower flexural moduli and strengths compared to their counterparts. It is possible this is related to local transverse platelet alignment within the model resulting in premature failure. However, this could also be due to instability in the model. The stable time step scaling factor could be adjusted to determine the root cause.

The average flexural modulus and strength were recorded for the models and can be seen in Table 6.1. The flexural modulus was not identical between the models using MAT 54 and MAT 58. In fact, the MAT 54 models were consistently stiffer than the MAT 58 models, however the difference between the values was less than a single standard deviation for each case. The average strengths for the models were similar, and the material models alternated between having the highest average strength for each level of collimation. The differences in strength were also generally within a single standard deviation.

The standard deviations for strength were similar for all of the $A_{11}$ levels. The standard deviation for the flexural modulus at $A_{11} = 0.33$ and $A_{11} = 0.83$ was higher because of the single models that failed prematurely. However, if those outliers were removed, the standard deviation would be consistent across all collimation levels.
### Table 6.1. Average flexural modulus and strength for simulation models with increasing amounts of platelet collimation

<table>
<thead>
<tr>
<th></th>
<th>A11 Average $E_f$ [GPa]</th>
<th>Std. Dev [GPa]</th>
<th>Average $\sigma_f$ [MPa]</th>
<th>Std. Dev [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>MAT 54</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.16</td>
<td>17.38</td>
<td>1.49</td>
<td>209.19</td>
<td>34.96</td>
</tr>
<tr>
<td>0.33</td>
<td>27.13</td>
<td>6.15</td>
<td>340.34</td>
<td>32.80</td>
</tr>
<tr>
<td>0.66</td>
<td>53.89</td>
<td>2.97</td>
<td>493.7</td>
<td>23.38</td>
</tr>
<tr>
<td>0.83</td>
<td>74.12</td>
<td>10.89</td>
<td>600.53</td>
<td>28.79</td>
</tr>
<tr>
<td><strong>MAT 58</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.16</td>
<td>16.73</td>
<td>1.61</td>
<td>224.43</td>
<td>34.16</td>
</tr>
<tr>
<td>0.33</td>
<td>25.89</td>
<td>5.91</td>
<td>333.59</td>
<td>35.44</td>
</tr>
<tr>
<td>0.66</td>
<td>51.92</td>
<td>3.04</td>
<td>526.64</td>
<td>35.07</td>
</tr>
<tr>
<td>0.83</td>
<td>71.64</td>
<td>12.11</td>
<td>579.92</td>
<td>37.79</td>
</tr>
</tbody>
</table>

### 6.4.2 Verification of PPMC Flexure Models with an Analytical Solution

One of the benefits to digitally creating PPMC samples with various levels of platelet alignment is all of the platelet orientations are known. Therefore, the second and fourth order orientation tensors can be calculated for each model. Advani and Tucker [79] provided equations to predict the effective mechanical properties of short fiber composites using the orientation tensor. This allows the simulation results of PPMC samples to be compared to an analytical solution in order to verify the modeling method.

In order to calculate the effective mechanical properties, the unidirectional material properties must be known. The compliance matrix, $[S_{ij}]$, for an orthotropic material is defined as
\[
[S_{ij}] = \begin{bmatrix}
\frac{1}{E_1} & -\frac{\nu_{12}}{E_1} & -\frac{\nu_{13}}{E_1} & 0 & 0 & 0 \\
-\frac{\nu_{12}}{E_1} & \frac{1}{E_2} & -\frac{\nu_{23}}{E_3} & 0 & 0 & 0 \\
-\frac{\nu_{13}}{E_1} & -\frac{\nu_{23}}{E_2} & \frac{1}{E_3} & 0 & 0 & 0 \\
0 & 0 & 0 & \frac{1}{G_{23}} & 0 & 0 \\
0 & 0 & 0 & 0 & \frac{1}{G_{13}} & 0 \\
0 & 0 & 0 & 0 & 0 & \frac{1}{G_{12}}
\end{bmatrix}
\]

(6.3)

Once inverted, the compliance matrix becomes the stiffness matrix \([C_{ij}]\). Components from the stiffness matrix are then used to calculate the orientation averaging coefficients

\[
B_1 = C_{11} + C_{22} - 2C_{12} - 4C_{66}
\]

(6.4)

\[
B_2 = C_{12} - C_{23}
\]

(6.5)

\[
B_3 = C_{66} + \frac{1}{2}(C_{23} - C_{22})
\]

(6.6)

\[
B_4 = C_{23}
\]

(6.7)

\[
B_5 = \frac{1}{2}(C_{22} - C_{23})
\]

(6.8)

These coefficients are used in conjunction with the fourth and second order orientation tensors to calculate the orientation averaged stiffness, \(\langle C_{ijkl} \rangle\).

\[
\langle C \rangle_{ijkl} = B_1(a_{ijkl}) + B_2(a_{ij}\delta_{kl} + a_{kl}\delta_{ij}) + \\
B_3(a_{ik}\delta_{jl} + a_{il}\delta_{jk} + a_{jl}\delta_{ik} + a_{jk}\delta_{il}) + \\
B_4(\delta_{ij}\delta_{kl}) + B_5(\delta_{ik}\delta_{jl} + \delta_{il}\delta_{jk})
\]

(6.9)
Inverting the orientation averaged stiffness gives the orientation averaged compliance $\langle S_{ijkl}\rangle$, and the orientation averaged properties, like $E_x$, can be calculated from this compliance. Using these equations and the material properties for the unidirectional prepreg, the effective axial modulus $E_x$ can be calculated for any value of $A_{11}$.

Ideally, the flexural modulus will approach the axial modulus of a sample. With this in mind, the effective axial modulus from equations can be compared to the flexural modulus from simulations in order to validate the LS Dyna modeling method.

The orientation tensor was calculated for each of the digitally created models, and the $A_{11}$ value was recorded. Even though the global collimation level was set to 0.16 for ten of the digitally created models, the PlateletGen code produced values closer to $A_{11} \approx 0.24$. The PlateletGen code does not produce platelets to perfectly match the requested collimation value but instead retrieves angles from the given orientation distribution. As was discussed previously, bias is a possibility when using this code, and this is a potential example.

The effective axial modulus calculated with the prepreg material properties and the equations provided above was compared to the flexural modulus produced in simulations. Figure 6.12 shows the curve for analytical solution versus the simulation. The simulation results line up well with the analytical solution, although they tend to be slightly lower than analytical solution. MAT 54 results are always stiffer than their MAT 58 counterparts as was seen in the unidirectional flexure comparison as well. However, the difference is generally negligible. There are a few instances of outliers around $A_{11} = 0.83$ and 0.3, which was seen previously in the load displacement curves. The simulation results were slightly lower than the analytical solution for a lower degree of axial alignment, but they are still comparable. These results indicate the modeling method is functioning and the mapping process works.

A model from each collimation level was reran with the reduced Young’s modulus, $E_1 = 119$ GPa and compared to the results from the stiffer simulations. Figure 6.13 shows that effective flexural modulus from simulation only reduced when the $A_{11}$
value increased to 0.66 or 0.8. The simulations matched the analytic solution for a reduced E1 as well. This provides further confidence in the modeling method.

### 6.4.3 Validation with Experiment

Once the modeling method was verified, the results were compared to those from experiment in order to validate the simulations. Three PPMC plates were manufactured using a 250 ton Wabash National press. The plates were manufactured with a 15” x 15” flat plate tool that was heated to 140°C. The charge was placed in the center of the heated tool, and was pressed at 10 tons for 10 seconds before ramping up to 250 tons of force. The 10 second hold at 10 tons allows for heat to transfer into the material system prior to flow.

Each of the plates had different sized charges and resulting final thicknesses. The variation in charge dimension for each plate was used as a way to increase the amount
of flow in the material during molding, which would lead to an increase in global platelet collimation. Table 6.2 provides the name of the manufactured plates, the charge dimensions, and the final plate thicknesses. As the charge length decreased, the number of layers required to fill the mold increased. In addition, the distance the material had to flow increased with a decrease in the charge length.

Table 6.2. Manufacturing charge dimensions and final thickness for experimental flexure samples

<table>
<thead>
<tr>
<th>Plate</th>
<th>Charge Width [in]</th>
<th>Charge Length [in]</th>
<th># Layers</th>
<th>Final Thickness [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>121203-04</td>
<td>12</td>
<td>12</td>
<td>3</td>
<td>2.44</td>
</tr>
<tr>
<td>150507-02</td>
<td>15</td>
<td>5</td>
<td>7</td>
<td>2.74</td>
</tr>
<tr>
<td>150308-01</td>
<td>15</td>
<td>3</td>
<td>8</td>
<td>1.92</td>
</tr>
</tbody>
</table>

Eleven flexure samples were cut out of each plate, 7 aligned along the flow direction of material and 4 aligned transverse to the flow direction. It was expected that
samples aligned along the flow direction would have a higher flexural modulus and strength because the flow of this material promotes collimation of platelets [23]. The farther the material flows, the platelets start to align more, which in turn increases the longitudinal stiffness of the samples. Using this logic, the samples from the flow direction of the plate 150308-01 would have the largest flexural modulus because that charge flowed the most.

The samples were tested with a span of 76.8 mm and a top pin velocity of 3 mm/min. Table 6.3 lists the experimentally characterized strength and flexural modulus for each of the plates and sample types. The samples from the 121203-04 plate had similar flexural moduli and strength for both the flow and transverse directions. This plate had limited flow of the material, and the flow that did occur was not along a single axis. If the PPMC sheets started with an in-plane random distribution of platelets, $A_{11} = 0.5$, the resulting collimation from plate 121203-04 post molding would have a similar value. Meanwhile, the samples in the flow direction of plates 150507-02 and 150308-01 had higher stiffness and strengths than their transversely aligned counterparts. This was expected due to the increased amount of flow during the plate’s manufacturing.

Table 6.3. Modulus and strength results of experimental PPMC flexure testing

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>121203-04 Flow</td>
<td>33.37</td>
<td>1.12</td>
<td>460.4</td>
<td>8.10</td>
</tr>
<tr>
<td>121203-04 Transverse</td>
<td>33.34</td>
<td>0.28</td>
<td>493.64</td>
<td>3.36</td>
</tr>
<tr>
<td>150507-02 Flow</td>
<td>48.92</td>
<td>2.89</td>
<td>602.23</td>
<td>19.95</td>
</tr>
<tr>
<td>150507-02 Transverse</td>
<td>20.07</td>
<td>1.51</td>
<td>246.19</td>
<td>33.06</td>
</tr>
<tr>
<td>150308-01 Flow</td>
<td>66.17</td>
<td>4.68</td>
<td>738.55</td>
<td>47.59</td>
</tr>
<tr>
<td>150308-01 Transverse</td>
<td>21.97</td>
<td>3.01</td>
<td>267.92</td>
<td>16.38</td>
</tr>
</tbody>
</table>

In order to compare the experimental flexural moduli to the simulation results, a level of collimation had to be estimated for each manufactured plate. The exact solution of Jeffery’s equation with planar random initial conditions under lubricated
1D squeeze flow was used to estimate the A11 value for each of the plates [120, 123]. This produces an equation to estimate global collimation:

\[
A11 \approx \frac{d_f}{d_i} \frac{d_f}{d_i + 1}
\]  
(6.10)

where \(d_f\) is the final width of the material after flow, and \(d_i\) is the width of the initial charge. As an example, the 150308 plate had an initial charge width of 3" and a final width of 15". The A11 component in the flow direction would therefore 0.83. Table 6.4 lists the estimated A11 components for the samples associated with each manufactured plate and the direction of the samples.

<table>
<thead>
<tr>
<th>Samples</th>
<th>Estimated A11 from Jeffery’s equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>121203-04 Flow direction</td>
<td>0.5</td>
</tr>
<tr>
<td>121203-04 Transverse direction</td>
<td>0.5</td>
</tr>
<tr>
<td>150507-02 Flow direction</td>
<td>0.75</td>
</tr>
<tr>
<td>150507-02 Transverse direction</td>
<td>0.25</td>
</tr>
<tr>
<td>150308-01 Flow direction</td>
<td>0.83</td>
</tr>
<tr>
<td>150308-01 Transverse direction</td>
<td>0.17</td>
</tr>
</tbody>
</table>

Using these values, the experimentally determined flexure modulus was then compared to the analytical solution and can be seen in Figure 6.14. While the experimental results are similar to simulation at lower values of A11, the results diverge at A11 = 0.5. The experimental values are significantly lower than the analytical solution with E1 = 140 GPa, and are closer to the analytical values of E1 = 119 GPa. It is unclear why the results are so different, however the degree of alignment used for the experiment results are just estimates.

In order to identify the quality of the estimates for the A11 of the experimental samples, the remains of the flexural samples were sectioned and potted for microscopy.
The orientation state of each sample was determined using an in-house application developed by Dr. Nate Sharp for the orientation measurement of elliptical fibers using microscopy images [124].

Determining the platelet orientation distribution for each flexure sample is a multi-step process. First a microscopy image of the PPMC cross section is taken at 200X magnification. The image is then imported into the software ImageJ. Layers associated with individual platelets are outlined using the polygon tool within ImageJ (see 6.15 for an example). A series of individual fibers representing a 90° platelet are circled within the image (if available) using the elliptical tool, and the average major and minor axes of these ellipses are calculated.

The elliptical fiber analysis application is run using the microscopy image of the region, the individual platelets outlined, and the measured dimensions of the ellipses. The application produces estimated platelet orientations based off of multiple fiber measurements within each platelet. Figure 6.15 shows a color coordinated image of
fibers identified in the analysis application. Assuming the cross section looks along the longitudinal axis of the sample, the colors represent the in-plan orientation angle for fibers in the image. Shades of blue mean the fibers are aligned closely with the longitudinal axis (angles close to 0°) while hotter colors indicate larger angles.

Although a platelet contains hundreds of fibers, when calculating the orientation tensor, each platelet is assumed to contain one orientation angle. This prevents sampling bias, which is common when using microscopy to determine the orientation distribution of fibers. The second order orientation tensor was calculated for each of the potted samples and the A11 component along the longitudinal axis of the sample is listed in Table 6.5. The estimated A11 for the flow and transverse direction of an individual plate do not add up to 1.0 even though they theoretically should. This is because these measurements were made experimentally by measuring a few platelet groupings within the sample.

As only a subsection of samples were potted and analyzed for microscopy, there was not an estimated value of A11 for each sample. However, each plate and di-
Table 6.5. Estimated level of collimation for manufactured samples from microscopy measurements

<table>
<thead>
<tr>
<th>Samples</th>
<th>Estimated A11 from Microscopy</th>
</tr>
</thead>
<tbody>
<tr>
<td>121203-04 Flow direction</td>
<td>0.5</td>
</tr>
<tr>
<td>121203-04 Transverse direction</td>
<td>0.5</td>
</tr>
<tr>
<td>150507-02 Flow direction</td>
<td>0.657</td>
</tr>
<tr>
<td>150507-02 Transverse direction</td>
<td>0.387</td>
</tr>
<tr>
<td>150308-01 Flow direction</td>
<td>0.70</td>
</tr>
<tr>
<td>150308-01 Transverse direction</td>
<td>0.34</td>
</tr>
</tbody>
</table>

rection had at least two samples potted and analyzed for microscopy. The averaged microscopy results for each plate and direction were then used to represent the A11 for each experimental sample in that grouping. The flexural moduli from experiment were plotted using the averaged microscopy results for the estimated A11 (see Figure 6.16). The microscopy results reduced the spread of A11 and brought the experimental points closer together. The experimental results are closer to the analytical results with $E_1 = 119$ GPa, which provides another indication that $E_1 = 140$ GPa is an incorrect value. The experimental results are still lower than the analytical solution, but there could be several underlying causes for this. The simulations do not include out-of-plane orientation components, and these are present in reality. In addition, the samples tested were thin with only $\approx 16 - 20$ platelets through the thickness. This sample size is small enough that a random orientation distribution for the initial conditions of the platelets is unlikely. The results of flexure tests with PPMC components are also heavily dependent upon local orientations, and this could have an effect.

The strength of the experimental samples was also plotted against the estimated A11 from microscopy to compare to simulation results (see Figure 6.17. The simulation and experimental results show a considerable amount of spread for each level of collimation, and both show an increase in strength with A11. However, the simu-
I
lation results under predict the strength at higher values of A11, but that could be related to the failure strain set in the material models. Increasing the failure strains could result in a better match. The trend for strength versus A11 component has a higher slope for the experimental results versus the simulations.

Incorporating the orientation distribution of platelets within the flexure models significantly affected the estimated flexural modulus and strength. The simulations were verified against an analytical solution, but the experimental moduli were slightly lower than prediction. Additional work studying the cause of the discrepancy would increase confidence in the modeling method when it is applied to advance geometries with flexure loading conditions.
6.5 Mapped Crush Tubes

A similar mapped crush tube study was completed in order to understand how tube performance is affected by collimation of platelets. As there is not a closed form solution available for verification, the results were simply compared to experiment for validation.

6.5.1 Sensitivity Study

Forty crush tubes were created using the baseline model with varying levels of global collimation in order to understand how the simulated energy absorption is affected by A11. Groups of ten tubes were created with the following levels of collimation, $A11 = \{0.16, 0.3, 0.66, \text{ and } 0.83\}$. Figure 6.18 shows the surface platelets of a sample from each of the four groups. The platelets are more aligned along the hoop.
direction for $A_{11} = 0.16$ while the collimation along the longitudinal axis increases as $A_{11}$ increases.
The tubes were assigned a thickness of 2 mm, and the rigid wall velocity was 0.32 m/s. A bevel was defined on the tubes, and all simulations were run twice, once with MAT 54 and once with MAT 58. The load-displacement curve for each simulation was calculated using the post processing technique described in the previous chapter. Figures 6.19 and 6.20 show the load displacement curves for the stable tube models using MAT 54 and MAT 58.

Figure 6.19. Simulated load-displacement curves for tubes using MAT 54

As the level of collimation increased, the stable crushing load increased. This was to be expected as the estimated longitudinal stiffness increases with collimation. The groupings by level of collimation are tightly packed and do not show significant variation. The crushing loads associated with MAT 58 were consistently higher than those of MAT 54. This is a little unexpected because MAT 54 was consistently stiffer in the 3-point bend models. However, it is possible the higher stable crushing load is due to the increased failure strain for MAT 58. In addition, the results for A11 = 0.83 with MAT 58 had little separation from the A11 = 0.66 results.
As global collimation increased, several models with $A_{11} = 0.83$ went unstable. This was also seen in the continuous fiber simulations when the percent of axial fibers increased past 80%. Rerunning the simulations with a reduced stable time increment scaling factor would likely mitigate this issue. The load-displacement curves for the unstable models were simply removed to provide a clear visual of the trends for the crush tubes.

6.5.2 t/D Study

There is not significant variation in the load displacement curves for models that have the same global collimation level, but these models are created with identical geometries and material properties. The only difference between the models is the platelet orientation distribution.

Significant variation in specific energy absorption was noticed in experimental results when the geometry of the crush tubes differed slightly, especially when looking
at the thickness-to-diameter ratio. A simulation study was completed to see if the
SEA was significantly affected when the model geometry was altered.

Twelve PPMC models with a global collimation of $A_{11} = 0.5$ were created with
the same keyword definitions as the baseline model. However, the tube thickness or
inner diameter was changed for each of the models in order to alter the t/D ratio.
The t/D ratios simulated varied from 0.030 to 0.045, which was representative of the
range seen experimentally. Six of the models kept a constant inner diameter and had
their thickness altered, while the other six models kept a constant thickness and had
the inner diameter adjusted. Table 6.6 provides the specifications for each model.
Each model was created separately using the PlateletGen application, so there are
likely to be slight differences in the orientation distributions.

Table 6.6. Thickness, inner diameter, and number of platelets for t/D model study

<table>
<thead>
<tr>
<th>Model</th>
<th>t [mm]</th>
<th>ID [mm]</th>
<th>Number of Platelets</th>
<th>t/D ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>ID1</td>
<td>1.52</td>
<td>50.75</td>
<td>17</td>
<td>0.030</td>
</tr>
<tr>
<td>ID2</td>
<td>1.67</td>
<td>50.75</td>
<td>19</td>
<td>0.033</td>
</tr>
<tr>
<td>ID3</td>
<td>1.82</td>
<td>50.75</td>
<td>20</td>
<td>0.036</td>
</tr>
<tr>
<td>ID4</td>
<td>1.98</td>
<td>50.75</td>
<td>22</td>
<td>0.039</td>
</tr>
<tr>
<td>ID5</td>
<td>2.13</td>
<td>50.75</td>
<td>24</td>
<td>0.042</td>
</tr>
<tr>
<td>ID6</td>
<td>2.28</td>
<td>50.75</td>
<td>25</td>
<td>0.045</td>
</tr>
<tr>
<td>t1</td>
<td>2.0</td>
<td>66.67</td>
<td>22</td>
<td>0.030</td>
</tr>
<tr>
<td>t2</td>
<td>2.0</td>
<td>60.6</td>
<td>22</td>
<td>0.033</td>
</tr>
<tr>
<td>t3</td>
<td>2.0</td>
<td>55.55</td>
<td>22</td>
<td>0.036</td>
</tr>
<tr>
<td>t4</td>
<td>2.0</td>
<td>51.28</td>
<td>22</td>
<td>0.039</td>
</tr>
<tr>
<td>t5</td>
<td>2.0</td>
<td>47.6</td>
<td>22</td>
<td>0.042</td>
</tr>
<tr>
<td>t6</td>
<td>2.0</td>
<td>44.4</td>
<td>22</td>
<td>0.045</td>
</tr>
</tbody>
</table>

The specific energy absorption was calculated for each model, and the results are
plotted against the t/D ratio in Figure 6.21. There was an increase in SEA with the
t/D ratio, and this is consistent with what was seen experimentally.
In general, the constant inner diameter models had a higher specific energy absorption than models with a constant thickness. As the constant inner diameter models increased in thickness, the number of platelets through the thickness increased. This could be a cause for the higher SEA compared to the constant thickness models which maintained 22 platelets through the thickness of all models. The range of SEA was from about 55 kJ/kg to 61 kJ/kg for the simulations. When compared to the variability in SEA for experimental samples with similar t/D ratios, the range in simulation results is much too small (see Figure 6.22). The differences between the simulation and experimental results could have numerous causes. This includes manufacturing variability present in the experimental samples that cannot be captured with simulation. In all likelihood, the experimental samples have a larger variability in SEA because the consolidation and cure cycle is not consistent from one tube to the next, even while trying to repeat the process.
The SEA can be directly related to the thickness-to-diameter ratio for metal tubes like aluminum. With this knowledge, an additional t/D simulation study could be performed in LS Dyna with isotropic materials that display consistent behavior. If the variability in SEA is still not captured with the simulation results, the cause could be the LS Dyna model and adjustments can be made.

![Graph showing SEA results for t/D simulation study versus experiment](image)

Figure 6.22. SEA results for t/D simulation study versus experiment

The simulation study may not capture the magnitude of the effect t/D has on SEA, but it at least shows a positive trend between SEA and t/D ratio. This does not invalidate the crush tube simulations but reveals a limitation which should be addressed.
6.5.3 Validation with Experiment

Once the PPMC crush tube models were run, the load-displacement curves were compared to that of experiment (see Figures 6.23 and 6.24). Assuming the PPMC sheets have a random in-plane orientation, the experimental results should lie somewhere between the simulation results of $A_{11} = 0.3$ and $A_{11} = 0.6$. This is generally the case when looking at the load-displacement comparison between experiment and the LS Dyna simulations. However, the $A_{11} = 0.3$ results for MAT 58 start to overlap with some of the experimental results. The increase in stable crushing load for the models using MAT 58 is likely due to the increased failure strain within the material definition.

Figure 6.23. Load-displacement results for MAT 54 simulations versus experiment

In reality, the direct load-displacement comparison is not the best format for a validation comparison because it does not take the geometry difference into account. Although manufactured with the same method, each of the experimental crush samples had slightly different weights and cross sectional areas. Meanwhile, the simulated
tubes represent the ideal case and were all perfectly consolidated with identical geometry.

A better comparison would be to look at the specific energy absorption for the simulation and experiment because this factors in differences in geometry. Figure 6.25 plots the SEA of the simulations and experiment versus the global level of collimation. It was assumed that $A_{11} = 0.5$ for experiments. The experimental results are bounded by the simulation results and line up well with results from MAT 54 and MAT 58. Even though the simulation results do not display as much variability in SEA for a given $A_{11}$ grouping, the trend in data with increasing platelet collimation matches the experimental results. These results provide confidence in the modeling method for PPMC and crush tubes.
Figure 6.25. SEA results from crush tubes using MAT 54 and MAT 58 compared to experiment

6.6 Mapping Advanced Geometries

The orientation mapping process introduced in this chapter is not limited to simple geometries like flat plates and tubes, but there are limitations in the present workflow that need to be addressed. As previously stated, data from CT scans, flow simulations, and models from the PlateletGen application can all be mapped onto LS Dyna simulations. However, the PlateletGen application is currently limited to simplified geometries including rectangular plates of any dimension and hollow tubes. Additional development would be required to incorporate a more diverse group of geometries.
Another issue that comes into play when thinking about advanced geometries is the size of the component. The largest geometries analyzed in this research were 100 mm long crush tubes. This allowed the element size in simulations to be less than 2.4 mm x 2.4 mm while the platelets were 25.4 mm x 4.75 mm x 0.09 mm. A single platelet could be defined with up to 20 elements using these dimensions. However, as soon as the components become larger (say the size of a deck lid or a lift gate), it will be impractical to keep the element dimensions as small. Modeling individual platelets for large components is not practical in terms of computational time and model fidelity. In addition, as the size of the components increases, the ability to map CT scan information over to new models will reduce. CT scans are monetarily expensive, and there is a balance between the resolution of the scan and the size of the part. Increasing the size of the scanned part will result in a lower resolution, possibly preventing the analysis method from working as well. Flow simulations are the most likely source of orientation information for larger components, but those simulations also have to balance model size with result resolution.

A potential solution for this is to model larger components without including individual platelets. If an element is larger than a single platelet, the orientations through the thickness can be angles chosen from the supplied orientation distribution to represent a grouping of platelets. This would allow the model to incorporate orientation distributions without including individual platelets. However, this method would need to be tested before confidence could be developed.

Overall, the customer must be willing to provide guidelines regarding the accuracy of the solution needed as well as any computational constraints. Once these project parameters are put in place, simulation solutions can be designed to meet the requirements.
7. CONCLUSIONS

7.1 Review of Work

This project investigated the performance of a new material system, prepreg platelet molding compound, in crash conditions. Components with loading conditions representative of those seen in vehicular crash (flexure and compression) were experimentally tested and simulated to identify the effect of orientation distribution and manufacturing flow on performance.

A manufacturing method for the creation of PPMC tubes was developed, and an experimental study concluded that PPMC crush tubes perform similarly to continuous fiber tubes with the same effective laminate stiffness. In addition, the PPMC tubes performed as well as aluminum and better than steel tubes with similar thickness-to-diameter ratios. This all assumed the PPMC tubes had a random in-plane platelet orientation when manufactured. An increase in alignment of platelets during the manufacturing process of PPMC would theoretically increase the specific energy absorption, as seen with continuous fiber tubes.

PPMC tubes presented a different failure morphology than continuous fiber tubes. These tubes failed primarily through delamination of the platelets, and the fronds produced from the crushing process had little to no bending stiffness. When crushed at dynamic speeds, the PPMC tubes no longer produced fronds at all, and instead sprayed small platelet groups and dust into a cloud surrounding the tube. The speed of testing achieved 8 m/s or roughly 18 mph, and the energy absorption of the material system was not significantly affected by the change in test speed (from 4 mm/s for quasi-static to 8 m/s for dynamic). This is a positive result if PPMC were to be used in vehicles with a potential for high speed collisions.
On average the material system absorbed about 50 kJ/kg for t/D ratios spanning from 0.030 to 0.045. This is comparable to reported values of continuous fiber carbon/epoxy tubes with similar t/D ratios and layup stiffnesses.

From this review, it can be concluded that PPMC is a viable option for use on structural members of vehicles because it absorbs similar amounts of energy per unit mass to current structural members. In addition, with higher flow alignment, the material could outperform material systems currently in use for energy absorption.

A simulation methodology was also developed to create manufacturing informed performance models that can predict crash performance of composite components. The methodology used a building block approach that increased in complexity with each block of modeling. For example, the unidirectional material was characterized and used in a single-element model to confirm material model behavior. A four-element model was then created to understand progressive failure of elements in compression. From there, a crush tube with a continuous fiber laminate was modeled and compared to experiment. Once the UD model performance was confirmed, orientations of platelets were included in the model to determine the effect of platelet orientation distribution on energy absorption. This process created a simulation method that can test additional loading conditions and platelet orientation distributions to capture the influence of material flow and orientation on performance.

The manufacturing informed models for crush tubes matched experiment and were capable of extrapolating results in an orientation regime that was not investigated experimentally. The 3-point bend simulations produced flexural moduli that were slightly higher than experiment while the simulated strengths were lower than experiment. It is not clear why there is a discrepancy in flexural modulus, but local platelet orientation and out-of-plane orientation components are likely candidates.

While there is still work to be done regarding the flexure models, the simulation methodology was largely successful. Verifying and validating simulations on each building block led to predictive results for the SEA of PPMC tubes. Tuning parame-
ters were kept constant and the simulations were not adjusted and reran to compare to experiment as is commonly seen in industry.

The tools and processes presented in this work form a complete simulation package for creating manufacturing informed performance models in LS Dyna for PPMC components that are reliant on orientation information. Multiple sources of orientation information can be used, like CT scans, flow simulations, microscopy, and digitally generated components. An engineer working for an OEM could use this toolkit to analyze more complex geometries, new material systems, and different loading conditions.

7.2 Limitations in Crash Modeling

As was discussed throughout this work, there are limitations when using LS Dyna and the selected material models for crash simulations. The composite damage material models are only valid for shell elements because of their plane stress assumptions. This makes capturing through thickness effects and modeling individual platelets impossible. For example out of plane components of platelets are not included in the model. The material angles applied to integration points for shell elements are confined to in-plane angles only. Including this information in an LS Dyna model would reduce the estimated stiffness and potentially strength from simulation. The use of solid elements could also alleviate this problem to provide more accurate results, but it is not an available option in LS Dyna.

When implementing platelet orientations on integration points through the shell thickness, information regarding the platelet geometry is lost. The finite element model does not “know” the boundaries of a single platelet. This method relies on changes in local stiffness to provide information regarding platelet orientations and boundaries within the model.

The stability of the model is dependent upon the stiffness of the material and the mesh density, so engineers must be careful when adjusting either of these. However,
the stability of the model can be increased by reducing the stable time increment scaling factor. An example of this was seen with PPMC crush tubes as the level of platelet collimation increased beyond $A11 = 0.8$.

LS Dyna also has a known problem with repeatability in crash modeling, in that individual models can be run several times and produce different results each time. This is a common problem with explicit analyses, but the setting to correct this issue in LS Dyna is not completely effective. As a consequence, it is unclear whether the variability in simulation results are from the differences in orientation distribution from model to model or from the LS Dyna solver itself. A study quantifying the level of variability in LS Dyna results for a single model would be useful in understanding this problem.

LS Dyna does not have an acceptable method of dealing with brittle materials in compression when combined with element deletion. This is not a new issue, and alternatives were presented in this work and by Greve et al in 2008 [87]. It was found that a moving boundary condition could potentially solve the issue, but an investigation needs to be performed to understand if this boundary condition would have an impact on wave transfer or inertia within the system.

7.3 Future Work

While a test study was performed at dynamic speeds, additional high speed testing of PPMC tubes needs to be completed in order to quantify the variability of PPMC performance for each drop height and fill out the high rate regime. The amount of energy the fixture is absorbing through friction and contact also needs to be quantified.

Rate dependent mechanical properties were not incorporated into the material models. Therefore, simulating a high speed impact at this stage would only affect the magnitude of the impulse and not the material properties itself. The material properties need to be characterized at higher speeds to see if the change in test speed would affect energy absorption.
Platelet delamination was not incorporated into the finite element models even though it was a primary failure mechanism in crushing and flexure. Including cohesive elements drastically increases the computational and model prep time. However, a study should still be completed to investigate whether incorporating cohesive elements makes a significant difference in results.

Ideally, platelet boundaries could be added to the model by including multiple layers of shell elements and resin rich pockets at the end of platelets as is seen experimentally. The resin rich pockets would potentially provide locations of lower stiffness more likely to be a failure site. This would require the applications and tools presented in this work to be adjusted for the added complexity.

Altering the manufacturing method to incorporate thermoplastic PPMC materials would be an interesting addition to this work. Thermoplastics generally absorb more energy than thermoset materials and confirming this trend with PPMC materials would be beneficial. Attempting to identify a connection between the fracture toughness and specific energy absorption would be useful for researchers studying crash performance of composites.
REFERENCES


A. EXAMPLE LS DYNA KEYWORD FILE

This appendix provides an example of a continuous fiber crush tube model in LS Dyna. The element and node definitions are left out for brevity.

```
$# LS-DYNA Keyword file created by LS–PrePost(R) V4.3 – 12Dec2016 (05:00)
$# Created on May–10–2019 (14:23:12)

*KEYWORD
*TITLE
$#
LS-DYNA keyword deck by LS–PrePost

*CONTROL_ACCURACY
$# osu inn pidosu iacc
0 2 0 0

*CONTROL_CONTACT
$# sfsfac rwpnal islchk shlthk penopt thkchg orien enmass
0.1 0.0 1 0 1 0 1 0

$# usrstr usrfrc nsbcis interm xpene sssthk ecdt tiedprj
0 0 0 0 4.0 0 0 0

$# sfric dfrc ic ed vfc th th sf pen sf
0.0 0.0 0.0 0.0 0.0 0.0 0.0

$# ignore frceg skiprg outseg spotsp spotdl spothin
0 1 0 0 0 1 0 0

$# isym nserod rwgaps rwgdtb rwrksf icov swradf ithoff
0 0 1 0.0 1.0 0 0.0 0

$# shledg pstiff ithcnt tdcnof ftall unused shltrw
0 0 0 0 1 0 0

*CONTROL_ENERGY
$# hgen rwen snten rylent
2 2 2 2

*CONTROL_HOURGLASS
$# ihq qh
4 0.1

*CONTROL_PARALLEL
$# ncpu numrhs const para
4 0 1 0

*CONTROL_SHELL
$# wrpang esort irnxx istudp theory bwc miter proj
7.0 0 –1 1 2 2 1 0
```
$# rotascl  intgrd  lamsht  csty6  tshell
  0.0  0  1  1  0
$# psstupd  sidt4tu  cntco  itsflg  irquad
  0  0  0  0  2
$# nfail1  nfail4  psnfail  keepcs  deffr  drcpsid  drcprm
  0  0  0  0  0  0  1.0
*CONTROL_TERMINATION
$# endtim  endcyc  dtmin  endeng  endmas
  0.125  0  0.0  0.01.000000E8
*CONTROL_TIMESTEP
$# dtinit  tssfac  isdo  tslimt  dt2ms  lctm  erode  msist
  5.00000E−6  0.9  0  0.0  0.0  0  0  0
$# dt2msf  dt2mslc  imscl  unused  unused  unused  rmscl
  0.0  0  0  0.0
*CONTROL_UNITS
$# length  time  mass  temp
  0.0  0.0  0.0  0.0
*DATABASE_RWFORC
$# dt  binary  lcur  ioopt
  1.00000E−8  0  0  1
*DATABASE_BINARY_D3PLOT
$# dt  ldcdt  beam  npltc  psetid
  1.00000E−5  0  0  0  0
$# ioopt
  0
*DATABASE_EXTENT_BINARY
$# neiph  neips  maxint  strflg  sigflg  epsflg  rltflg  engflg
  0  16  3  1  1  1  1  1
$# cmpflg  ieverp  beamip  dcomp  shge  stssz  n3thdt  ialemat
  1  0  0  1  1  1  2  1
$# nintslid  pkpSen  sclp  hydro  msscl  therm  intout  nodout
  0  0  1.0  0  0  0  0  0
$# dtdt  resplt  neipb
  0  0  0
*DATABASE_HISTORY_SHELL_SET
$# id1  id2  id3  id4  id5  id6  id7  id8
  5  0  0  0  0  0  0  0
*BOUNDARY_SPC_SET_ID
$# id
  1FixedTopNodes
$# nsid  cid  dofx  dofy  dofz  dofrx  dofry  dofrz
  7  0  1  1  1  1  1  1
*SET_NODE_LIST_TITLE
TopRowNodesTube
```
<table>
<thead>
<tr>
<th>cid</th>
<th>title</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>Contact Definition</td>
</tr>
</tbody>
</table>

```
Cylinder

$\# \text{secid elform shrif nip propt qr/irid icomp setyp}$
1 2 1.0 5 3.0 $\text{-2020}$ 1 1

$\# t1 t2 t3 t4 nloc marea idof edgset$
0.016 0.0016 0.0016 0.0016 $\text{-1.0}$ 0.0 0.0 0

$\# bi bi bi bi bi bi bi bi$
0.0 90.0 90.0 90.0 90.0 0.0 0.0 0.0

**MAT ENHANCED COMPOSITE DAMAGE TITLE**
MAT054_FordDowMat

$\# \text{mid ro ea eb ( ec ) prba ( prca ) ( prcb )}$
2 1800.01.40000E117.440000E97.440000E9 0.017 0.017 0.017

$\# \text{gab gbc gca ( kf ) aopt 2way}$
5.730000E95.730000E95.730000E9 0.0 2.0 0.0

$\# xp yp zp a1 a2 a3 mangle$
0.0 0.0 0.0 0.0 0.0 1.0 0.0

$\# vl v2 v3 d1 d2 d3 dfailm dfails$
0.0 0.0 0.0 0.0 1.0 0.0 0.053 0.23

$\# \text{tfail alph soft fbrt yfac dfail t dfails}$
1.000000E$-9$ 0.0 0.5 0.0 2.0 0.014 $\text{-0.018}$ 0.0

$\# xc xt yc yt sc crit beta$
1.110000E91.654000E92.350000E85.860000E79.170000E7 54.0 1.0

$\# pel epsf epsr tsmd soft2$
100.0 0.0 0.0 0.0 0.0 0.5

$\# slim1 slimc1 slimt2 slimc2 slims ncyred softg$
0.2 0.8 0.2 0.8 1.0 0.0 1.0

**PART**

$\# \text{title}$

BevelLower

$\# \text{pid secid mid eosid hgid grav adpopt tmid}$
4 2 2 0 0 0 0 0

**SECTION SHELL TITLE**

BevelLower

$\# \text{secid elform shrif nip propt qr/irid icomp setyp}$
2 2 1.0 5 3.0 $\text{-2020}$ 1 1

$\# t1 t2 t3 t4 nloc marea idof edgset$
5.000000E$-45.000000E-45.000000E-45.000000E-4 $\text{-1.0}$ 0.0 0.0 0

$\# bi bi bi bi bi bi bi bi$
0.0 90.0 90.0 90.0 90.0 0.0 0.0 0.0

**PART**

$\# \text{title}$

BevelUpper

$\# \text{pid secid mid eosid hgid grav adpopt tmid}$
**SECTION_SHELL_TITLE**
BevelUpper

```
$#  secid  elform  shrf  nip  propt  qr/irid  icomp  setyp
  3  2  1.0  5  3.0  -2020  1  1
$#  t1  t2  t3  t4  nloc  marea  idof  edgset
  7.50000E-47.50000E-47.50000E-47.50000E-4  -1.0  0.0  0.0  0
$#  bi  bi  bi  bi  bi  bi  bi  bi
  0.0  90.0  90.0  90.0  90.0  90.0  0.0  0.0
```

**MAT_RIGID_TITLE**
Rigid

```
$#  mid  ro  e  pr  n  couple  m  alias
  1  36600002.07000E11  0.3  0.0  0.0  0.0
$#  cmo  con1  con2
  0.0  0  0
$#  lco  or  a1  a2  a3  v1  v2  v3
  0.0  0.0  0.0  0.0  0.0  0.0  0.0
```

**RIGIDWALL_GEOMETRIC_FLAT_MOTION_ID**

```
$#  id  title
  1Rigid Wall
$#  nsid  nsidex  boxid  birth  death
  0  0  0  0.01.00000E20
$#  xt  yt  zt  xh  yh  zh  fric
  0.0  0.0  0.0  0.0  0.0  1.0  0.0
$#  xhev  yhev  zhev  lenl  lenm
  1.0  0.0  0.0  0.0
$#  lcid  opt  vx  vy  vz
  1  1  0.0  0.0  1.0
```

**DEFINE_CURVE_TITLE**
Displacement Curve

```
$#  lcid  sidr  sfa  sfo  offa  offo  dattyp  lcint
  1  0  1.0  1.0  0.0  0.0  0  0
$#  al  ol
  0.0
  0.125  0.04
```

**SET_NODE_LIST_TITLE**
CrushTube

```
$#  sid  dal  da2  da3  da4  solver
  3  0.0  0.0  0.0  0.0  0.0MECH
$#  nid1  nid2  nid3  nid4  nid5  nid6  nid7  nid8
  121  122  123  124  125  126  127  128
  129  130  131  132  133  134  135  136
...```

**SET_NODE_LIST_TITLE**
BevelLower
$\# \ s i d \ da1 \ da2 \ da3 \ da4 \ solver$
4 0.0 0.0 0.0 0.0MECH
$\# \ nid1 \ nid2 \ nid3 \ nid4 \ nid5 \ nid6 \ nid7 \ nid8$
1 2 3 4 5 6 7 8
9 10 11 12 13 14 15 16
...
*SET_NODE_LIST_TITLE
BevelUpper
$\# \ s i d \ da1 \ da2 \ da3 \ da4 \ solver$
5 0.0 0.0 0.0 0.0MECH
$\# \ nid1 \ nid2 \ nid3 \ nid4 \ nid5 \ nid6 \ nid7 \ nid8$
61 62 63 64 65 66 67 68
69 70 71 72 73 74 75 76
...
*SET_NODE_LIST_TITLE
BottomNodes
$\# \ s i d \ da1 \ da2 \ da3 \ da4 \ solver$
8 0.0 0.0 0.0 0.0MECH
$\# \ nid1 \ nid2 \ nid3 \ nid4 \ nid5 \ nid6 \ nid7 \ nid8$
1 2 3 4 5 6 7 8
9 10 11 12 13 14 15 16
...
*SET_PART_LIST_TITLE
Tube
$\# \ s i d \ da1 \ da2 \ da3 \ da4 \ solver$
2 0.0 0.0 0.0 0.0MECH
$\# \ pid1 \ pid2 \ pid3 \ pid4 \ pid5 \ pid6 \ pid7 \ pid8$
1 4 5 0 0 0 0 0
*SET SHELL LIST_TITLE
BevelLower
$\# \ s i d \ da1 \ da2 \ da3 \ da4$
3 0.0 0.0 0.0 0.0
$\# \ eid1 \ eid2 \ eid3 \ eid4 \ eid5 \ eid6 \ eid7 \ eid8$
1 2 3 4 5 6 7 8
9 10 11 12 13 14 15 16
...
*SET SHELL LISTITLE
BevelUpper
$\# \ s i d \ da1 \ da2 \ da3 \ da4$
4 0.0 0.0 0.0 0.0
$\# \ eid1 \ eid2 \ eid3 \ eid4 \ eid5 \ eid6 \ eid7 \ eid8$
61 62 63 64 65 66 67 68
69 70 71 72 73 74 75 76
For Print out

<table>
<thead>
<tr>
<th>sid</th>
<th>da1</th>
<th>da2</th>
<th>da3</th>
<th>da4</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>eid1</th>
<th>eid2</th>
<th>eid3</th>
<th>eid4</th>
<th>eid5</th>
<th>eid6</th>
<th>eid7</th>
<th>eid8</th>
</tr>
</thead>
<tbody>
<tr>
<td>4070</td>
<td>4071</td>
<td>4072</td>
<td>4073</td>
<td>4074</td>
<td>4075</td>
<td>4076</td>
<td>4077</td>
</tr>
<tr>
<td>4078</td>
<td>4079</td>
<td>4080</td>
<td>4081</td>
<td>4082</td>
<td>4083</td>
<td>4084</td>
<td>4085</td>
</tr>
</tbody>
</table>

GLOBAL Damping

<table>
<thead>
<tr>
<th>lcid</th>
<th>valdmp</th>
<th>stx</th>
<th>sty</th>
<th>stz</th>
<th>srx</th>
<th>sry</th>
<th>srz</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.3</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

Part Stiffness Damping

<table>
<thead>
<tr>
<th>pid</th>
<th>coef</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.1</td>
</tr>
<tr>
<td>4</td>
<td>0.1</td>
</tr>
<tr>
<td>5</td>
<td>0.1</td>
</tr>
</tbody>
</table>

Integration Shell

<table>
<thead>
<tr>
<th>irid</th>
<th>nip</th>
<th>esop</th>
<th>failopt</th>
</tr>
</thead>
<tbody>
<tr>
<td>2020</td>
<td>5</td>
<td>1</td>
<td>0</td>
</tr>
</tbody>
</table>

Element Shell

<table>
<thead>
<tr>
<th>eid</th>
<th>pid</th>
<th>n1</th>
<th>n2</th>
<th>n3</th>
<th>n4</th>
<th>n5</th>
<th>n6</th>
<th>n7</th>
<th>n8</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4</td>
<td>1</td>
<td>2</td>
<td>62</td>
<td>61</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>2</td>
<td>3</td>
<td>63</td>
<td>62</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

Node

<table>
<thead>
<tr>
<th>nid</th>
<th>x</th>
<th>y</th>
<th>z</th>
<th>tc</th>
<th>rc</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0</td>
<td>0.0254</td>
<td>0.0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>-0.002655</td>
<td>0.025261</td>
<td>0.0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

END
B. DROP TOWER TEST FIXTURE SCHEMATICS

Figure B.1. Schematic of bottom plate for drop tower test fixture
Figure B.2. Schematic of middle plate for drop tower test fixture
Figure B.3. Schematic of top plate for drop tower test fixture
VITA

Rebecca Cutting was born and raised in Baytown, Texas. She attended Purdue University and obtained a Bachelor’s of Science in Aeronautics and Astronautics in May 2013. While completely her undergraduate studies, she participated in a 5-term co-op experience with ATA Engineering, Inc. and was introduced to finite element modeling. During her junior year in college, Rebecca took a class taught by Dr. Johnathan E. Goodsell (then a PhD student of Dr. R. Byron Pipes), and she started working on composites research related to performance of open hole unidirectional composites.

After obtaining her BS AAE, she completed an internship with the structural dynamics lab at The Boeing Company in Tukwila, Wa. There she was introduced to dynamic testing methods and test-analysis correlation. Rebecca came back to Purdue University in the fall of 2013 to work on her Master’s in Aeronautics and Astronautics under the supervision of Dr. R. Byron Pipes. Her research focused on the nondestructive evaluation of prepreg platelet molding systems using dynamic analysis. She completed another internship with the structural dynamics lab at Boeing before finishing her MS AAE in spring 2015 and returning to Boeing as a full time test engineer.

In the fall of 2016, Rebecca returned to West Lafayette to work for Dr. R. Byron Pipes as a validation engineer at the newly formed Composites Manufacturing and Simulation Center in Purdue Research Park. In addition to her engineering responsibilities, she has been researching the crash performance of prepreg platelet composites as part of her dissertation work. Rebecca defended her dissertation in July 2019.