Title
Experimental and Numerical Investigation of Wide Area Blunt Impact Damage to Composite Aircraft Structures

Permalink
https://escholarship.org/uc/item/9431812g

Author
Chen, Zhi Ming

Publication Date
2015

Peer reviewed|Thesis/dissertation
UNIVERSITY OF CALIFORNIA, SAN DIEGO

Experimental and Numerical Investigation of Wide Area Blunt Impact Damage to
Composite Aircraft Structures

A Dissertation submitted in partial satisfaction of the requirements for the degree Doctor
of Philosophy

in

Structural Engineering

by

Zhi Ming Chen

Committee in charge:

Hyonny Kim, Chair
David Benson
Olivia A. Graeve
Francesco Lanza di Scalea
Vlado Lubarda

2015
The Dissertation of Zhi Ming Chen is approved, and it is acceptable in quality and form for publication on microfilm and electronically:

________________________________________________________________________
________________________________________________________________________
________________________________________________________________________
________________________________________________________________________

________________________________________________________________________
________________________________________________________________________
________________________________________________________________________
________________________________________________________________________

University of California, San Diego
2015
TABLE OF CONTENTS

Signature Page.................................................................................................................................. iii
Table of Contents............................................................................................................................ iv
List of Figures................................................................................................................................... v
List of Tables.................................................................................................................................... xv
Acknowledgement....................................................................................................................... xvi
Vita.................................................................................................................................................... xvii
Abstract of the Dissertation........................................................................................................... xix
Chapter 1. Introduction and Literature Review........................................................................... 1
Chapter 2. Large-Scale Experiments ............................................................................................. 17
Chapter 3. Small-Scale Experiments and Modeling ..................................................................... 76
Chapter 4. Modeling of Large-Scale Experiments......................................................................... 148
Chapter 5. Conclusions................................................................................................................ 178
References ...................................................................................................................................... 184
Appendix. Additional Data from Large-Scale Fuselage Impact Experiments......................... 189
# LIST OF FIGURES

| Figure 1.1 | A luggage loader operating near an aircraft | 3 |
| Figure 1.2 | Building block approach to composite product development | 5 |
| Figure 1.3 | Road map for approaching experiments and numerical simulations of high-energy blunt impact | 6 |
| Figure 1.4 | Typical impact energy vs. damage size curve | 9 |
| Figure 2.1 | Overview of specimen design | 19 |
| Figure 2.2 | Stringer and skin cross-sectional geometry and stringer-skin connection details (in mm). The shear ties are omitted from this figure | 21 |
| Figure 2.3 | Shear tie cross-sectional geometry and connection to the skin (in mm) | 22 |
| Figure 2.4 | C-frame cross-sectional geometry and connection to shear tie (in mm) | 22 |
| Figure 2.5 | Front view of the two specimen test configurations (a) two-stringer specimen loaded between the stringers; (b) three-stringer specimen loaded on the middle stringer | 24 |
| Figure 2.6 | The 76 mm wide impactors: (a) solid aluminum with 76.2 mm radius of curvature, and (b) TUG rubber bumper with a non-constant, approximately 102 mm radius of curvature | 26 |
| Figure 2.7 | StringerXX test setup | 27 |
| Figure 2.8 | Test setup showing impactor and boundary conditions | 28 |
| Figure 2.9 | Contact between the aluminum indenter and Stringer01's external skin | 30 |
| Figure 2.10 | Stringer01 contact force vs. indentation depth plots | 30 |
| Figure 2.11 | Stringer01 local skin penetration | 31 |
| Figure 2.12 | Stringer01 skin penetration and delamination damage | 31 |
Figure 2.13 Contact between the rubber bumper and Stringer02's external skin.......................................................... 32
Figure 2.14 Stringer02 contact force vs. skin indentation plots.......................... 33
Figure 2.15 Microphotography showing radial delamination of a shear tie..... 34
Figure 2.16 Stringer02 after fourth loading. White boxes indicate area of delamination damage incurred during 3rd loading. Dashed red box indicates area of delamination incurred during 4th loading..... 35
Figure 2.17 Stringer02 cumulative crushing damage on one of the shear ties after the fourth loading, the damage is circled in red............. 35
Figure 2.18 Stringer02 and Stringer05 contact force vs. skin indentation plots.......................................................... 37
Figure 2.19 High speed video still frames of Stringer05 test as viewed from the panel's interior. (a) t = 17.2 msec, (b) t = 199.0 msec, (c) t = 199.6 msec, and (d) t = 202.4 msec after impact.................. 38
Figure 2.20 Post-test state of Stringer05 showing surface cracks along the skin-stringer connection at the impact location....................... 40
Figure 2.21 Center image: post-test A-scan map of Stringer05; side images: crack formations along the stringer radii and on the flanges viewed from the interior side............................................. 40
Figure 2.22 Stringer00 and 01 final loading contact force vs. skin indentation plots........................................................................ 41
Figure 2.23 Primary load paths of the two-stringer and three-stringer specimen configurations based on the location of loading........... 42
Figure 2.24 Final delamination area of (a) Stringer02 and (b) Stringer03.... 43
Figure 2.25 Cross-section view of a stringer-reinforced skin being loaded over the stringer, concentrated stress is expected to develop at the skin-stringer joint......................................................... 43
Figure 2.26 Skin cracking and penetration damage observed during quasi-static indentation of Stringer00 with load applied on a stringer location. The initial crack was formed along the skin-stringer joint........................................................................................................... 44
<p>| Figure 2.27 | Illustration of the experiments after the D-shaped bumper has collapsed | 46 |
| Figure 2.28 | Side view of the panel. The impact experiments can be idealized into two dimensional beam bending of the stringers, the shear force diagram of stringer is also shown | 48 |
| Figure 2.29 | Time-dependent deformation response of a dynamically impacted beam | 49 |
| Figure 2.30 | Strain vs. actuator displacement data taken at the impact zone (skin bending) and at the boundary area (shear tie compression) for both the quasi-static (Stringer02) and dynamic (Stringer05) experiments | 50 |
| Figure 2.31 | Side view of StringerXX panel: (a) before impact and (b) during impact. Both downward compression and moment loading were exerted on the shear ties | 52 |
| Figure 2.32 | General trend of damage based on the StringerXX series experiments | 54 |
| Figure 2.33 | Isometric view of the Frame01 Panel test setup. Steel boundary conditions were attached to the ends of the C-frames | 56 |
| Figure 2.34 | Frame01 panel test setup with the cylindrical bumper indenter | 57 |
| Figure 2.35 | Top view of Frame01 panel configuration, the spacing between C-frames is 508 mm | 58 |
| Figure 2.36 | (a) Side view the Frame01 with compression loading applied directly to the shear ties, and (b) fracture of the shear tie (F01H) at the curved corner after the second loading | 59 |
| Figure 2.37 | Contact force vs. skin indentation of Frame01. The skin indentation was measured in the middle of the loading zone | 60 |
| Figure 2.38 | Post third loading major damage: interior view of Frame01 | 61 |
| Figure 2.39 | Top view of Frame03 and 04 panel configuration, the spacing between C-frames is 457 mm | 62 |
| Figure 2.40 | Experimental setup for Frame03 and 04 dynamic experiments | 63 |</p>
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.41</td>
<td>Actuator displacement history plots for Frame03 specimen</td>
<td>64</td>
</tr>
<tr>
<td>2.42</td>
<td>Frame03 contact force vs. skin indentation plots</td>
<td>65</td>
</tr>
<tr>
<td>2.43</td>
<td>Shear tie corner crushing failure. The shear tie was painted white to visually accentuate cracks</td>
<td>65</td>
</tr>
<tr>
<td>2.44</td>
<td>High speed video stills of Frame03 2nd Loading</td>
<td>68</td>
</tr>
<tr>
<td>2.45</td>
<td>Frame and shear tie damage of Frame03 after 2nd loading</td>
<td>69</td>
</tr>
<tr>
<td>2.46</td>
<td>Frame03 C-frame fracture failures near boundary supports</td>
<td>69</td>
</tr>
<tr>
<td>2.47</td>
<td>Frame03 external skin after sustaining internal C-frame damage</td>
<td>70</td>
</tr>
<tr>
<td>2.48</td>
<td>Frame04-1 and Frame03 contact force vs. skin indentation plots</td>
<td>71</td>
</tr>
<tr>
<td>2.49</td>
<td>The aluminum 7075 shear ties (a) mounted to the Frame04 panel, and (b) overall dimensions</td>
<td>72</td>
</tr>
<tr>
<td>2.50</td>
<td>Frame04-2 local shear failure of the C-frames under the impactor</td>
<td>73</td>
</tr>
<tr>
<td>2.51</td>
<td>Contact force vs. skin indentation plot of Frame04-2</td>
<td>74</td>
</tr>
<tr>
<td>3.1</td>
<td>Analysis of the fuselage panel failure modes</td>
<td>77</td>
</tr>
<tr>
<td>3.2</td>
<td>Traction separation response of cohesive zone modeling</td>
<td>81</td>
</tr>
<tr>
<td>3.3</td>
<td>Dimensions of the ENF specimen. The pre-crack is represented by the red line</td>
<td>84</td>
</tr>
<tr>
<td>3.4</td>
<td>Setup for ENF specimens. The span of specimen was 60 mm and the starter crack was 15 mm</td>
<td>85</td>
</tr>
<tr>
<td>3.5</td>
<td>Force vs. cross head displacement plot of the ENF coupons (a = 15 mm, 2L = 60 mm)</td>
<td>86</td>
</tr>
<tr>
<td>3.6</td>
<td>2D wireframe view of the ENF model with its boundary conditions. The red line represents the surface where the cohesive surface interaction was defined</td>
<td>87</td>
</tr>
</tbody>
</table>
Figure 3.7  ENF Model deformation at the time of initial crack growth (the crack tip nodal pair is circled)........................................................ 89
Figure 3.8  Color contour plot showing the cohesive surface damage parameter at the time of initial crack growth....................................... 89
Figure 3.9  Force vs. cross head displacement plot of the ENF coupons and FE model (a = 15 mm, 2L = 60 mm)...................................................... 90
Figure 3.10  Peak force vs. crack tip element length for the ENF model................. 91
Figure 3.11  Illustration of StringerXX panel: (a) before impact and (b) during impact......................................................................................... 92
Figure 3.12  Illustration of FrameXX panel: (a) before impact and (b) during impact............................................................................................ 93
Figure 3.13  Dimensions of the curved beam coupons. The diameter of the bolt holes was 6.35 mm................................................................. 94
Figure 3.14  Curve beam opening test setup, the curved beam was painted partially white..................................................................................... 95
Figure 3.15  Free body diagram of the curved beam loading.............................. 97
Figure 3.16  Microphotography showing radial delamination of a curved beam coupon at multiple interfacial layers. Three of the crack also showed bridging into the fabric plies (cracks in the radial direction)........................................................................ 98
Figure 3.17  Moment vs. deflection plots for the curved beam coupons............. 99
Figure 3.18  (a) Curve beam layup, (b) sublamine (ply group) configuration for modeling.......................................................................................... 101
Figure 3.19  Half symmetry curve beam model with a 6 element layer construction. The orange mesh represents the moment arms, and the red lines represent the cohesive interface zones......................... 102
Figure 3.20  (a) Elastic bending response of the curved beam model, and (b) the model after delamination failure.............................................. 104
Figure 3.21  Moment vs. deflection plot of the curved beam coupons and the FE Model.......................................................................................... 105
Figure 3.22 Close up view of the delamination simulation of the curve beam model.

Figure 3.23 Shear tie compression coupon dimensions. The diameter of the bolt holes was 6.35 mm.

Figure 3.24 Shear tie compression coupon test setup (specimen painted white).

Figure 3.25 Simplified free-body diagram of the shear tie compression experiment.

Figure 3.26 The shear force and moment diagrams of the shear tie’s curved corner.

Figure 3.27 Force vs. crosshead displacement plot of a shear tie compression coupon.

Figure 3.28 (a) Elastic deformation of shear tie compression coupon, (b) initial delamination at the curved corner, (c) crushing failure and flattening of the corner, (d) post-buckling bending failure of the flange.

Figure 3.29 Modified stress-strain relationship governing the damage degradation relationship.

Figure 3.30 Shear tie compression model with a 6 element layer construction. The orange meshes represent the aluminum base plate, and the red lines represent the cohesive zone interfaces.

Figure 3.31 (a) Unloaded shear tie continuum shell model, (b) initial delamination at shear tie corner, (c) crushing failure and flattening of the corner, (d) buckling failure of the flange with large distortion of the corner elements.

Figure 3.32 Load vs. displacement plots of the shear tie compression experimental and 6 layer continuum shell FE model results.

Figure 3.33 Updated shear tie model configuration, the red lines are cohesive zone interfaces.
Figure 3.34  (a) Unloaded shear tie solid model, (b) close up view of the initial delamination at shear tie corner, (c) flattening of the shear tie corner, and (d) crushing failure and severing at the shear tie corner............................................................. 123

Figure 3.35  Load vs. displacement plots of the shear tie compression experimental results and 12 layer solid FE model results............ 124

Figure 3.36  Compression force vs. flange tip displacement plot for the shear tie model with different number of element layers through the thickness........................................................................... 125

Figure 3.37  Mesh sensitivity results of the shear tie models.................. 126

Figure 3.38  A D-shaped rubber bumper. The bumper's base width and depth were 198 mm and 76 mm, respectively.............................. 127

Figure 3.39  D-shaped bumper compression test setup (left) and fully collapsed bumper (right)........................................................................ 128

Figure 3.40  D-shaped bumper compression test data................................. 128

Figure 3.41  Half model of the bumper compression experiment................ 129

Figure 3.42  The fully collapsed rubber bumper model (with the Ogden material definition).............................................................................. 132

Figure 3.43  D-shaped bumper compression test data and FE result............... 132

Figure 3.44  Dimetric view of the stringer element specimens cut from a pristine StringerXX panel............................................................. 134

Figure 3.45  Cross-sectional view of the stringer element specimen showing its potted region............................................................... 135

Figure 3.46  (a) The stringer element specimen potted in Polyepoxy in an aluminum C-channel, and (b) Quasi-static indentation test setup of the stringer element........................................................ 135

Figure 3.47  Free body diagram of the stringer element subjected to quasi-static indentation against the bumper. The locations of maximum bending and shear stress are marked with red squares.. 136

Figure 3.48  Bending stress and shear stress profiles through the laminate skin thickness........................................................................ 137
Figure 3.49  Load vs. displacement plot of the shear tie Compression experimental results.............................................................................................. 138

Figure 3.50  SE01 after skin crack failure; with a glancing light source, two visible skin cracks were found at the skin-stringer joining locations........................................................................................................ 139

Figure 3.51  Photomicrographs of the SE01 specimen: (a) before the test, and (b) after the test, with ply cracks and delaminations at the skin-stringer joining location.................................................................................. 140

Figure 3.52  SE02 with through-thickness skin crack at one of the skin-stringer joint location.............................................................................................................................................. 141

Figure 3.53  Finite element model of the stringer element indentation test...... 142

Figure 3.54  Load vs. displacement plot of the stringer element experimental results and FE Model.................................................................................................................. 144

Figure 3.55  Stringer element model's deformed shape at failure................. 144

Figure 3.56  External ply damage of the SE01 based on Hashin-Rotem failure criteria................................................................................................................................. 145

Figure 4.1  Cross-section view of the Stringer05 half model......................... 149

Figure 4.2  Side view of the Stringer05 half model, the overhanging skin and stringer past at the ends of the panel were omitted from this figure.............................................................................................................. 150

Figure 4.3  Cross-section view of the skin-stringer joint showing the cohesive zone interface area between the skin and stringer (red line)...................................................................................................................................... 153

Figure 4.4  Side view of the 4-layer shear tie showing the cohesive zone interface area between the element layers......................................................... 153

Figure 4.5  Stringer05 model: (a) unloaded, and (b) at peak bumper displacement. The steel plate above the bumper was omitted from the view.............................................................................................................. 154
Figure 4.6  Side view of the Stringer05 model at peak displacement. The steel plate above the bumper was omitted from the view.

Figure 4.7  Comparison of Stringer05 experimental data and FE model result.

Figure 4.8  Stringer05 (a): cohesive surface damage in the shear tie, and (b): the exterior view of the same shear tie in the experiment showing surface undulations.

Figure 4.9  Stringer05 (a): Hashin ply failure by fiber tension at the skin's external surface, and (b): the exterior view of panel skin with the though-thickness skin crack.

Figure 4.10 Stringer05 (a): Hashin ply failure by fiber compression at the stringer's surface, and (b): interior view of the panel's stringer with the buckling induced crack.

Figure 4.11 Stringer05 (a): cohesive surface damage in the skin-stringer joint, and (b): the exterior view of panel skin with the skin-stringer delamination (white hatched zone).

Figure 4.12 Frame03/04 baseline FE model.

Figure 4.13 Front view of the Frame03/04 baseline FE model.

Figure 4.14 Load history of the combined Frame03 and Frame04-1 tests and baseline FEA Model.

Figure 4.15 Close-up view of Frame03 shear tie buckling failure as predicted by the baseline model.

Figure 4.16 The updated Frame03/04 model with the six center shear ties replaced by refined models.

Figure 4.17 Contact force vs. indentation plot of the Frame03 test and FE model with shell element shear ties.

Figure 4.18 Close-up view of the impact shear tie in the FrameXX model. Shear tie crushing damage modeled with continuum shell elements, before the simulation crashed.

Figure 4.19 Contact force vs. indentation plot of the Frame03 test and FE model with solid element shear ties.
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.20</td>
<td>Close-up view of the shear tie fracture failure simulated with solid elements</td>
</tr>
<tr>
<td>4.21</td>
<td>Updated Frame03 half model: (a) impacted shear tie corner fracture damage, (b) C-frame twisting, and (c) adjacent shear tie fracture at the corner</td>
</tr>
<tr>
<td>4.22</td>
<td>Frame03 experiment: (a) impacted shear tie corner fracture damage, (b) C-frame twisting, and (c) adjacent shear tie bending failure at the bolt line</td>
</tr>
<tr>
<td>4.23</td>
<td>Side views of the Frame03/04 model showing increasing degrees of C-frames twisting</td>
</tr>
<tr>
<td>4.24</td>
<td>(a) A cross-section view of the Frame03/04 model summarizing all the predicted damage along a C-frame, and (b) a photograph of the Frame03 specimen that shows similar failures along a C-frame</td>
</tr>
<tr>
<td>A.1</td>
<td>Stringer03 contact force vs. actuator displacement plots</td>
</tr>
<tr>
<td>A.2</td>
<td>Barely visible skin cracks (between the two white lines)</td>
</tr>
<tr>
<td>A.3</td>
<td>Stringer crack at the curved corner between the stringer flange and diagonal wall</td>
</tr>
<tr>
<td>A.4</td>
<td>The exterior of the Stringer03 panel after final failure</td>
</tr>
<tr>
<td>A.5</td>
<td>Stringer03 and 06 contact force vs. actuator displacement plots</td>
</tr>
<tr>
<td>A.6</td>
<td>Through-thickness skin cracks of the Stringer06 specimen</td>
</tr>
</tbody>
</table>
## LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 2.1</td>
<td>Laminate Layup Information</td>
<td>23</td>
</tr>
<tr>
<td>Table 2.2</td>
<td>StringerXX specimen test setup summary</td>
<td>26</td>
</tr>
<tr>
<td>Table 3.1</td>
<td>X840 Z60 carbon/epoxy lamina elastic properties</td>
<td>79</td>
</tr>
<tr>
<td>Table 3.2</td>
<td>X840 Z60 carbon/epoxy lamina in-plane material failure strengths in tension and compression</td>
<td>79</td>
</tr>
<tr>
<td>Table 3.3</td>
<td>X840 Z60 PW carbon/epoxy lamina in-plane fracture energies (amount of energy required to create a new crack in the material) in tension and compression</td>
<td>79</td>
</tr>
<tr>
<td>Table 3.4</td>
<td>Cohesive surface interaction failure parameters</td>
<td>82</td>
</tr>
<tr>
<td>Table 3.5</td>
<td>Ogden material parameters N = 2</td>
<td>131</td>
</tr>
<tr>
<td>Table A.1</td>
<td>StringerXX panels result summary</td>
<td>193</td>
</tr>
<tr>
<td>Table A.2</td>
<td>FrameXX quasi-static, 3-frame panels result summary</td>
<td>194</td>
</tr>
<tr>
<td>Table A.3</td>
<td>FrameXX dynamic (0.5 m/sec), 5-frame panels result summary</td>
<td>194</td>
</tr>
</tbody>
</table>
ACKNOWLEDGEMENTS

I would like to thank my adviser Professor Hyonny Kim, for his guidance and support throughout my long and arduous journey in graduate school. His technical expertise and immense patience helped me overcome many obstacles. I would also like to thank my fellow graduate and undergraduate research students. The large-scale impact experimental work presented in this dissertation is a joint effort with another graduate student, Gabriela DeFrancisci. In addition, I am grateful to Daniel Whisler, Sho Funai, Sara White, Monica Chan, Jonathan Hughes, Sean Luong, Isabella Cole, and Sarah Fung for the assistance they have provided. Finally, I would like to thank the FAA for providing the funding for the research, and aerospace industry partners for their insights that helped shaped the project.

Chapter 2, in part, contains materials appearing in the following papers: Ground Vehicle Blunt Impact Damage Formation to Composite Aircraft Structures 2012, Kim, Hyonny; DeFrancisci, Gabriela; and Chen, Zhi Ming, Springer Netherlands 2012. Estimation of Ground Service Equipment Contact Damage in Composite Aircraft Panels 2012, Kim, Hyonny; DeFrancisci, Gabriela; and Chen, Zhi Ming, AIAA 2012. The dissertation author was a co-investigator and co-author of these papers.

Chapter 4, in part, is a reprint of the materials appearing in Experimental and Modeling Investigation of Blunt Impact to Stringer-Reinforced Composite Panels 2013, Chen, Zhi Ming; Kim, Hyonny; DeFrancisci, Gabriela, Destech Publications 2013. The dissertation author was the primary investigator and author of this paper.
VITA

2009   Bachelor of Science, University of California, San Diego
2009-2015  Graduate Student Researcher and Teaching Assistant, University of California, San Diego
2011   Master of Science, University of California, San Diego
2014   Student Research Intern, NASA Langley Research Center
2015   Doctor of Philosophy, University of California, San Diego

FIELD OF STUDY

Structural Engineering, with focus on Mechanics of Composite Materials

PUBLICATIONS


ABSTRACT OF THE DISSERTATION

Experimental and Numerical Investigation of Wide Area Blunt Impact Damage in Composite Aircraft Structures

by

Zhi Ming Chen

Doctor of Philosophy in Structural Engineering

University of California, San Diego, 2015

Professor Hyonny Kim, Chair

Due to their high performance and weight efficiency, carbon fiber composites are increasingly being used in aircraft primary structure applications. Exposed composite structures (e.g., fuselage lower body) are susceptible accidental impacts by ground service equipment (GSE). The very high mass (over 10,000 kg) of GSE impact can involve high energy (over 1000 J) and thus can induce significant internal damage. Furthermore, the large contact area potentially involved with GSE impact can create significant internal delamination and fiber failure without leaving exterior-visible signs that any damage has occurred. The objectives of the research described herein are to: (1) conduct experimental investigation into the composite aircraft damage caused by GSE
impact, (2) examine the small-scale failure modes in focused, element-level studies, (3) establish a finite element modeling methodology involving detailed simulation capability that is validated via small-scale tests, and (4) apply these modeling capabilities to accurately predict full-scale structural behavior without adjustment (e.g., tuning) of modeling input parameters.
Chapter 1. Introduction and Literature Review

1.1 Overview and Background

Over the past 40 years, high-performance fiber reinforced laminated polymer matrix composite have seen expanded use as structural components [Roeseler et al. 2007, Armstrong et al. 2005]. In particular, the aerospace industry has gradually adopted the use of glass fiber and carbon fiber composites initially in tertiary, secondary, and now quite aggressively in primary components. Composites are attractive because of their high specific strength and stiffness compared to metals, as well as customizable mechanical properties in different directions via laminate patterns and ply buildup [Armstrong et al. 2005]. These advantages led to improved aircraft performance and reduced weight. Composite structures have replaced aluminum in geometrically complex rotor blades, wings, stabilizers, and in fuselage structures. The Boeing 787 commercial aircraft now consists of 50% composites by weight. It is predicted that the use of composite material is will continue to proliferate in all types of aircraft [Roeseler et al. 2007].

While implementing polymer matrix composites in aircraft and vehicles have great advantages, there are also some significant drawbacks. First all, being a relatively new structural material, they are less well understood compared to traditional structural metals. Secondly, due to the heterogeneous nature of composite materials and the varying properties of its constituents, composite materials are inherently more complex
and difficult to characterize than homogeneous metals. So extensive testing is required
before certification and field application. Thirdly, composite damage mechanisms differ
greatly from metals. Composite damage can occur at the matrix, fiber reinforcement, or
the matrix-fiber interface. Finally, damage detection is a major issue because matrix
damage modes (particularly delamination) lack externally visibility [Schoeppner and
Abrate 2000].

Mechanically, while laminated composites benefit from having high in-plane
properties, they also have weak out-of-plane properties and impact resistance [Cantwell
1991]. Their out-of-plane stiffness, strength, and fracture toughness are typically two
orders of magnitude lower than their respective in-plane properties [Daniel and Ishai,
2005]. Thus, out-of-plane impact loading particularly presents a threat source to exposed
laminated composite parts, which can produce internal delamination, matrix cracking,
and fiber breakage [Abrate 1991]. Impacts commonly occur during assembly, normal
operations, and maintenance [Armstrong et al. 2005], from sources such as hail, bird
strike, tool drop, runway debris, and collision with ground vehicles. Low energy impacts
can generate large areas of delamination, and cause significant reductions in residual
strength [Cantwell, 1991]. Up to 50% of the load-bearing capability of the composite can
be loss due to delamination [Cantwell et al. 1983]. Whereas high energy impacts can
lead to fiber breakage and perforation, severely undermining the integrity of the aircraft.

In particular, accidental collision with ground service equipment (GSE) is
accountable for 50% of major damage to commercial aircrafts and 60% of minor damage
[Internal Air Transportation Association 2005]. Figure 1.1 shows a GSE operating close to an aircraft. The mass of these GSE can range from 2,300 kg to 3,000 kg for smaller GSE such as belt loaders, and up to 10,000 kg for large cargo loaders. Upon collision, these heavy “projectiles” can impart large kinetic energy levels to the aircraft even while moving at low velocities of 0.5-1 m/s. Ground service impacts can induce significant internal damage to the fuselage structures [Kim et al. 2014]. In addition, GSEs are typically outfitted with soft rubber bumpers that act as the contact medium during collision with aircraft, in order to blunt the impact. Blunt impactors reduce the local contact pressure and suppress the formation of externally visible signs of failure [Whisler and Kim 2012]. Thus, the damage can go undetected and delay the necessary repair works. The focus of this research is to conduct experimental and numerical investigation into the damage that can develop in commercial composite aircraft fuselage structures undergoing high energy, blunt impact events.

Figure 1.1. A luggage loader operating near an aircraft.
1.2 Scope of Research

The first objective of this research was to conduct experimental investigation of composite commercial aircraft fuselage undergoing high energy, blunt impact events. To achieve this goal, carbon fiber fuselage specimens were manufactured that were representative of current commercial airframe construction. High energy impacts events were then recreated by using fast servohydraulic actuator or quasi-static uniaxial testing machine in order to observe the damage modes. Of critical importance was the external visibility of damage in relation to the degree of internal damage. External visibility is relied upon for damage detection in current aircraft operations, which may not be a suitable method in the case of composite aircraft subjected to blunt impact sources.

The second objective of this research was to develop comprehensive analysis procedure for predicting blunt impact damage. Since solid mechanics-based close-form analysis is not practical for geometrically complex structures, the finite element method was used for majority of the analysis. Finite element models capable of predicting fuselage damage during blunt impact events were developed to gain insight to the deformation, stress buildup, and damage formation in the composite panel. The paths of failure progression and stiffness degradation were also modeled. This will serve as a valuable tool for predicting damage during blunt impacts and can be used to analytically explore various scenarios of impact locations and GSE threats (geometry, velocity).
An inverted pyramid approach to testing and simulations was taken. This approach was based on the pyramid-like "building block" approach, as shown in Figure 1.2, used in commercial aircraft development [Rouchon 1990]. Certification testing was conducted progressively from small scale coupons and elements to large scale subcomponents and components. The part's complexity is increased moving up the pyramid, providing comprehensive information from the most basic material properties to structural strength.

Figure 1.2. Building block approach to composite product development.

Figure 1.3 illustrates the concept of an inverted pyramid approach. By inverting the test pyramid, large scale subcomponent experiments were conducted first to identify
the relevant damage modes and locations. Then the testing moved down in length scale to the simpler element and coupon levels, where the each damage mode was isolated and examined. These failure modes were recreated in small-size simple element and coupon specimens, while maintaining similar boundary conditions and loading. Their deformation and failure process were recorded to aid the analysis. In the simulation stage of the investigation, finite element models were first created at the coupons and elements-scale for each specimen configuration. Since these specimens have simpler geometries, smaller size and lower part count, finite element model development was achieved more efficiently. Once the basic models were completed, the same modeling definitions were applied to more complex large-scale panel simulations to achieve comprehensive finite element models.

Figure 1.3. Road map for approaching experiments and numerical simulations of high-energy blunt impact.
1.3 Literature Review

Impact Experiments

Previous studies of composite impact properties largely focused on composite coupons and small panels. Coupon level experiments include the Charpy and Izod impact tests used for basic characterization of fracture energy [Meyers and Kumar. 1998]. Panel specimens are tested quasi-statically by indentation, and dynamically in weight drop, pendulum, and gas gun setups [Cantwell and Morton 1991]. Spherical, metallic impactors are typically used in these experiments. The quasi-static experiments can simulate low-velocity impacts, while allowing finer control and more consistent insight to the damage formation and progression [Wardle et al. 1998]. Dynamic experiments provide information on the impact damage mechanisms and dynamic responses of the composite and the impactor [Abrate. 1991].

However, composite panel impact test results do not transfer directly to composite structures, due to their complex geometry. Geometric effects from varying skin thickness and internal stiffening components change the local boundary conditions around the site of impact. Thus, the target's ability to store energy and its impact resistance are also affected [Cantwell and Morton, 1991]. Low-velocity impact study against stringer-reinforced panels showed that different energy absorption mechanism may arise depending on the impact location, panel geometry, and the dynamic response of the structure [Greenhalgh et al. 1996]. Currently, there are no publications that focused on
large-scale impact experiments on composite fuselage consisting structural elements such as skins, stringers, shear ties and C-frames. The research presented in this thesis aims to fill the gap in the existing body of knowledge.

**Effects of Impact Energy and Damage Mechanisms**

Several important impact parameters have been identified that can generally predict the damage mode and damage size for impacts against composite plates. One of these parameters is the impact energy, which affects the contact force. The degree of damage formation is a function of the impact energy [Kim et al. 2012]. Figure 1.4 illustrates this concept. Increasing level of initial impact energy always corresponds with increasing degree of damage until through-thickness penetration damage occurs. In a typical impact scenario, energy is transferred from the incoming projectile into the target. Upon collision, the target deforms, and the kinetic energy from the projectile is momentarily stored as strain energy in the composite [Lopresto and Caprino 2012]. At low energy levels, impact is essentially elastic, and no matrix or fiber failure occurs. The composite returns to its original shape after the elastic impact. However, as impact energy is increased and the composite deforms beyond its elastic limit, further impact energy is released inelastically by various laminate level failure mechanisms [Schoeppner and Abrate 2000]. The threshold load and energy levels that correspond to a laminate's elastic limit before damage initiation are known as Damage Threshold Load and Failure
Threshold Energy, respectively. The magnitudes of these damage thresholds are dependent on the laminate matrix properties [Elber 1985], and the geometric features of the impactor [Whisler and Kim, 20012] and the laminate [Cantwell 1988].

![Figure 1.4. Typical impact energy vs. damage size curve.](image)

Impact damage in fiber reinforced polymer composites always initiates in the form of matrix (or resin) cracks, since the polymer matrix material has low fracture toughness [Daniel and Ishai 2005]. Resin cracks are created between the lamina or within the lamina. They have minimal effect on the part's structural integrity, and instead act as points of stress concentrations that allow further damage to develop [Sjoblom 1988]. Subsequently, delaminations and fiber breakage occur with additional impact energy due to the pre-existing matrix cracks. These damage modes can dramatically affect the local and global stiffnesses of the composite, and are usually accompanied by
load drops and audible cracking noises [Gardiner and Pearson 1985]. Delamination is driven by the interlaminar shear and peel stresses [Elber 1983], and can reduce the flexure rigidity and influence the buckling of the laminate when subjected to in-plane compression [Olsson et al. 2005]. When the impact load is large enough to initiate delamination, the area of delamination is found to be proportional to the load [Wu and Shyu 1993]. On the other hand, fiber breakage is driven by bending stresses, and will affect the in-plane tension strength, bending rigidity, and compression strength of the laminate [Abrate 1991]. With higher impact energy, the damage size eventually stabilizes. Impacts that involve very high energy can produce penetration damage. Penetration is a localized damage at the impact site, and therefore the size of the damage zone is limited.

**Effect of Impactor Radius**

The second important parameter is the impactor radius. Most of the existing body of knowledge has focused on low-velocity impacts of small, simply-supported plates with metallic, spherical-tip impactors in a pendulum or weight drop test setup [Cantwell and Morton 1991]. The geometry of the impactor is controlled by varying the radius of its tip. It is found that sharp-tip impactors cause damage at lower energy levels, while wide-tip blunt impactors increase the energy threshold for damage initiation [Delany 2013, Whisler and Kim 2012]. This is because increasing the impactor radius results in increased contact forces during impact, as predicted by the Hertzian contact law [Abate
2001], but reduced contact areas. The result is an overall reduction in the local contact stress. Therefore, higher impact energy is required to initiate damage. In addition, the impactor radius influences the type of damage created at a given impact energy level and laminate [Whisler and Kim 2012]. Matrix cracking and delamination are associated with wider impactor radius, and fiber failure is associated with sharp impactor radius.

The contact stresses are also affected by the impactor material, as it is in the case of the rubber bumpers outfitted to GSE. These bumpers are made of a soft rubber material with a hollow interior. When compressed, rubber deforms hyperelastically to create a large contact area and reduce the local contact stress at the site of impact [Heimbs et al. 2012]. During low energy collisions, the rubber bumpers mounted to GSE help prevent the formation of local cracks and skin penetration damage resulting from otherwise high contact stresses. On the other hand, high energy GSE collision with composite fuselage bodies can produce significant damage to their internal stiffening elements [Kim et al., 2013]. In addition, due to the lack of ductility, the composite panels do not display noticeable global denting [Elber 1983]. Thus, the internal damage is effectively "masked" from visual inspections, and any internal damage may go undetected. Therefore, proper characterization of GSE blunt impact against composite fuselage structures is required in order to establish certification and operational standards.
Effect of Impact Velocity

The projectile velocity is the third important parameter in impact experiments. High velocity impact threats are encountered during aircraft operation events such as hail strike, bird strike and runway debris impact. Whereas low velocity impacts are encountered during aircraft maintenance, caused by accidental tool drops and collisions with ground service equipment. The projectile velocity is directly related to its kinetic energy at impact. Therefore, when all other variables are held constant, increasing the impact speed also results in increasing level of damage, from matrix cracks and delaminations to fiber breakage, and eventually perforation [Abrate 1991].

In addition, impact velocity determines the amount of global response in the impacted structure [Kim et al. 2012]. In high velocity impacts, the peak contact force is reached when the deformation is still small because there isn't enough time for global deformation to develop. The incident energy is dissipated over a small region, and resulting damage tend to be localized at the impact zone [Cantwell and Morton 1991]. Therefore, the boundary conditions are of less importance in fast impacts. On the other hand, low velocity impacts allow sufficient time for the deformation wave to propagate throughout the target, and target's boundary conditions and global response become significant [Cairns and Lagace 1989]. In low-velocity impact of a fixed kinetic energy, the varying boundary conditions and size of the target plate also influence the peak
contact force [Jackson and Poe 1993]. Due to the nature of low-velocity impacts, boundary conditions can also become locations of failure.

Lower speed impact scenarios, typically accepted as slower than 10 m/s, can be experimentally represented using equivalent quasi-static tests. Equivalence was shown for fracture tests [Maikuma et al. 1990, Crosley and Ripling 1998], for impact to composite plates [Lee 1991, Kwon et al 1993], and for composite shells [Meyer 1988]. Quasi-static indentation tests can provide more insight to the damage progression and interaction of damage modes than dynamic impact tests since direct observations are more easily made. This allows damage mechanisms to be compared between specimens. Vibrations that occur when a composite shell is impacted with a low velocity, high mass projectile are considered to be higher order effects and were found to have negligible influence on the panel when compared to the damage produced in quasi-static tests [Wardle and Lagace 1998].

**Modeling Impact Damage in Composites**

Comprehensive finite modeling technology already exists and is in widespread use. While the simulation results presented in this dissertation were achieved using existing simulation tool, the methodology by which these simulations tools are used to model blunt impact events are novel. Energy dissipation of the composite system must
come from both fiber failure and delamination [Bouvet et al. 2012]. Earlier lamina failure criteria were based on strength theories [Orifici et al. 2008]. They examine the stress state of the lamina and determine specific failure modes using material strength limits, or general failure using interactive criteria that combine multiple strength limits. These strength-based failure criteria are often related to each other. Various failure criteria are shown to be constrained forms of a reduced strain or stress energy density criterion [Michopoulos 2008]. The accuracy of leading failure theories was assessed in the World-Wide Failure Exercise to predict failures various laminate layups subjected to an array of biaxial loading cases [Hinton et al. 2004]. It was found that most failure criteria cannot account for features such as large deformation prediction, 3-D stress analysis, micro-mechanics, and crack density prediction. They are also unable to account for constrained ply cracking and interlaminar failures [Talreja 2014]. However, it is important to note that most of them can successfully predict the initial matrix and fiber failures within the plies [Hinton et al. 2000].

Modeling of delamination damage has been achieved using Linear Elastic Fracture Mechanics (LEFM) approaches, or Cohesive Zone Modeling (CZM) created between the laminate plies [Alfano 2009]. The fracture mechanics approaches model the extension of pre-existing cracks by evaluating the strain energy release rate at the crack tip [Krueger 2013]. Crack growth is expected when the strain energy release rate exceeds the interlaminar fracture toughness. Cohesive Zone Modeling (CZM) was more widely applicable because the interfacial material is represented with elements or surface based interactions governed by traction-separation laws [Zhang and Cross 1995]. Failure in the
interfacial layer is defined by the material strength and fracture energy [Liljedahl 2006]. Without the pre-crack requirement, CZM is more suitable for analyzing both delamination initiation and growth.

A popular approach is to model the composite laminate with volumetric elements arranged in a multi-layer fashion, where each layer of elements representing each ply in the laminate. Interface element or cohesive elements are then inserted between plies to model delamination [Qiu et al. 2014]. This approach was successful for modeling composite tubes in aircraft crash studies [Kiani et al 2013]. Finally, variations of the strength or fracture energy-based failure criteria are used to evaluate intra-laminar matrix and fiber failures. This approach can account for matrix cracking, delamination, and fiber failure [Bouvet et al. 2009], as well as accurately predict the extent and pattern of impact induced damage [Shi et al. 2014]. Computation time of modeling impact failure can be reduced by using a coarse global mesh and locally refining mesh density and applying failure criteria at locations where damage is expected to occur [Caputo et al. 2014].

In recent years, composite impact modeling methodology have matured, and thus a high degree of success was found in predicting impact induced damage in both high and low velocity impact scenarios. Of particular importance is the severe numerical instability that arises during contact, large localized deformation, and damage evolution. Special contact and element formulations within explicit dynamic finite element analysis were used to overcome these instability issues. Explicit FEA techniques are used to
predict the shock and stress wave propagation in laminated composites in ballistic impacts, as well as the formation of perforation damage [Gama and Gillespie 2011]. High velocity soft body impacts, such as bird strikes at flight speed, are also simulated with explicit FEA [Smojver and Ivancevic 2010, Nishikawa et al. 2010].
Chapter 2. Large-Scale Experiments

2.1 Overview

High-energy, wide-area blunt impacts experiments were conducted on laminated composite fuselage structures representative of the current composite, wide-body airliner constructions. These test specimens were designed to have similar materials, layup, and geometrical features as the Boeing 787 and Airbus 350 fuselages. Figure 2.1 shows a Boeing 787 fuselage prototype (Figure 2.1a), along with the test specimens designed to represent it (Figure 2.1b and 2.1c). The fuselage barrel consisted of a cylindrical skin structure with a 2.78 meter radius of curvature. Stringers and frames were attached to the fuselage interior to provide structural rigidity. Hat-shaped stringers were co-cured to the skin along the longitudinal direction and circumferential C-frames were bolted to the skin via connecting shear ties.

Two series of panel specimens were manufactured to test the fuselage response in different GSE impact scenarios. They were designated as “StringerXX” and "FrameXX" specimens. The StringerXX specimen series was designed to observe accidental TUG belt loader vehicle collisions against the fuselage that occur between C-frame supports. These panel specimens consisted of a 931 mm x 931 mm outer skin, and had stringers as the primary stiffening components, as shown in Figure 2.1c. In addition, shear ties bolted to the panels were used to mount the specimen to steel boundary conditions for testing. The TUG vehicles were typically equipped with 76.2 mm wide and 203 mm long D-shaped rubber bumpers, which were the contact medium between the TUG vehicle and
the fuselage skin during impact. An assumption was made that the maximum contact area would be limited to size of the base of the bumper. Hence, a D-shaped rubber bumper was mounted on a servo-hydraulic actuator to apply load to the specimens. The resulting impact damage found on the StringerXX specimens were largely confined within the local boundary conditions of the impact zone. The damage modes included shear tie delamination, wide-spread skin-stringer delamination, fracturing of the stringer, and skin perforation.

On the other hand, the FrameXX specimen series was designed to examine impact events caused by large heavy cargo loaders or catering vehicles that dock against the side of the aircraft. Figure 2.1b shows FrameXX specimen drawings. These cargo loading GSE are equipped with cylindrical rubber bumpers that can make contact with the fuselage during service. The bumper can over 2 meter in length and 0.20 meters in diameter. Thus, impacts involving cargo loaders and catering trucks have wider contact areas, and exert load across multiple bays, over multiple rows of shear ties and C-frames. The resulting damage was wide-spread, and can be found away the impact zone. Of particular importance for the FrameXX specimen series was the interaction between the shear tie failures and C-frame deformations during impact that can drive global failure modes. Local damage found in the FrameXX specimens included shear tie delamination and fractures within the impact zone. Global damage includes shear tie fractures outside the impact zone, skin-stringer delamination, and C-frame fractures.
Figure 2.1 Overview of specimen design. (a) Prototype of Boeing 787 fuselage on display, (b) design of FrameXX series specimens, (c) design of StringerXX series specimens
2.2 Specimen Construction

The FrameXX and StringerXX series of test specimens consisted of four main structural elements: skin, stringers, shear ties, and C-frames, as shown in Figure 2.1. Detailed drawings these components are shown in Figures 2.2 to 2.4. These geometries and dimensions reflect the construction of current carbon/epoxy composite wide-body aircraft fuselage. The outer skin structure was a cylindrical shell with a 2.79 meter radius of curvature. Hat-shaped stringers were co-cured to the skin structure, along the longitudinal direction of the cylinder at a regular spacing of 305 mm. In the hoop direction, L-shaped shear ties served as connecting elements between the skin structure and the C-frame reinforcements. Rows of shear ties were bolted to the skin structure at a regular spacing of 508 mm. Aerospace grade bolts were used to fasten the shear ties, specifically the HiLok HL19 PB8-5 countersunk bolts (6.35 mm shank diameter) and HL70-8 aluminum collars. The FrameXX specimens featured additional C-frames with 2.69 meter radii of curvature. The C-frames were bolted to the rows of shear ties at the vertical flanges of the shear ties with HL18 PB8-4 protruding head bolts (6.35 mm shank diameter). The StringerXX specimens did not feature C-frames, and their shear ties were bolted directly to steel fixtures boundary conditions for impact testing.

Manufacturing of the specimens took place at the University of California at San Diego (UCSD), and San Diego Composites. The specimen materials were pre-impregnated, carbon fiber-reinforced toughened epoxy supplied by Cytec Engineered
Materials. Specifically, they were the X840 unidirectional tape, and X840 3K and 6K woven fabrics. A monolithic laminate construction was used for all of the specimen components. The layup material, sequences, and laminate thickness are summarized in Table I. The skin structure used a layup of unidirectional tape, with an additional single layer of 3K fabric on each outer face. Similarly, the stringers had a layup of unidirectional tape with the 0° direction oriented along the stringer main axis direction. The shear ties and C-frames both used bi-directional fabrics exclusively in their constructions due to the higher formability of the fabric helped to deposit plies into the doubly curved regions of these parts. Furthermore, each layer of the C-frame laminate consisted of shorter plies spliced together in order to fit them into the doubly curved aluminum mold.

![Figure 2.2. Stringer and skin cross-sectional geometry and stringer-skin connection details (in mm). The shear ties are omitted from this figure.](image-url)
Figure 2.3. Shear tie cross-sectional geometry and connection to the skin (in mm). The view is perpendicular to that shown in Figure 2.2. The stringers are omitted from this figure.

Figure 2.4. C-frame cross-sectional geometry and connection to shear tie (in mm). The view is perpendicular to that shown in Figure 2.2.
Table 2.1. Laminate Layup Information

<table>
<thead>
<tr>
<th>Element</th>
<th>Material</th>
<th>Layup Sequence (Degrees)</th>
<th>Thickness (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Skin</td>
<td>Tape, with 3K fabric ply on each outer face</td>
<td>[0/45/90/-45]$_{2S}$ 0° dir. along stringer direction</td>
<td>2.65</td>
</tr>
<tr>
<td>Stringer</td>
<td>Tape, with 3k fabric ply on each outer face</td>
<td>[0/45/-45/90/-45/0]$_{S}$ 0° dir. along stringer main axis</td>
<td>2.37</td>
</tr>
<tr>
<td>Shear Tie</td>
<td>Fabric</td>
<td>[±45/0]$_{3S}$ 0° dir. perpendicular to skin</td>
<td>2.50</td>
</tr>
<tr>
<td>C-Frame</td>
<td>Fabric</td>
<td>Web: [±45/0]$<em>{3S}$ Flange: [-±45/0/±45/0/±45/0]$</em>{S}$ 0° dir. along frame main axis</td>
<td>Web: 2.50 Flange: 2.92</td>
</tr>
</tbody>
</table>

2.3 Stringer Specimen Tests

2.3.1 Stringer Specimen Test Matrix

Seven StringerXX specimens were tested at varying impact locations, test speeds, and impactor types. The test matrix is shown in Table II. Two different StringerXX specimens configurations were tested to determine the effect of varying impact location: panels with two stringers and panels with three stringers, all co-cured to the skin at 305 mm spacing apart. The impact tests were conducted against the exterior skins, at the center of the panels. Thus, impact load was applied between the stringers in the two-stringer specimen configuration, and on the middle stringer for the three-stringer
specimen configuration. Figures 2.5a and 2.5b show the two impact locations. Due to the difference in local boundary conditions, the two panel configurations experienced different failure modes.

![Figure 2.5. Front view of the two specimen test configurations (a) two-stringer specimen loaded between the stringers; (b) three-stringer specimen loaded on the middle stringer.](image)

In addition, the specimens were tested at two impact speeds: quasi-statically at 0.0002 m/s, and dynamically at a low velocity of 0.5 m/s. Equivalence between low velocity impacts and quasi-static indentation was established by previous studies on small square plates, with negligible effects caused by the dynamic response of the panel. Thus, the quasi-static test speed allowed for better observation of damage initiation and progression because the test could be stopped and unloaded at any point after various levels of damage were generated. In this way, multiple loading cycles were applied to the quasi-static test specimens until significant damage had been accumulated. The dynamic experiments were designed to replicate realistic impact velocity of GSE observed LAX airport [Kim et al 2012]. Unlike the quasi-static experiments, the dynamic tests were conducted in one load stroke up to a fixed actuator displacement.
This dynamic impact displacement was determined based on results from the quasi-static experiments in order to produce significant damage.

Lastly, two different impactors were used in the experiments and are shown in Figure 2.6. The first was a TUG OEM rubber bumper with a width of 76.2 mm and overall length was 198 mm. The bumper's outer radius of curvature was not constant and approximately 102 mm. The second impactor was a solid aluminum bumper of equal width and 76.2 mm of radius of curvature. The blunt impact damage generated with a rubber bumper impactor was compared with that generated with an aluminum impactor, which is typically used by composite impact research groups. The effect of varying impactor sharpness on stringer-reinforced panels was observed. The comparison drawn here was used to correlate traditional impact test damage with metal impactor to those created by the more deformable rubber bumpers.

![Figure 2.6. The 76 mm wide impactors: (a) solid aluminum with 76.2 mm radius of curvature, and (b) TUG rubber bumper with a non-constant, approximately 102 mm radius of curvature.](image)
Table 2.2. StringerXX specimen test setup summary.

<table>
<thead>
<tr>
<th>Quasi-static (0.0002 m/s)</th>
<th>Loaded on skin</th>
<th>Loaded on Stringer</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stringer01</td>
<td>Stringer00</td>
<td>Al Impactor</td>
</tr>
<tr>
<td>Stringer02</td>
<td>Stringer03 &amp; 04</td>
<td>Rubber Impactor</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Dynamic (0.5 m/s)</th>
<th>Loaded on skin</th>
<th>Loaded on Stringer</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stringer05</td>
<td>Stringer06</td>
<td></td>
</tr>
</tbody>
</table>

2.3.2 Stringer Specimen Test Setup

Fixed boundary conditions at the shear ties were used for the StringerXX specimens. This was accomplished by bolting the specimens' shear ties directly to 6.35 mm thick steel fixtures in lieu of the C-frames, as shown in Figure 2.7. The steel fixtures were designed to match the radius of curvature and bolt hole locations of the C-frames. Subsequently, the steel fixtures were bolted into the test apparatus. Blunt impact load was applied to the specimens by mounting the impactor to the test machines. The assembled test setup is shown in Figure 2.8. While quasi-static indentation experiments were conducted with a 2700 kN SATEC uniaxial test machine, the dynamic impact experiments used a 980 kN MTS servohydraulic actuator to increase the test speed. Beyond the different test apparatus, the setups for the quasi-static and dynamic experiments were otherwise the same. For the quasi-static experiments, the specimens
were bolted to the lower moving platform of the SATEC test machine, and the impactor was mounted to the upper fixed plate of the machine. For the dynamic experiments, the specimens were bolted to fixed steel blocks, and the impactor was attached to the actuator's face-flange that was suspended from a steel scaffold.

![Figure 2.7. StringerXX test setup.](image)

Data acquired during experiments included the actuator displacement, contact force, panel surface strains, and the indentation depth of the impactor. The load data was collected via 220 kN (50 kip) load cells. 6 mm strain gauges were attached to various locations of the specimens for modeling correlation purposes. Additionally, a displacement potentiometer was attached to the interior face of the panel to track the indentation depth (i.e., deformation of the skin) underneath the impactor. Due to the faster loading rate of the dynamic tests, a Phantom v.7.3 high speed camera running at 6006 fps was used to observe the interior side of the panel. Post-test damage evaluation of
the specimens was conducted visually and via non-destructive ultrasound scans using a Physical Acoustics Pocket UT portable system with 3.3 and 5 MHz transducers. The two sets of test results (quasi-static and dynamic) were compared to determine potential dynamic effects on how composite fuselage structures respond to this type of blunt loading.

![Figure 2.8. Test setup showing impactor and boundary conditions.](image)

### 2.3.3 StringerXX Specimen Results

Detailed reports of the two-stringer panels experiments (i.e. Stringer01, 02, and 05) are presented below to illustrate the general damage modes. These two-stringer panel specimens were loaded between the reinforcing stringers. Stringer01 and Stringer02 were quasi-statically indented with the aluminum and rubber bumper impactors, respectively. The effect of the impactor sharpness was determined by comparing their results. Stringer05 was dynamically impacted with the rubber bumper impactor. The
effects of loading speed were determined by comparing Stringer05 to Stringer02's results. Next, the results from the three-stringer panels (i.e., Stringer00, 03, and 06) are presented. The three-stringer panel specimens were loaded at the middle reinforcing stringer. The effect of loading locations is illustrated by comparing the two-stringer panel results to the three-stringer panel results.

**Stringer01 Experimental Results**

Stringer01 was quasi-statically indented with the 76.2 mm radius aluminum indenter on the skin between the stringers. Figure 2.9 shows the panel surface during contact with the indenter. Since the aluminum indenter was essentially rigid compared to the thin, carbon fiber skin, it resulted in the panel skin locally contouring around the indenter. The contact stresses were the highest at the edges of the indenter and was the main contributing factor to the panel's failures. The Stringer01 panel was indented over four progressively increasing loading cycles. Figure 2.10 shows the contact force vs. indentation plots for the load cycles. During the first loading, popping sounds were observed. While the damage was not detectable, it was surmised that delamination damage initiated in the shear tie corner locations. During the second loading, local skin delamination damage was detected underneath the indenter at a load of 13.34 kN and an indentation displacement of 14.73 mm. The delaminations were oval-shaped, and their locations coincided with indenter's edges, where the contact stresses were the highest. During the third loading, small clicking sounds were heard after 13.79 kN. The test was
continued until the next large failure event occurred at 22.83 kN and an indentation displacement of 20.57 mm. The damage mode was local skin delamination growth under the indenter. The specimen was then reloaded for a fourth loading, during which a loud crack was heard at 26.71 kN and 21.84 mm indentation displacement. Local skin penetration damage was generated, as shown in Figure 2.11 and 2.12.

![Contact between the aluminum indenter and Stringer01's external skin.](image)

Figure 2.9. Contact between the aluminum indenter and Stringer01's external skin.

![Stringer01 contact force vs. skin indentation plots.](image)

Figure 2.10. Stringer01 contact force vs. skin indentation plots.
Figure 2.11. Stringer01 local skin penetration.

Figure 2.12. Stringer01 skin penetration and delamination damage.
Stringer02 Experimental Results

Stringer02 was quasi-statically indented with the TUG D-shaped rubber bumper on the panel's skin between the stringers. Figure 2.13 shows the contact between Stringer02 and the rubber bumper. In this case, the contact stresses were reduced due to the hyperelastic behavior of the rubber, and thus local skin delamination was suppressed. The StringerXX panel was loaded five times. Figure 2.14 shows the force vs. indentation curves for Stringer02 for each loading cycle. Inflections in the data plots were observed at low-level loads due to the stiffening behavior of the rubber bumper as its open D-shaped cavity deformed and collapsed. When the load reached approximately 2.89 kN, the cavity closed and allowed the bumper to stiffen significantly.

Figure 2.13. Contact between the rubber bumper and Stringer02's external skin.
The first two loading cycles produced only shear tie radial delamination failures, as the result of undergoing opening deformation due to the impact load. A sample microphotography of shear tie radial delamination is shown in Figure 2.15. During the first loading, clicking noises were observed at approximately 35 kN of contact force, signaling potential delamination failures in the shear ties. The panel held a load of 44.84 kN and an indentation level of 27.7 mm when a loud cracking noise was heard. When the panel was unloaded, the shear ties showed residual bending strains and visible crushing damage in the shear tie corners. During the second loading, active clicking sounds were again observed past 44.48 kN. Large panel deformation led to opening of the shear ties, and caused radial delamination at these locations. These shear tie failures did not cause load drops, but they did contributed to stiffness losses.
Major failures occurred from the third load cycle onwards in the form of stringer-to-skin delamination. In addition, continued shear tie delamination and crushing damage also occurred throughout these load cycles. During the third loading, a load drop occurred at 57.96 kN and an indentation of 32.3 mm. This failure coincided with the two stringers delaminating at the shear tie support locations, away from the loaded zone. Figure 2.16 shows the area of skin-stringer delamination damage accumulated up to this point. This delamination grew during the fourth loading, at the loaded area and at support locations. Major load drop occurred at 61.33 kN contact force and 34.5 mm skin indentation due to the large-sized delamination growth. However, no visible damage was found on the skin’s external surface after the panel was unloaded, despite the significant stringer-skin delamination damage. Additional fiber crushing damage occurred in the shear ties. Figure 2.17 shows the cumulative crushing damage at shear tie's curved corner. Crushing damage occurred at the edge of the shear tie where they made contact.
with stringer flange which was in the primary load path. Even after the load drop at 61.33 kN, the panel still held a load of 47.06 kN.

Figure 2.16. Stringer02 after fourth loading. White boxes indicate area of delamination damage incurred during 3rd loading. Dashed red box indicates area of delamination incurred during 4th loading.

Figure 2.17. Stringer02 cumulative crushing damage on one of the shear ties after the fourth loading, the damage is circled in red.
During the fifth load cycle, asymmetrical loading of the panel was observed because the delaminated stringer softened the skin structure. The load was redirected towards the second stringer which remained intact. The panel held a load of 55.34 kN before extensive skin-stringer delamination occurred on the second stringer as well. The delamination of the second stringer extended to the free edge of the panel. It was very likely that the final delamination damage grew from the existing delaminations because of the high crack tip stress.

Significant stiffness loss was observed for each load cycles, as shown in Figure 2.14. This was due to the cumulative damage sustained at the shear ties corners, and the delamination growth of the stringer flanges during the fourth and fifth load cycles. Sudden damaging events, such as rapid delamination growth resulted in load drops. Whereas, slow damage growths resulted in gradual change in the slope of the loading curves towards the end of each load cycle. The load level at which gradual damage growth occurred was inferred by the softening seen at approximately 56 kN in the fourth loading cycle and 51 kN in the fifth loading cycle in Figure 2.14. These damaging events were accompanied by audible cues of composite failure.

**Stringer05 Experimental Results**

Stringer05 was dynamically impacted, at 0.5 m/sec, with the TUG D-shaped rubber bumper on the panel's skin between the stringers. Only one load cycle was conducted for Stringer05. The dynamically-loaded Stringer05 load vs. indentation plot is
shown in Figure 2.18, along with a combined loading plot for Stringer02. As shown in this figure, the two panels show similar stiffness response and peak loads. However, their initial failure modes and path of failure propagation were different.

![Contact Force vs. Skin Indentation Plots](image)

**Figure 2.18.** Stringer02 and Stringer05 contact force vs. skin indentation plots.

A high speed camera was used to monitor the panel's interior over the duration of the impact. The camera captured a view of the panel at the impact zone, specifically focusing on a stringer-skin connection. Figure 2.19 shows a series of high speed video still-captures of the damaging events, each event's correspond time stamp was noted. Figure 2.19a shows the stringer-skin connection at 17.2 msec after impact. At this point in time, the panel deformation was minimal and no damage has been incurred. The panel interior was partially painted white to accentuate damage. The metal rod in this figure was attached to the panel to track the skin deflection during impact. The green
arrow in Figure 2.18a points to the edge of the stringer-to-skin bond, while the red arrow points to the angled connection between the stringer flange and its diagonal wall. Both of these locations would experience failure.

![Images of high-speed video frames](image)

**Figure 2.19.** High speed video still frames of Stringer05 test as viewed from the panel's interior. (a) \( t = 17.2 \) msec, (b) \( t = 199.0 \) msec, (c) \( t = 199.6 \) msec, and (d) \( t = 202.4 \) msec after impact.

Two significant load-drops were observed at 36.1 and 38.1 mm of skin indentation, each corresponded to failures occurring at a stringer in the impact zone. The first load drop at 36.1 mm of indentation was caused by skin-to-stringer delamination and stringer fracture failures on first stringer. Although its failures were not captured within the view of the high speed camera, carbon fiber debris was seen ejecting from that
stringer location. Figure 2.19b shows the white carbon fiber debris falling from the out-of-view stringer at 199.0 msec after impact. The timing of this event coincided with the first load drop, indicating stringer failure. This first stringer delamination and stringer fracture failures were confirmed after the experiment. The second load drop, at 38.1 mm of indentation, was caused by similar failures at the second stringer in the impact zone. This time, the sequence of damaging events was captured by the high speed camera. Figure 2.19c shows that the skin-to-stringer delamination initiated underneath the impactor 199.6 msec after the impact, and then the delamination grew towards the boundary shear ties that were outside of the impact zone towards the boundary conditions. Figure 2.19d shows that immediately afterwards, stringer fracture failure occurred at 202.4 msec after impact.

Post-test examination of the panel revealed that a through-thickness skin crack was formed along the skin-stringer joint underneath the impactor. The exact timing of the skin crack damage was not known. However, it was assumed that the skin crack occurred at roughly the same timeframe as the skin-stringer delamination and stringer cracking because of high shear stress buildup. Figure 2.20 shows the panel surface with the through thickness skin crack. Despite the level of damage, there is almost no global residual deformation of the skin. The only external sign of impact damage was the formation of skin crack. Figure 2.21 shows the post-test damage from both the exterior and interior views of the panel. The skin-to-stringer delamination area is indicated by white hatch marks on the exterior view. It is important to note that all failure modes were created along the skin-stringer connections, in the primary load path. Similar to the
Stringer02 panel, permanent softening of the shear ties due to radial delamination and corner crushing was also found.

Figure 2.20. Post-test state of Stringer05 showing surface cracks along the skin-stringer connection at the impact location.

Figure 2.21. Center image: post-test A-scan map of Stringer05 (hatched areas are locations of skin-stringer Delamination); side images: crack formations along the stringer radii and on the flanges viewed from the interior side.
2.3.4 Effect of Varying Impact Location

The specimen Stringer00, 03, and 06 were all three-stringer specimens and were indented or impacted on the middle stringer. These specimens were analogous to the Stringer01, 02, and 05 specimens (loaded between stringers) in terms of their testing speed and impactor type. The stringer-reinforced areas of the skin were locally stiffened, so higher forces were generated when the load was applied over a stringer compared to loading between the stringers. This was evident from the comparing Stringer00 and Stringer01 loading plots in Figure 2.22. Both were quasi-statically indented with the aluminum indenter and exhibited the same skin penetration damage (see Figure 2.26 below), but Stringer00 showed a higher initial stiffness before local skin delamination was found.

Figure 2.22. Stringer00 and 01final loading contact force vs. skin indentation plots.
Another effect of varying impact location was the location of the damage. Figure 2.23 shows the two-stringer and three-stringer panel configurations and their primary load paths. Since the impact occurred between the shear tie boundary conditions, loading was transferred from the stringers into the shear ties. These load paths represented the locations where damage would most likely develop. Figure 2.24 shows the final delamination damage of the Stringer02 and Stringer03 panels. In the Stringer02 (loaded on the panel skin) case, the bumper made contact with the two adjacent stringers. Since the load was carried by both stringers, it was evenly distributed in the panel. Therefore, the delamination damage was more widespread. In the Stringer03 (loaded on stringer) case, the transverse load was carried primarily by the middle stringer since it was a direct load path to the shear ties boundary conditions. Thus, the delamination damage was confined to the impacted stringer.

Figure 2.23. Primary load paths of the two-stringer (left) and three-stringer (right) specimen configurations based on the location of loading.
In addition, the impact location has an important effect on damage mode. The skin and stringer joints were locations of drastic stiffness change and stress concentration. Transverse loading applied over the stringer induced high bending and transverse shear stresses at the skin-stringer joining points. This is illustrated in Figure 2.25. As such, impact load applied over the stringers always produced fiber cracks along the skin-stringer joints, as shown in Figure 2.26. In contrast, impacts occurring at an un-reinforced skin location would only result in skin-stringer joint cracks if the impact speed was increased. The degree of skin cracking was also reduced when the impact occurred at the un-reinforced skin.
2.3.5 Effect of Impactor Type

Metal Impactor

The aluminum impactor was used in both the Stringer00 and Stringer01 quasi-static indentation experiments. Both specimens showed the same damage modes and similar failure loads. Due to their significant stiffness differences, the panels' skin contoured around the aluminum impactor upon contact. Local delaminations occurred within the skin layup. According to the Hertzian Contact law, the contact stresses are inversely proportional to the contact radius. Thus, the skin delamination initiated at skin contact zone near the edges of indenter, where the contact shear stresses were the highest. The shape of the delamination was two oval-shaped patches, that later grew into a
peanut-shaped patch (see hatched zones in Figures 2.13 and 2.26). Following the delamination, increasing contact forces eventually led to local skin perforations.

**Rubber Bumper**

On the other hand, the rubber bumper produced wide-spread damage. During the rubber bumper loaded experiments, the bumper deformed considerably as it flattened before the buildup of higher contact forces. Its bumper contact area increased to approximately 8400 mm$^2$. Thus, the contact pressure and the resulting interlaminar stresses were lower relative to metal indenters. When the bumper was fully flattened and the panel had undergone significant deformation, transverse loading was applied over the stringer-skin connections (i.e., stringer flanges), allowing a direct load path to the stringers, as shown in Figure 2.27. The bumper's contact stresses could not be predicted with the Hertzian Contact Law. Although contact forces from spherical rubber indentation could be approximated with trigonometric functions [Sun and Grady 1988], the D-shaped bumper was difficult to analyze due to its hollow, irregular shape and contact with a target that had varying stiffness. Therefore, stress analysis was deferred to finite element sections.
The Stringer02 (loaded on skin) and Stringer03 (loaded on stringer) specimens were both quasi-statically indented with the rubber bumper. Unlike the rigid aluminum impactor experiments, these experiments did not produce local, intra-skin delamination. The local contact stresses were reduced due to the rubber bumper, and resulted in widespread skin-stringer delamination along the load path. The Stringer02 specimen sustained skin-stringer delamination along two stringer flanges adjacent to the bumper loaded area, as shown in Figure 2.16. This delamination damage initiated at a location away from the impact zone, at the boundary support locations. With increasing contact load, the delamination grew inward, towards the middle of the panel. Likewise, the Stringer03 specimen initially developed stringer delamination away from the impact zone at the edge of the panel. However, local skin crack damage was also found in Stringer03 due to its impact location.
2.3.7 Effect of Loading Speed

The effect of loading speed was observed by comparing the Stringer02 vs. Stringer05 results. Deformation of the rubber bumper played a significant role in the damage formation of both tests. As the hollow D-shaped bumper was compressed and flattened, the bumper made contact with the stiffened region of the panel, allowing load to transfer directly via the stringers. Thus, damage was primarily found along this load path. Since the panel skin's flexural rigidity was significantly lower compared to the stringers, the impact loading was effectively carried by the stringers. The analysis of these experiments was idealized as two dimensional, beam bending problems. This concept is illustrated in Figure 2.28, with the shear force diagram of the stringers.

As shown in the diagram, the stringers were effectively 508 mm long beams supported at the ends by the shear ties. A distributed pressure load was applied over a section at the mid span of the beam by the bumper. The shear forces produce transverse shear stresses, which developed higher values at the impact zone, and at the geometrical transitions between the stringer and shear ties. These locations were the most susceptible to delamination failure. This viewpoint helped to explain the locations of the initial skin-stringer delamination failure, and the locations of the stringer and skin cracks, as observed in the two tests. Dynamic localization of the panel response can be used to explain the differences between the two test results. In the Stringer02 (quasi-static) test, skin-stringer separation initiated at the shear tie support locations, whereas in the
Stringer05 (dynamic) test, the same damage initiated locally under the impactor. The Stringer05 panel also sustained additional stringer and skin cracks at the impact zone. These damage modes indicate that dynamic localization can influence the panel deformation shape and stresses, even at a low impact speed of 0.5 m/sec due to the large specimen size. Figure 2.29 illustrates the time-dependent deformation response of a beam undergoing dynamic three-point bending at various times after the impact.

Figure 2.28. Side view of the panel. The impact experiments can be idealized into two dimensional beam bending of the stringers, the shear force diagram of stringer is also shown.
The first three diagrams in Figure 2.29 show the initial deformation responses of a beam immediately after a dynamic impact. These deformation modes are influenced by inertial effects and higher order mode shapes, for which the central portion of the panel is mainly reacting against the dynamic load. The boundary reactions do not balance with the loading, and localization of the loading does not allow equal shear force distribution across the beam length. Thus, higher stresses existed at the center of the beam. As the time after initial contact increases or as the loading speed decreases, deformation of the beam transitions into a quasi-static response, similar to Stringer02, as illustrated by the last sketch in Figure 2.29. In this case, the applied load is balanced by the reaction forces at the outer boundaries. Since transverse shear stresses at the indented zone were
reduced by the soft rubber contact, skin-stringer delamination did not initiate at the impact size in Stringer02. Instead, the damage started at the locations of the outer boundaries where contact with the stiff shear tie would induce higher shear stresses. Strain gauge data also showed supporting evidence of the dynamic localization effect. Strains from both the impact zone (skin bending, in-plane tension) and the boundary area (shear tie compression) were compared between the two specimens in Figure 2.30. The overall strain pattern was similar for the two specimens. However, they showed deviation at between 50 and 90 mm of actuator displacement, after the bumper had collapsed. The dynamic specimen experienced a higher tensile strain at the impact zone than the quasi-static specimen. Meanwhile, its compressive strain at the boundary area was lower compared to the quasi-static specimens. The observed dynamic strain deviations corresponded with the dynamic response diagram shown in Figure 2.29.

Figure 2.30. Strain vs. actuator displacement data taken at the impact zone (skin bending) and at the boundary area (shear tie compression) for both the quasi-static (Stringer02) and dynamic (Stringer05) experiments.
2.3.8 StringerXX Panel Discussion

Regardless of the test configuration, the damage always initiated at the shear ties due to the downward compression and moment loading on the shear ties, as shown in Figure 2.31. Opening moment was induced by both skin rotations at the shear tie supports, as well as lateral movement of the panel, resulting in interlaminar tension stress at the shear tie corners. The composite material's interlaminar fracture toughness in tension is low, in the order of 700 J/m². Thus, shear tie delamination was always the first failure mode observed. As the compression load was increased, fiber breakage at the shear ties was also found due to crushing from the impact load. The shear ties failures were considered to be minor failure modes due to their progressive nature and minimal effect on the structural integrity of the panel.

On the other hand, major failure modes had significant effect on the structural integrity of the panel. They were strongly dependent upon the impactor type, impact location, and impact speed. These major failure modes included skin penetration, stringer delamination from the outer skin structure, skin cracking, and stringer fracturing. Skin penetration damage occurred in impact tests involving the metal impactor due to the localized contact stresses around the periphery of the impactor. Stringer-skin delamination occurred in cases where the rubber impactor was used. The rubber impactor reduced the local stresses at the impact site and suppressed penetration damage at the point of impact. As higher loads were built up, the interlaminar shear stresses
between the stringer and the skin eventually led to the development of delamination damage. In addition, skin cracks and stringer fracture were also observed in the case of rubber bumper impact at the local boundary conditions created by the stringer stiffener, depending on the geometry of the impacted panel.

![Diagram of StringerXX panel](image)

Figure 2.31. Side view of StringerXX panel: (a) before impact and (b) during impact. Both downward compression and moment loading were exerted on the shear ties.

### 2.3.9 StringerXX Experiment Conclusions

In conclusion, blunt impacts induced damage on stringer-reinforce composite panels could be externally visible (skin penetration and cracking) or masked (skin-stringer delamination). They could also be localized or wide-spread. These damage
characteristics are strongly dependent on impactor sharpness, impact location, and impact velocity. Figure 2.32 summarizes the effects of these three factors on damage visibility. First of all, the impactor stiffness and impact locations affect the panel deformation by changing the local contact stiffness between the impactor and the panel specimens. Higher contact stiffness generated higher contact stresses, which created localized skin cracking with increased external visibility at the point of impact. Conversely, lower impact stiffness reduced the contact stresses, and generated wide-spread internal damage at points of stiffness changes. This internal damage was obscured from the external surface. Secondly, the impact speed contributed to stress and damage localization by changing the deformation mode shape. Higher speed impacts resulted in higher order deformation mode shape confined around the impact zone, whereas quasi-static loadings resulted in first order mode shapes. It was found that unlike the previous study on composite square plates, the dynamic response of the fuselage panels do have an important effect on the specimen's responses, even for low velocity experiments (0.5 m/s). This was due to the larger length-scale of the StringerXX panels changing the dynamic reaction time compared to the lab-scale plates.
Figure 2.32. General trend of damage based on the StringerXX series experiments.
2.4 FrameXX Specimen Tests

The FrameXX series specimens were larger compared to the StringerXX specimens, and they featured additional C-frame stiffening elements (see Figure 2.1). This series was designed to study the fuselage response during collision with large GSE, where the contact area is much larger. Transverse impact loading was applied to the panels via long, cylindrical rubber bumpers across two or more C-frames. The FrameXX experiments were first conducted quasi-statically and then dynamically. The quasi-static test specimens, Frame01 and 02, each had three C-frames, while the dynamic test specimens, Frame03 and 04, were wider and each had five C-frames.

2.4.1 FrameXX Quasi-static Experiments

All four FrameXX specimens used similar experimental setups. The Frame01 test specimen with its boundary conditions is shown in Figure 2.33. The FrameXX specimens were simply supported at the ends of all of their C-frames with added rotational spring stiffness to resist bending of the panel. Steel plates with bending stiffness of 1,130 kN-m/rad were added to the C-frame boundary conditions to simulate the resistance of a full barrel fuselage structure [DeFrancisci, 2013]. The steel plate's bending rigidity was determined using finite element models, by matching the elastic response of the FrameXX panel model to that of a full-barrel fuselage model. The panel skin and stringers were unconstrained at the edges. The full experimental setup is shown
in Figure 2.34. Transverse loading was applied at the exterior skin using servohydraulic actuators. The FrameXX specimens were loaded using a single degree of freedom actuator. An aluminum box beam with a cross-section dimension of 152.4 mm by 152.4 mm was mounted to the actuator to form a flat surface to support the rubber bumper and distribute the impact load. The rubber cylindrical bumper purchased from the original equipment manufacturer SAGE Parts was then attached to the box beam. The bumper had an outer diameter of 178 mm, an inner diameter of 127 mm. The length of the bumper varied between the test specimens.

Figure 2.33. Isometric view of the Frame01 Panel test setup. Steel boundary conditions were attached to the ends of the C-frames.
Figure 2.34. Frame01 panel test setup with the cylindrical bumper indenter.

Frame01 Specimen Test Results

The results of the quasi-statically indented panel Frame01, is summarized in this thesis. A detailed report of both quasi static experiments can be found in Dr. Gabriela DeFrancisci's PhD thesis [DeFrancisci, 2013]. Figure 2.35 shows the panel's layout and its loading zone. The panel was 1,066 by 2,439 mm in dimension, and had three reinforcing C-frames and four stringers. It was loaded externally on the skin between two stringers. The length of the bumper indenter was 572 mm, and applied transverse load over the skin across two shear ties. The shear ties in turn transferred the load into
the two reinforcing C-frames, as shown in Figure 2.35. A side view of the panel during the load up is show in Figure 2.36a. Once fully compressed, the bumper flattened between the box beam and the panel surface. The testing process consisted of four multiple loading cycles that continued until a new damage major mode was observed.

Figure 2.35. Top view of Frame01 panel configuration, the spacing between C-frames is 508 mm.
Figure 2.36. (a) Side view the Frame01 with compression loading applied directly to the shear ties, and (b) fracture of the shear tie (F01H) at the curved corner after the second loading.

Figure 2.37 shows the Frame01 contact force vs. skin indentation plots. With each successive load cycle, the panel lost overall stiffness due to the accumulation of damage. During the first loading, audible clicking noises were observed at a load of 28.91 kN, followed by a load drop. It was determined that loaded shear ties F01H (see location in Figure 2.33) experienced delamination at its curved corner near where the shear tie contacted the skin. During the second loading, the same shear tie fractured at the curved corner at a load of 57.5 kN, as shown in Figure 2.36b. These initial shear tie failures occurred at a load of 57.53 kN due to the direct, compressive load from the bumper. The C-frames connected to the shear ties were significantly stiffer compared to the shear tie, thus, the early damage accumulated at the shear tie.
Figure 2.37. Contact force vs. skin indentation of Frame01. The skin indentation was measured in the middle of the loading zone.

During the third loading, the fractured shear tie allowed for contacts between the C-frame and the stringers. Also, local twisting of the C-frame was observed because it was unrestrained by the shear tie. At 53.3 mm of indentation, fracture of shear tie F01C occurred (see location in Figure 2.33). The final failure modes of the panel were contact failures between the C-frames and the stringers in the impact zone. At 62.23 mm of skin indentation and 44.5 kN, severing of a stringer was observed at a point of contact between a C-frame and a stringer. This was accompanied by the delamination of the top stringer flange from the skin. Figure 2.38 shows the damage from the interior of the specimen. During the fourth loading, through-thickness cracking in Frame #2 was observed where the stringer contacted the C-frame. The test was stopped at a load of
47.36 kN. Even after accumulating widespread, internal damage, there was no clear, externally visible sign of damage on the panel except a few skin cracks by the bolts of the center shear ties.

Figure 2.38. Post third loading major damage: interior view of Frame01.

2.4.2 FrameXX Dynamic Experiments

The Frame03 and Frame04 panel were dynamically impacted. The two specimen layout were identical, but were tested under different conditions. Figure 2.39 show the layout of these specimens. These panels were 1829 mm by 2438 mm in dimension. Each panel had four stringers arranged at 305 mm apart, and five C-frames arranged at 457 mm
apart. The impact zone spanned across the three middle C-frames (see Figure 2.39). The impact location was between stringers as it represented a more critical impact scenario where visually undetectable damage could be generated.

Figure 2.39. Top view of Frame03 and 04 panel configuration, the spacing between C-frames is 457 mm.

Figure 2.40 shows the FrameXX dynamic experimental setup, with a zoom out drawing of the setup to the side of the figure. The specimen rested on top of two concrete blocks to allow for access underneath the panel. The C-frame's simply supported and rotational boundary conditions from the quasi-static FrameXX experiments were reused in this test setup. Dynamic loading was applied to the specimen by a servohydraulic
actuator to which a 1.0 m long cylindrical bumper was mounted. The actuator was suspended from a scaffold made out of steel I-beams.

![Figure 2.40. Experimental setup for Frame03 and 04 dynamic experiments.](image)

**Frame03 Experimental Results**

The Frame03 specimen was dynamically loaded two times with displacement controlled actuator strokes. Figure 2.41 shows the actuator displacement history plots for this specimen. Each loading was applied at a constant velocity of 0.5 m/sec, followed by a 0.5 sec pause before unloading. These displacement values were decided based on the quasi-static experimental results to generate similar shear tie and C-frame damage modes.
Figure 2.42 shows the contact force vs. skin indentation plots for both load cycles. The first loading had a total actuator displacement of 159 mm, which included closing the initial gap of 6.4 mm between the bumper and panel surface. The cylindrical bumper had a hollow inner diameter of 127 mm, so the resulting displacement of the specimen skin was on the order of only 20 mm. The initial failure mode was delamination at the curved corners of the shear ties directly under the impactor. Also, moderate crushing damage of these shear tie corners was found, as shown in Figure 2.43. Note that the interior of the carbon fiber panel was painted white to visually accentuate damage. This damage occurred in all three shear ties in the impact zone shown in Figure 2.39. They were created due to high localized bending and shear stresses induced by direct compression. The failures were indicated as event D1 in Figure 2.42. Based on ultrasonic inspection form the panel exterior, no delamination between the skin and stringers occurred.
The maximum actuator displacement of the 2nd Loading was 222 mm. As shown in Figure 2.42, the initial portion of the re-loading curve was less steep compared to the
1st loading. Also, the second loading curve does intercept the first loading curved where the first test was stopped. These implied that the stiffness of the panel was reduced due to the pre-existing damage in the shear ties.

Figure 2.44 shows a series of time lapse still images (taken with the high-speed camera) of a C-frame during the second impact loading. In this figure, the C-frame was viewed from the specimen's interior. The middle shear tie in these figures was in the impact zone. Figure 2.44a shows further crushing damage at the shear tie corners that corresponded to events D1 in Figure 2.42. Upon fracturing of the three impact-loaded shear ties, the load dropped significantly at a skin displacement of 35 mm due to the loss of load bearing capability in the broken shear ties. Following this failure, the gaps between the stringers and the C-frames closed as they came into direct contact (Figure 2.44b). Therefore, the impact load increased between 40 and 60 mm. Similar to Frame01 failures, the severed shear ties allowed for C-frame twisting and contact with the stringers, which led to wide-spread shear tie and C-frame failures outside of the impact zone. High contact forces existed during this stringer-frame contact, which was confirmed by the scraping marks left on the stringers by the C-frames. However, unlike in the quasi-static experiments, the C-frame and stringer contacts did not prevent further C-frame twisting. As the impactor displacement increased, the C-frames twisted, putting moment load on the shear ties adjacent to the impact zone. Failure of these outer shear ties occurred next, as shown in Figure 2.44c, and corresponded to the events indicated as D2 in Figure 2.42. The resulting major load drop at 63 mm occurred due to the frames being able to freely twist. Final failure occurred at 83 mm skin displacement in the C-
frames. The C-frames fractured near the boundary fixture under a combination of torsion, bending, and shear load as shown in Figure 2.44d. Note that the series of images in Figure 2.44 showed increasingly widespread damage as the actuator displacement and the degree of C-frame twisting increased. While the damage initiated with shear tie at impact zone, C-frame rotation have allowed shear tie fractures outside the impact zone, and eventually C-frame fracture at the boundary conditions of the specimen.

The final internal damage state is shown in Figures 2.45 and 2.46. All three frames fractured during event D3 in Figure 2.42 and nine shear ties in the impact zone and adjacent to the impact zone fractured as depicted in these figures. A view of the outside surface of the specimen after impact is shown in Figure 2.47. The thin skin design was very flexible when impacted against a non-stringer reinforced location. As a result, the skin rebounded after the experiment. No external skin cracking damage was visible, and the permanent residual skin denting was measured to be 4.5 mm 24 hours after the test. The skin deformation was measured by a photogrammetric technique in which the panel surface was digitally constructed based on photographs of the panel. The visual detectability of a 4.5 mm dent existing over a ‘dent span’ of approximately 1 m was considered to be difficult to perceive by visual observation.
Figure 2.44. High speed video stills of Frame03 2nd Loading: (a) fracture of the impacted shear tie, (b) contact between the C-frame and stringers, (c) fracture of the shear ties adjacent to the impact zone, (d) off-screen fracture of the C-frame at the boundary locations.
Figure 2.45. Frame and shear tie damage of Frame03 after 2nd Loading.

Figure 2.46. Frame03 C-frame fracture failures near boundary supports.
Frame04 Experiments

Frame04-1

Two other loading scenarios were explored using the Frame04 specimen. Under the assumption that Frame03’s second loading data were compromised by the pre-existing shear tie damage from the first loading, another dynamic experiment was conducted in an attempt to fill in the data gap. Specifically, the load at which the shear tie fracture failure occurred was questionable due to the pre-existing delamination (see Figure 2.43) allowing higher stress concentrations to develop at the shear tie corner. Thus, the Frame04-1 panel specimen was designed to be identical to Frame03, but was impacted
for a maximum actuator displacement of 180.3 mm to determine the panel response at this displacement level without the initial shear tie delamination failure that softened the panel. The Frame04-1 contact force vs. skin indentation result is plotted along with the Frame03 results in Figure 2.48 for comparison. The Frame04-1 panel generated a similar level of shear tie crushing failures as Frame03 panel (see Figure 2.44a) at 180.3 mm displacement stroke. However, the maximum load was increased to 90.56 kN due to the continuous actuator stroke.

![Figure 2.48. Frame04-1 and Frame03 contact force vs. skin indentation plots.](image-url)
**Frame04-2**

With the early shear tie delamination and fracture failures observed in the Frame03 and Frame04-1 experiments, an alternative testing scenario was considered in the Frame04-2 experiment. Frame04-2 was identical to Frame03 in the specimen layout and loading, however the carbon fiber shear ties in the impact zone were replaced with more robust, shear ties made out of aluminum 7075. These shear ties were 3.2 mm thick and 254 mm wide at the flange. Figure 2.49a shows the shear ties as they were mounted to Frame04 panel at the impact zone and adjacent to the impact zone. Figure 2.49b shows the overall dimensions of these shear ties. The stiffened shear ties did not show brittle failure as the carbon fiber composite shear ties did. In addition, the lack of a laminated construction allowed the aluminum shear ties to carry much higher load before failure.

![Figure 2.49. The aluminum 7075 shear ties (a) mounted to the Frame04 panel, and (b) overall dimensions.](image)
However, due to their higher strength, the aluminum shear ties did not allow for an energy releasing mechanism. As a result, the reinforcing C-frames experienced high local strains at the bolt connections between aluminum shear ties and the C-frames. This eventually led to the local shear fracture failures of all three of the impacted C-frame, as shown in Figure 2.50. Figure 2.51 shows the contact force vs. skin indentation plot of the Frame04-2 specimen test along with the combined plot for the Frame03 and Frame04-1 tests. According to the figure, a much higher peak contact force of 125.4 kN was achieved before the initiation of failure. After that, three load drops were observed that corresponded to the fracture of each C-frame. Although all of the failures were local, this specimen configuration with stronger shear ties is undesirable due to the lack of intermediate failures before major C-frame failures.

Figure 2.50. Frame04-2 local shear failure of the C-frames under the impactor.
2.4.3 FrameXX Specimen Conclusions

When the FrameXX specimens loaded between the stringers, the impact loading was transferred directly into the shear ties and the C-frames. In response, the shear ties in the impact zone delaminated at the corner and subsequently fractured in buckling mode. Fractured shear ties could no longer resist the load, and the load path was redirected to the outer shear ties. With increasing contact forces, the C-frames twisted due to load eccentricity away from their shear centers. Then the C-frames made contact with the stringers. In the case of the quasi-statically indented Frame01 specimen, a contact lock was formed between the stringers and C-frames. Thus, high contact stresses led to failures such as local crack formation and penetration in the affected C-frames and the
stringers. A drastically different damage mechanism was found in the dynamic experiment of Frame03 where the C-frames experienced large twisting deformation after the shear ties fractured. Rather than being restrained by contact locking with the stringers, the C-frames twisted continuously and scraped against the stringer's surface until failure has occurred. This led to fracture of the impacted C-frames in combined torsion, shear, and bending mode, rather than local contact stresses, as in the quasi-static case. The locations of the C-frame fractures were all adjacent to the C-frame's boundary conditions.

Whereas stringer delamination and stringer cracking were the major damage modes for the StringerXX series of experiments, the FrameXX specimen failures were more complex due to their larger size and the addition of C-frames. The wider impact zone allowed for direct load transfer from the bumper to the shear ties and C-frames (see Figures 2.36). Since the stringers were no longer the primary load path, skin-stringer delamination was not a major failure for these specimens. Fiber cracking failure was found in the other structural elements including the shear ties, C-frames, and stringers.

Chapter 2, in part, contains materials appearing in the following papers: Ground Vehicle Blunt Impact Damage Formation to Composite Aircraft Structures 2012, Kim, Hyonny; DeFrancisci, Gabriela; and Chen, Zhi Ming, Springer Netherlands 2012. Estimation of Ground Service Equipment Contact Damage in Composite Aircraft Panels 2012, Kim, Hyonny; DeFrancisci, Gabriela; and Chen, Zhi Ming, AIAA 2012. The dissertation author was a co-investigator and co-author of these papers.
Chapter 3. Small-Scale Experiments and Modeling

3.1 Overview

In the previous chapter, the large-scale ground service vehicle impact experiments against two series of mockup composite fuselage panels were summarized. Also, the major damage modes were identified. This chapter presents the development of a physically-accurate analysis methodology capable of predicting the failure processes observed in the impact experiments. However, directly analyzing an entire fuselage panel as a whole was an overwhelming task. Failures in multiple structural elements occurred in sequences before the final failure. Each failure mode altered the local stiffness of the panel and re-directed the load path. Therefore, the analysis of the fuselage structure was broken down into smaller components (i.e., the structural elements), and the individual failure modes were examined. In order to isolate and understand the failure modes, several series of small-scale coupon and element-level studies were conducted. Figure 3.1 shows the fuselage failure modes of interest and the small-scale experiments performed to analyze those failures. The experiments for end notched flexure coupons, curved beam opening coupons, shear tie compression coupons, and stringer elements compression were conducted. These experiments investigated the damage initiation mechanisms of the individual structural elements from the large fuselage panels. To maintain consistency, the coupons were made from the CFRP materials used in the fuselage panel construction. Also, the geometries and dimensions of the structural elements were preserved in the coupon specimens. Finally, coupon test
fixtures were design to create similar loading and boundary conditions as the fuselage panel's structural elements during GSE impact. Therefore, the small-scale experimental results were representative of the failure modes observed in the large fuselage panels.

These coupon-level experiments provide the basis for developing and refining finite element model definitions. Due to their simple geometries, loading conditions, and isolated failure modes, they allowed for efficient stress and failure analysis development. Modeling techniques available in the Abaqus commercial code were explored to

Figure 3.1 Analysis of the fuselage panel failure modes.
determine an analysis methodology that can accurately simulate the physics of the impact event. Physically-accurate properties best representing the CFRP composite materials were consistently applied across all of the finite element models. These material properties were not adjusted to attain better correlation with the experiments. The end results were coupon-level models that could capture the deformation, damage initiation, and damage propagation. Once the coupon models were validated, the modeling definitions were transferred to the large-scale StringerXX and FrameXX fuselage panel impact experiments.

3.2 Finite Element Modeling Approach

3.2.1 Material Properties

The composite material used to manufacture the impact test specimens were aerospace-grade carbon fiber reinforced toughen epoxy. Specifically, they were the Cytec X840 Z60 Plain Wave 6K fabric and X840 Z60 12K tape. The linear elastic material properties for the Cytec fabric and tape are listed below in Table 3.1. The damage-related properties such as the materials' strengths and in-plane fracture energies, listed in Tables 3.2 and 3.3, respectively. In these tables, the 1 and 2 coordinate directions are the in-plane directions of the lamina, and 3 is out-of-plane direction. Basic material characterization tests were not conducted in this thesis work because the majority of the linear-elastic, strength, and fracture energy properties were provided by the material manufacturer. Some of the out-of-plane properties necessary for finite
element simulations were not available, therefore supplemental data were taken from a similar carbon fiber toughen epoxy composite, the AGP370-5H/3501-6S, as published by Daniel and Ishai 2005.

**Table 3.1. X840 Z60 carbon/epoxy lamina elastic properties.**

<table>
<thead>
<tr>
<th>Material</th>
<th>$\rho$ (g/cm$^3$)</th>
<th>$E_1$ (GPa)</th>
<th>$E_2$ (GPa)</th>
<th>$v_{12}$</th>
<th>$G_{12}$ (GPa)</th>
<th>$G_{13}$ (GPa)</th>
<th>$G_{23}$ (GPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tape</td>
<td>1.63</td>
<td>168</td>
<td>10.3</td>
<td>0.27</td>
<td>6.89</td>
<td>6.89</td>
<td>3.72</td>
</tr>
<tr>
<td>Fabric</td>
<td>1.61</td>
<td>80.0</td>
<td>80.0</td>
<td>0.06</td>
<td>6.48</td>
<td>5.10</td>
<td>4.07</td>
</tr>
</tbody>
</table>

**Table 3.2. X840 Z60 carbon/epoxy lamina in-plane material failure strengths in tension and compression.**

<table>
<thead>
<tr>
<th>Material</th>
<th>$F_{1t}$ (MPa)</th>
<th>$F_{1c}$ (MPa)</th>
<th>$F_{2t}$ (MPa)</th>
<th>$F_{2c}$ (MPa)</th>
<th>$F_{12}$ (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tape</td>
<td>2799</td>
<td>1620</td>
<td>57.2</td>
<td>227</td>
<td>75.8</td>
</tr>
<tr>
<td>Fabric</td>
<td>993</td>
<td>772</td>
<td>855</td>
<td>896</td>
<td>71.0</td>
</tr>
</tbody>
</table>

**Table 3.3. X840 Z60 PW carbon/epoxy lamina in-plane fracture energies (amount of energy required to create a new crack in the material) in tension and compression.**

<table>
<thead>
<tr>
<th>Material</th>
<th>$G_{1t}$ (kJ/m$^2$)</th>
<th>$G_{1c}$ (kJ/m$^2$)</th>
<th>$G_{2t}$ (kJ/m$^2$)</th>
<th>$G_{2c}$ (kJ/m$^2$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tape</td>
<td>91.6</td>
<td>79.9</td>
<td>0.2</td>
<td>0.2</td>
</tr>
<tr>
<td>Fabric</td>
<td>45.8</td>
<td>39.9</td>
<td>45.8</td>
<td>39.9</td>
</tr>
</tbody>
</table>
3.2.2 Element Type

Two continuum elements in the Abaqus element library were used in modeling the composite coupon test specimens: 8-node solid element (C3D8R), and the 8-node continuum shell element (CS8R). Both elements were 3 dimensional and used reduced integration schemes with Lagrangian material description. The solid element was based on 3D Cauchy stresses. It is a general purpose element. In addition, enhanced hourglass and distortion controls were used with to prevent spurious deformation modes and excessive element distortion. On the other hand, the Abaqus continuum shell element is a 3D element with shell formulation. The shell element's transverse shear stresses were estimated to match the shear strain energy in the element. It was found that while the continuum shell element was more efficient and performed better in bending, the solid element was more robust in preventing excessive element distortion caused by compressive and shear loads.

3.2.3 Modeling Delamination Failure

Cohesive zone modeling was utilized to model the inter-ply delamination in the composite specimens. This method was used extensively to simulate crack initiation and growth. The interface materials were modeled with either cohesive elements or cohesive surface interactions between the lamina. The cohesive element zone's linear elastic and fracture behaviors were governed by a bilinear traction-separation interaction applied to
matching nodes of the bonded surface pair. Mode I (tension) and Mode II (shear) nodal separations were tracked separately. Figure 3.2 shows the bilinear constitutive law for cohesive zone modeling.

![Bilinear Constitutive Law](image)

**Figure 3.2. Traction separation response of cohesive zone modeling.**

An artificial penalty stiffness, $K$, was assigned to the surface interaction to define its elastic response. The penalty stiffness should be significantly higher than the transverse stiffness of the sublaminate to avoid affecting the compliance of the composite [Turon et al. 2007]. $K$ is defined as:

$$K = \alpha \frac{E_{33}}{t}$$  \hspace{1cm} (3.1)

where $E_{33}$ is the transverse Young's modulus of the sublaminate, $t$ is the thickness of the lamina, and alpha is a parameter suggested to be greater than 50. Damage was initialized when the max traction stress is reached. Stiffness matrix of the cohesive zone was degraded with a damage multiplier $D$, until the traction carried by the cohesive zone was
reduced to zero. The traction followed a linear damage evolution curve until final failure, as shown in Figure 3.2. The area under the traction-separation curve is the material's interlaminar fracture energy, $G$, under the particular mode of loading. This was used to govern the material's degradation curve. When multiple modes of crack tip loading were present, effective max traction and critical fracture energies were used to evaluate failure initial and level of stiffness degradation. The cohesive material properties are listed in Table 3.4. These properties were taken directly from the manufacturer's material datasheets and supplemented with data for the AGP370-5H/3501-6S material.

**Table 3.4. Cohesive surface interaction failure parameters.**

<table>
<thead>
<tr>
<th>Material</th>
<th>$K$ (GPa/mm)</th>
<th>$T_n$ (MPa)</th>
<th>$T_s$ (MPa)</th>
<th>$T_t$ (MPa)</th>
<th>$G_{nc}$ (J/m²)</th>
<th>$G_{sc}$ (J/m²)</th>
<th>$G_{tc}$ (J/m²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tape</td>
<td>170</td>
<td>45.9</td>
<td>77.2</td>
<td>77.2</td>
<td>403</td>
<td>1629</td>
<td>1629</td>
</tr>
<tr>
<td>Fabric</td>
<td>170</td>
<td>45.9</td>
<td>77.9</td>
<td>77.9</td>
<td>771</td>
<td>3152</td>
<td>3152</td>
</tr>
</tbody>
</table>

The cohesive surface interaction was particularly useful in this study because it reduced the interface material to simple spring interactions, without the need to define its mass and volume. Without the additional cohesive elements, the computation time was greatly reduced. Also, unlike the computational fracture mechanics approach, an existing pre-crack was not required for cohesive zone modeling. Thus, it was more widely applicable for simulating delamination failures in originally pristine specimens.
However, the size and location of the cohesive zone still must be known before the simulation.

3.3 End Notched Flexure Coupons Experiments and Modeling

3.3.1 End Notched Flexure Experiments

The first set of small-scale coupons was End Notched Flexure (ENF) specimens that were used to examine the interlaminar shear fracture behavior of the X840 Z60 6K fabric carbon/epoxy. Transverse loading on a laminated plate or beam structure induces interlaminar shear stresses between the plies. Interlaminar shear stresses buildup in the interfacial layer cause delamination when the interfacial material shear strength is exceeded. These shear stresses are higher for joint structures with geometrical and stiffness discontinuities, such as the joint between the stringers and the skin of composite fuselage. During the StringerXX series impact experiments, the interlaminar shear stresses from the transverse impact loading were the direct cause of the stringer-skin delamination failure. Hence, it is important to be able to accurately simulate the delamination failure at the coupon level before attempting to model the delamination at the panel level.

The ENF specimens were traditionally used to determine a material's Mode II (shear) fracture toughness, or the strain energy input required to generate new crack
surface. In this study, they served as a benchmark for developing the finite element modeling capability. Figure 3.3 shows the ENF specimen dimensions used in this study. The specimen was a 120 mm long by 25.4 mm wide laminated beam, with a 40 mm long initial delamination or pre-crack. The specimen's layup was identical to the shear tie's layup, with fabric plies stacked in a \([\pm45/0]_{3S}\) sequence, therefore the ENF specimens were also 2.50 mm thick. During manufacturing, a 0.025 mm thick Teflon release film was inserted into the middle of the layup to create the pre-crack.

![Figure 3.3. Dimensions of the ENF specimen. The pre-crack is represented by the red line.](image)

The ENF coupon specimens were loaded in a three-point bending flexural test setup as shown in Figure 3.4. Aluminum rollers with 3.175 mm diameter were used to load the specimens as shown in Figure 3.4. The bottom support rollers were set to 60 mm apart, with the top roller located center between the bottom rollers. The pre-crack length placed between the bottom rollers was 15 mm. A MTS uniaxial testing machine was used to apply a transverse load to the specimens. The ENF tests were conducted quasi-statically with a displacement-controlled actuator stroke at 2 mm/min until crack
length extension occurred. The sides of the ENF specimens were painted white to make crack propagation more visible. Upon loading, a pure Mode II deformation would be generated at the crack tip of the ENF coupon to drive crack extension. Damage growth at the crack tip was often described with the Mode II interlaminar fracture toughness:

$$G_{IIc} = \frac{9a^2p_C\delta_C}{2B(2L^3 + 3a^3)} \quad (3.1)$$

where $P_C$ is the critical force applied to the coupon normalize by its width at crack extension, and $\delta_C$ the critical deflection at the top roller location at crack extension.

Figure 3.4. Setup for ENF specimens. The span of specimen was 60 mm and the starter crack was 15 mm.

Four ENF coupons were tested. The force vs. cross-head displacement plots for these tested coupons are shown in Figure 3.5. All of the coupons were consistent and behaved linear-elastically until stable crack growth occurred at 4.13 mm of displacement.
The recorded initial crack length growth were approximately 1 mm in length. Crack extension reduced the overall bending stiffness of the coupons and caused minor load drops in the force vs. displacement plots. After the initial failure, stable and continuous crack growth occurred with additional cross-head displacement.

![Figure 3.5. Force vs. cross head displacement plot of the ENF coupons (a = 15 mm, 2L = 60 mm).](image)

### 3.3.2 End Notched Flexure Simulation

A model of the ENF specimens were created to access the feasibility of using cohesive surface interactions to model delamination, as well as the accuracy of the material strength and fracture energy parameters from Tables 3.2 and 3.3. The ENF
coupon model geometry was simplified compared to the experimental coupons. Figure 3.6 shows a 2D view of the coupon model with its boundary conditions. The length of the model was reduced to 60 mm to remove the overhanging length past the bottom support rollers shown in Figure 3.4 in order to reduce computational time. Beyond that, the specimen width, thickness and initial crack length were consistent with the experimental coupons. The coupon model was separated into two layers of continuum shells at the plane containing the starter crack, as shown in Figure 3.6. The half of the laminate layup definition was assigned to each shell layer via the composite layup manager. Simulation of the crack grow and propagation process was achieved by defining cohesive surface interactions between the contacting surfaces of the two continuum shells to simulate the path of crack growth in front of the crack tip (red line in Figure 3.6). In order to reduce the complexity of the coupon model, an assumption was made for the crack to remain in the same plane during crack propagation, instead of bridging into the fibers and other interfacial layers.

![Figure 3.6. 2D wireframe view of the ENF model with its boundary conditions. The red line represents the surface where the cohesive surface interaction was defined.](image)

To simulate the three-point bending boundary conditions, simply supported constraints were applied to the ends of the coupon, and a downward displacement was
applied at mid length of the model, as shown in Figure 3.6. The rollers were not modeled. Continuum shell elements with 0.5 mm length were used, in order to remove mesh dependent effects. Finally, since the simulation was conducted within the Abaqus/Explicit framework, the loading speed was increased to 50 mm/sec in order to reduce the computation time. Although, the simulation speed was drastically increased (1500 times) compared to the experiment, it was still considered to be quasi-static. Since the coupon's length scale was small, the effects from the speed increase should not influence the elastic deformation and the failure initiation load of the model.

Good correlation was found between the model and the experimental results. The model's deformation at crack growth initiation and the crack tip nodal displacements (circled) are shown in Figure 3.7. The nodal pairs at and in front of the crack tip experienced shear separation by relative displacement. The crack tip nodes also showed normal separation, as a sign of cohesive surface failure, thus releasing all constraints on the nodal pair. Figure 3.8 shows a color contour plot displaying the cohesive surface damage (CSDMG) parameter of the connected surfaces at the same time step. The CSDMG parameter displays the amount of "damage" sustained and amount of penalty stiffness degradation. Nodal pairs that have been completely separated would have a CSDMG value of 1 (gray in the Figure 3.8 contour); whereas a CSDMG value less than 1 indicated that the cohesive interaction between nodes remained intact. The contour map shows that failure initiated at the outer nodes at the time of pre-crack propagation. It should be noted that the cohesive surface penalty stiffness was a numerical parameter
artificially set to be magnitudes higher than the laminate's out-of-plane stiffness in order to remove its effects on the elastic behavior of the composite.

Figure 3.7. ENF Model deformation at the time of initial crack growth (the crack tip nodal pair are circled).

Figure 3.8. Color contour plot showing the cohesive surface damage parameter at the time of initial crack growth.
The FE model's force vs. displacement plot is shown in Figure 3.9 along with the experimental data. The stiffness of the finite element model was slightly higher compared to the experiments. This could be due to the differences in the boundary conditions of the experiments compared to the model. The support rollers used in the experiments gave rise to geometrical effects: as the specimens deformed, the contact points between the roller and the specimen changed, resulting in an apparent lowered stiffness. However, the model predicted the crack extension load to be 0.985 kN, which was on par with the experimental average of 0.943 kN. After failure initiation, the crack growth rate was less stable compared to the experiments and the load drop was more drastic possibly due to the dynamic effects in the sped up simulation.

Figure 3.9. Force vs. cross head displacement plot of the ENF coupons and FE model (a = 15 mm, 2L = 60 mm).
Mesh sensitivity study was conducted for the ENF test coupons. The peak force at crack growth initiation was monitored for various crack tip element sizes lengths, from 0.5 mm to 2.5 mm. Figure 3.10 shows the peak force at grow tip growth as a function of the element sizes. Note that the peak force remained at approximately 1 kN for all crack tip element lengths studied. Thus, the model is not mesh sensitivity for the range of element lengths between 0.5 mm to 2.5 mm. Larger element sizes were not examined due to the experimental crack tip growth length being approximately 1 mm. It would not make sense to use an element length that was greater than the damage size. Element length smaller than 0.5 mm was also not examined because it would become impractical to implement an extremely refined mesh in a larger fuselage panel impact test model.

![Figure 3.10. Peak force vs. crack tip element length for the ENF model.](image)
3.4 Shear Tie Coupons Experiments and Modeling

The next set of coupon-scale experiments were designed to examine the shear tie failure behaviors. As explained in the previous chapter, failures in the large-scale StringerXX and FrameXX impact experiments always initiated at the shear tie corners via delamination and fiber crushing/fracture, thereby softening the panel structure. Subsequent panel deformations were influenced by these early failure modes. In the StringerXX experiments, due to the fixed steel boundary conditions, sideway deflections of the shear ties were observed. Figure 3.11 shows the side view of the StringerXX experiment with lateral deformation of the panel and shear ties. Downward compression and opening moment loads were induced in the shear ties. The opening moment load at the shear tie corners led to interlaminar tension stress, and resulted delamination at these locations. Meanwhile, the downward compression of the shear ties caused crushing damage and local closing moments at the corner locations.

Figure 3.11. Illustration of StringerXX panel: (a) before impact and (b) during impact.
On other hand, the opening moment of the shear ties was counteracted by the C-frames' twisting moment during the FrameXX specimen impacts. Figure 3.12 shows the deformation of the FrameXX specimens' shear ties. In this case, movement of the shear tie was coupled to the twisting deformation of the C-frame. Impact loading was transferred from the bumper directly to the shear ties via contact, and then to the C-frames via the bolt connection. Since loading was applied away from the C-frame's shear center, the C-frames were subjected to twisting moments. Twisting moment in the C-frames countered the opening moment of the shear ties. Thus, the downward compression load on the shear ties was more crucial for the FrameXX specimen tests. Since the mixed moment and compression loading on the shear ties was complex to analyze, two separate series of shear tie coupons were designed to examine the effects of each individual loading.

Figure 3.12. Illustration of FrameXX panel: (a) before impact and (b) during impact.
3.4.1 Curved Beam Opening Experiments and Modeling

The first series of shear tie coupons were curved beam specimens designed to examine the shear tie response under pure opening moment. The curved beam coupon's geometry and construction were based on the shear ties from the panel experiments. Figure 3.13 shows the dimensions of the curved beam coupons. They were carbon fiber laminated beams with a 90 degree curved section of 10.16 mm outer radius. The flanges of the beams were 49 mm long. Their width and thickness were 38.1 mm and 2.54 mm, respectively. Unlike the shear ties, there was no curvature along the width direction of the curved beam coupons. In addition, 6.35 mm-diameter holes were drilled at the straight flanges of the coupons to mount them to the test fixtures. The layup was the same as the shear tie's layup, with twelve bi-directional fabric plies stacked in a \([\pm 45/0]_{3S}\) sequence.

Figure 3.13. Dimensions of the curved beam coupons. The diameter of the bolt holes was 6.35 mm.
The coupons were tested in a four point bending test setup using a uniaxial MTS testing machine. The experimental setup is shown in Figure 3.14. To generate a pure moment loading on the curved beam, two steel moment arms were bolted to the curve beam's flanges. The moment arms were 25.4 mm x 38.1 mm x 127 mm rectangular steel blocks, each with a 45 degree cut on one end. The 45 degree surfaces of the steel blocks interfaced with the flanges of the curved beam. The four point bending rollers interfaced with the moment arms. In addition, a slotted steel block was used to distribute loading for the bolt connection. Before the experiments, the sides of the coupons were painted white to visually accentuate any delamination damage.

![Figure 3.14. Curve beam opening test setup, the curved beam was painted partially white.](image)

The rollers in the four point bending test setup applied equal and opposite forces, equal to F, at the points of contact on the moment arms. The length of the moment arms,
L, was the distance between the pair of rollers in contact with each arm. A constant and pure moment load was generated on the curved beam, \( M = F \cdot L \). For this experiment, the moment arm length was set to 80 mm. An anisotropic curved beam subjected pure opening moment load would experience only normal stresses in the radial and hoop directions, and no shear stress [Lekhnitskii 1968]. Figure 3.15 shows the 2D free body diagram of a moment-loaded curved beam. The 2D stress components were defined as functions of \( r \) and \( \theta \) the in the polar coordinate system:

\[
\sigma_r = -\frac{M}{b^2 h g} \left[ 1 - \frac{1}{1 - c^{k+1}} \left( \frac{r}{b} \right)^{k-1} - \frac{1}{1 - c^{2k}} \cdot c^{k+1} \left( \frac{b}{r} \right)^{k-1} \right] \tag{3.3}
\]

\[
\sigma_\theta = -\frac{M}{b^2 h g} \left[ 1 - \frac{1}{1 - c^{k+1}} \left( \frac{r}{b} \right)^{k-1} - \frac{1}{1 - c^{2k}} \cdot c^{k+1} \left( \frac{b}{r} \right)^{k-1} \right] \tag{3.4}
\]

\[
\sigma_{r\theta} = 0 \tag{3.5}
\]

where

\[
c = \frac{a}{b}, \quad k = \sqrt{\frac{E_\theta}{E_r}} \tag{3.6}
\]

\[
g = \frac{1 - c^2}{2} - \frac{k}{k+1} \frac{(1 - c^{k+1})^2}{1 - c^{2k}} + \frac{k c^2}{k-1} \frac{(1 - c^{k-1})^2}{1 - c^{2k}} \tag{3.7}
\]

and \( a \) and \( b \) are the inner and outer radii as shown in Figure 3.15, \( h \) is the thickness of the beam, \( M \) is the applied moment, and \( E_\theta \) and \( E_r \) are the Young’s moduli in the \( \theta \) and \( r \) directions, respectively.
Of the two normal stresses, $\sigma_0$ (hoop stress) coincided with the 0 degree fiber direction (i.e., bending stresses). It was aligned with the laminate's "in-plane" directions and would not contribute to delamination. Conversely, $\sigma_r$ (radial stress) was oriented perpendicular to the plies and in tension. The interlaminar tension stress would pull the lamina apart in the radial direction and lead to delamination. A simplified, closed form solution to determining the maximum stress in a curved beam subjected to pure moment load was also presented [K Edward et al. 1989]. It was found that the maximum radial tension stress was located at the radial position $r = \sqrt{ab}$, and its value was

$$\sigma_{r,max} = \frac{3M}{2bh\sqrt{ab}}$$  (3.8)

Four curved beam coupons were tested at a quasi-static speed of 2 mm/min. During the experiments, the coupons exhibited linear elastic behavior up until failure.
Failure mode of the coupons was observed as sudden and catastrophic delamination at multiple interface layers between the plies in the curved region. A microphotograph of the delamination damage is shown in Figure 3.16. Four to five major delamination cracks were found at the interfacial layers between the fabric plies, with some cracks bridging into the fabric plies. However, the exact locations of the cracks were not consistent based on comparison with further curve beam coupons tests. Additional coupon testing showed that delamination damage could occur in all of the interfacial layers. While initial delamination location was expected to coincide with the location of peak radial tension stress, the initial delamination and its release of strain energy likely triggered delamination of the other interfacial layers. Delamination damage found was confined in the curved corner of the beam and did not grow into the flanges due to the lack of interlaminar tension stresses outside the corner.

Figure 3.16. Microphotography showing radial delamination of a curved beam coupon at multiple interfacial layers. Three of the crack also showed bridging into the fabric plies (cracks in the radial direction).
The forces and displacements data from these experiments were converted into normalized moments per unit width of the coupons and rotational deflections, and plotted in Figure 3.17. The moment vs. deflection plots reflected the catastrophic failure behavior of the beams. All four coupons exhibited linear elastic response up to a peak moment load, with consistent elastic rotational stiffness. At the point of failure, significant load drops and stiffness reductions can be observed across three of the coupons. The fourth coupon test not exhibiting large load drop because it was stopped early. The coupons failed at moment loads between 0.635 and 0.829 kN-mm/mm. According to the max radial stress equation (3.8), these failure moments corresponded to Interlaminar Tension (ILT) strengths of between 42.27 and 51.82 MPa, respectively, which were within the typical range of ILT strength values reported for the toughened matrix material.

Figure 3.17. Moment vs. deflection plots for the curved beam coupons.
3.4.2 Simulation of Curved Beam Opening Experiments

Development of accurate, physics-based shear tie finite element model involved accurately capturing its elastic and inelastic behaviors. A multi-layered modeling approach of composite materials was taken (see Figure 3.18) in order to accurately simulate the bending and failure responses. The number of elements in the thickness direction of the laminate had crucial effects on the simulation results. The Abaqus commercial FE code supported modeling of composite layup with a single element layer through the thickness with a composite layup definition, or as multiple elements stacked through the thickness each with its layup definition. A single layer of continuum shell elements was sufficient to model the elastic deformation behavior of composite plates. However, the stacked element model construction had numerous advantages for simulating damage. First of all, a multi-layered approached allowed for modeling delamination of the plies via cohesive zone modeling. Secondly, the accuracy of the model results generally improved with higher mesh density in the thickness direction. For solid elements, more elements through the thickness helped to prevent element lock up, and improve the deflection results. While for continuum shell elements (SC8R), transverse shear stresses were better estimated with more elements through the thickness. Finally, modeling the distinct plies allowed for more accurate prediction and enhanced visualization of the damage type, location, and size. For these reasons, a detailed, stacked-element model was chosen over the single element layer model.
However, the computational cost of such a detailed model was prohibitively high. So, ply groups were created in the multilayer structure to reduce the number of interface layers and simulation time. The curved beam coupon specimens originally consisted of twelve fabric plies, and delamination could occur in any of the interface layers between the plies. Sublamine groups were created where delamination would not occur within each group. Each sublamine consisted of multiple fabric plies and was modeled with a single element through the thickness. Cohesive surface interactions were then applied between the sublaminates to simulate delamination. Figure 3.18 illustrates the representation of the curved beam layup as sublamine groups. Six sublaminates were used for the curved beam opening model in the through-thickness direction. Each sublamine was represented with one layer of continuum shell element (CS8R) stacked in the thickness direction, defined with its own layup definition.

![Figure 3.18. (a) Curve beam layup, (b) sublamine (ply group) configuration for modeling.](image)

The multi-layered finite element model of the curved beam coupon was created in
Abaqus/Explicit, as shown in Figure 3.19. Six separate continuum shell layers were assemble to form the curved beam. The model's geometry was consistent with those shown in Figure 3.13. However, only half of the curved beam was modeled to take advantage of the symmetry plane of the curved beam at the mid-span location. The curve beam laminate buildup consisted of both 0 and 45 degree fabric layers. Using a half symmetry model was valid because the bi-directional fabric layers had similar stiffness in both fiber directions. Therefore, geometrical and material symmetry planes existed at the mid-length location, and the half model would not alter the simulation.

Figure 3.19. Half symmetry curve beam model with a 6 element layer construction. The orange mesh represents the moment arms, and the red lines represent the cohesive interface zones.

Cohesive surface interactions were applied to the contacting surface pairs between the element layers to model delamination failures. These are represented by the red lines in Figure 3.19. The cohesive surfaces were applied only to areas of potential failure, in
the curved corner of the beam. Displacement tie constraints were used to connect the contacting surface pairs outside of the areas of interesting, in the flange. Likewise, a dense mesh of 0.66 mm was used in the curved corner to properly capture the delamination failure, and a coarser mesh of 0.8 mm was used in the flange to capture the bending behavior.

Instead of the moment arms in the experiment, the rotational displacement was applied to the curved beam model via two steel blocks (orange meshes in Figure 3.19) attached to the sides of the curved beam's flange via surface tie constraints. The 6.35 mm diameter holes and the bolts were not modeled. The two steel blocks were kinematically coupled to displace as a single rigid unit throughout the simulation, similar to how the bolted moment arm behaved in the experiment. The steel blocks were given a very coarse mesh because they were rigid in the simulation. Intra-ply failure was model with the Hashin-Rotem failure criteria. Roller symmetry boundary conditions were applied to the model at the symmetry plane (see Figure 3.19). A roller was also boundary condition was also applied to the edge of the steel block below the curve beam model to allow the correct movement of the model. Finally, a rotational boundary condition of 0.3 radian (17.19 degree) was applied that point at a dynamic rate of 3 rad/sec.

The catastrophic delamination failure of the curved beam coupons was successfully predicted by the model. The elastic response, peak moment and failure mode correlate very well with the experimental data. Figure 3.20 show the elastic bending deformation of the model before and after failure. Figure 3.21 shows the
moment vs. deflection curves of the model and the coupon specimens. The half model's bending moment was found by integrating the normal stress component at the symmetry plane of the curved beam, and its opening deflections were multiplied by a factor of 2 because it was half the deflection of the full curved beam model. The model correctly predicted the linear elastic behavior of the loading curve, although the elastic stiffness of the model was slightly higher compared to the experiments. The predicted maximum moment was 0.742 kN-mm/mm at 0.584 radians of opening deflection. These fall between the maximum moment and deflections.

Figure 3.20. (a) Elastic bending response of the curved beam model, and (b) the model after delamination failure.
Figure 3.21. Moment vs. deflection plot of the curved beam coupons and the FE Model.

Figure 3.20b shows the model deformation immediately following the delamination failure and load drop. A close up view of the model's curved corner at failure is shown in Figure 3.22 with contours of cohesive surface damage parameter shown, where a value of 1 indicates failure. The delamination damage was predicted to occur in all of the cohesive interfaces. The extent of the delamination was more widespread compared to experimental coupon shown in Figure 3.16. This was due to the plies being lumped into element layers. The individual element layer could not failure in delamination, so it was stiffer compared to the group of plies it represented that could delaminate in the experiment. Thus, large relative displacement between the element layers and delamination failures were predicted by the model. Another reason for the model's extensive delamination damage was the difference in simulation's loading speed.
was significantly increased from the actual test speed. Extensive propagation of failure could be the result of dynamic effects after failure initiation. Other limitations of the model include that it could not capture delamination cracks at an interfacial layer unless the cohesive surfaces were pre-specified. Delamination was only allowed to occur at every other interfacial layer because the plies were lumped into element layers. For the same reason, the crack bridging seen in Figure 3.16 was also not captured as the elements failure would only occur when both plies experience fiber failure. However, the overall response of the model was still satisfactory.

![Figure 3.22. Close up view of the delamination simulation of the curve beam model.](image)

3.4.3 Shear Tie Compression Experiments

The second series of shear tie coupons were subjected to direct compression to recreate the loading scenario of shear ties under impact loading, as seen in the FrameXX
specimens. Compression load from the impact caused the shear ties to delaminate, crush, and finally fracture during post-buckling (see Figure 2.44). Once fractured, the shear ties completely lost their load bearing capability. The C-frames were then allowed to twist, leading to wide-spread panel failures. The goal of the second series of shear tie experiments was to understand the failure behavior and create models for to the shear tie crushing and fracture failures, in addition to delamination.

The shear tie compression coupons' dimensions are shown in Figure 3.23. These were 66 mm long sections of shear ties. Therefore, the coupon's cross-section geometry and construction were identical to the shear ties bolted to the FrameXX specimens. These coupon specimens retained the 2.79 meter radius of curvature in the length direction. In the FrameXX specimens, the shear ties were bolted to the panel's skin and the C-frames at the shear tie web and flange, respectively (see Figure 3.12).

During the impact experiments, load was transferred from the skin into the shear ties and into the stiffening C-frames. To replicate these boundary and loading conditions, two bolt holes (6.35 mm diameter) were drilled at the coupon's web, as shown in Figure 3.23, matching the hole locations in the impact experiment's shear ties. The web of the coupon would be bolted to a fixed aluminum base plate. Compression loading was applied directly at the top of the vertical flange. The 55 mm long vertical flange of the shear tie coupon was designed to match length of the flange section between the curved corner and the bolt line (see Figure 2.4), in order to replicate the buckling failure of the shear ties.
Figure 3.23. Shear tie compression coupon dimensions. The diameter of the bolt holes was 6.35 mm.

Figure 3.24 shows the compression coupon in test setup. The shear tie coupon was painted white to visually accentuate damage. It was bolted to a curved aluminum base plate, similar to the shear tie-to-skin connection of the StringerXX and FrameXX specimens. An aluminum loading plate was used to apply compression loading at the top of the shear tie flange via contact. The shear tie flange was fitted into a slot in the loading plate, thus creating a pinned boundary condition at the point of contact. This boundary condition was simplified compared to the C-frame and shear tie bolt connection in the large panel impact experiment. The tendency of the C-frames to twist during the impact experiment would induce some closing moments on the shear ties (see Figure 3.12). The moment component of the applied load was removed due to the complicated process of determining the exact moment load on a single shear tie caused by the C-frame
twisting in a redundant structural system. The simplified compression test setup presented in Figure 3.18 also simplified the analysis. The test setup was mounted to an MTS uniaxial testing machine to apply compression loading on the coupon specimen.

During loading, the compression force applied at the flange was converted into bending and shearing stresses at the curved corner of the shear tie. A simplified free body diagram of the shear tie compression test is shown in Figure 3.25. The shear tie corner was represented as a 90° section of a curved beam with a constant radius, R. It was fixed at point B, with a point load applied normal to the end of the beam at point A. According to Lekhnitskii [Lekhnitskii, 1968], the through-thickness transverse shear force varied sinusoidally with the angle \( \theta \), along the curvature of the beam. Figure 3.26 show the shear force and moment diagrams plotted against the Cartesian coordinate direction X. The highest transverse shear stress occurred at a location where the direction of the shear force coincided with the thickness direction of the curved beam. In this case,
it was found at the fixed end of the beam (location B in Figure 3.25). Meanwhile, the moment varied linearly with X. Its highest value also occurred at the fixed end of the beam (location B). The opening moment and through-thickness shear stresses contributed to the delamination damage via Mode I and II loading, respectively. Therefore, delamination damage was expected to initiate at or near location B.

Figure 3.25. Simplified free-body diagram of the shear tie compression experiment.

Figure 3.26. The shear force and moment diagrams of the shear tie’s curved corner.
The shear tie coupon compression experiments were conducted quasi-statically at 6 mm/min. Figure 3.27 shows the force vs. actuator crosshead displacement plot of that test, and Figure 3.28 shows the key failure modes observed during a shear tie coupon compression experiment. The force and displacement corresponding to each photograph in Figure 3.28 are indicated in Figure 3.27. Figure 3.28a shows the coupon's deformation under compression loading. As compressive displacement was applied, the shear tie curved corner showed opening deflection in response to bending moment. The flange also showed some lateral deflection in compression. This deformation mode corresponded to an initially linear elastic load vs. displacement response. Initial damage occurred at 2 mm of compressive displacement and 7.0 kN load. Figure 3.28b shows the two delamination cracks that were formed at the curved corner of the beam, causing the first load drop at point b in Figure 3.27. The delamination appeared to be closer to the fixed web end of the curved, as predicted based on the shear and moment analysis.

Figure 3.27. Force vs. crosshead displacement plot of a shear tie compression coupon. The letters labeled in the plot correspond to events shown in Figure 3.20.
Figure 3.28. (a) Elastic deformation of shear tie compression coupon, (b) initial delamination at the curved corner, (c) crushing failure and flattening of the corner, (d) post-buckling bending failure of the flange.

Increasing compressive displacement from that point onwards produced further delamination at the other interface layers. The cumulative delamination damage corresponded to the decreasing stiffness of the loading curve between 2 and 5 mm of cross head displacement (see Figure 3.27). At 4.9 mm of compressive displacement, the
shear tie corner had fully flattened. However, the flange remained vertically oriented, enabled due to delamination and local bending and crushing failure at the shear tie curved corner (i.e., initially curved corner turned into sharp corner geometry). Figure 3.28c shows the ply failures in the shear tie corner driven by the highly localized transverse shear and bending stresses.

Since the vertical flange made contact with the base plate, the crushed/softened shear tie corner no longer played a role in the load path. The compression load was now directly sustained by the vertical shear tie flange. Hence, the stiffness of the coupon increased past 5 mm crosshead displacement in Figure 3.27. Additional compression caused bending and lateral deflection of the shear tie flange until buckling failure of the flange occurred at 13.9 mm of displacement (at 10.4 kN of compression load). The final failure is shown in Figure 3.28d. Two additional shear tie compression experiments were conducted with the identical failure modes and very similar load vs. displacement plots. These additional shear tie data plots are shown in Figure 3.32, along with the finite element model comparison.

### 3.4.4 Shear Tie Compression FE Model

The shear tie compression experiment was simulated in Abaqus/Explicit. Cohesive surface interactions were used for simulating inter-ply delamination. A multi-
layered shear tie coupon model was built in order to capture the elastic bending response, transverse shear stresses, and the complex, mixed failure modes that occurred at different locations. An attempt was made Abaqus’ default fiber composite failure criterion was the Hashin failure criteria. However, due to the existence of through-thickness lamina failure modes, the Hashin failure criterion was inadequate because it could not account for intraply transverse shear failure modes (i.e., crushing and fracture damage). Fiber fracture damage at the shear tie corners occurred due to transverse shear and bending stresses. To model the intraply failures, Abaqus’ Hill failure criterion for anisotropic materials was utilized to achieve the desired 3D stress failure modes. The Hill criterion failure envelope was defined as:

\[ I_F = \frac{\sigma_{11}^2}{F_1^2} + \frac{\sigma_{22}^2}{F_2^2} + \frac{\sigma_{33}^2}{F_3^2} - \frac{\sigma_{11}\sigma_{22}}{F_1^2 F_2^2} - \frac{\sigma_{22}\sigma_{33}}{F_2^2 F_3^2} - \frac{\sigma_{11}\sigma_{33}}{F_1^2 F_3^2} + \frac{\sigma_{12}^2}{F_1^2 F_2^2} + \frac{\sigma_{23}^2}{F_2^2 F_3^2} + \frac{\sigma_{13}^2}{F_1^2 F_3^2} = 1 \]  (3.9)

Failure initiation was triggered when a stress component exceed the material's strength. Also, the mixed normal stress mode failures were also accounted for. The variables F1, F2, F3, F12, F23, and F13 were material strengths partially defined in Table 3. This failure criterion was similar to the Hill-Tsai criterion, with the exception that compressive and tensile material strengths are not distinct. For example, F1 represented both the compressive and tensile stress limit in the 1-1 direction. However, this was acceptable for modeling the X840 Z60 plain wave carbon fiber fabric because there was no significant difference between its in-plane compressive and tensile strengths. According to Daniel and Ishai 2005, the in-plane tensile strengths of a similar carbon
The elastic to brittle failure behavior of the carbon fiber reinforced epoxy was modeled by using a modified damage law for ductile damage. By setting the equivalent plastic strain at failure to zero, the plasticity zone was removed and the material transitioned into the damage regime once its elastic limits were reached. Figure 3.29 shows the stress-strain curve describing the intra-ply failures. At the onset of failure, equivalent strain was used in this damage law to govern the overall failure parameter. A single fracture energy value was assigned for the combined intraply damage modes. Since ply fracture occurred via bending (in-plane tension and compression stresses) and out-of-plane shearing of fibers, the fracture energy was approximated as 45.8 KJ/m² based on the fracture energy of the T300-913 carbon fiber fabric reinforced toughened epoxy [Pinho et al. 2006].
A shear tie compression model was created in Abaqus/Explicit with its layup separated into ply groups to allow for the delamination modeling, as shown in Figure 3.18. This model is shown in Figure 3.30. Geometrically, the model was identical to the test specimen. It was built based on the curved beam opening model from the previous section with six continuum shell (SC8R) element layers stacked in the laminate thickness direction. The cohesive surface interactions (red lines in Figure 3.30) were applied in the curved corner and 1 mm into the flange and web, delamination failure could grow into. Displacement tie constraints were used to connect the contacting surface pairs outside of the areas of potential failure. The aluminum base plate from the experiment was also modeled to simulate the contact between the shear tie corner and the plate during the experiment. The aluminum plate was modeled with solid C3D8R elements (orange in Figure 3.30). However, the bolts between the shear tie and the plate was not modeled. The shear tie coupon model was fixed to the aluminum support plate via surface based displacement tie constraints at the bolt locations. Contact between the shear tie elements
and the aluminum plate element were accounted for by using a "hard" pressure-overclosure relationship between their surfaces. Penalty contact constrain was used to minimize the penetration of the shear tie node into another aluminum plate surface, while the contact pressure are transmitted between the surfaces. The contact formulation was frictionless.

Figure 3.30. Shear tie compression model with a 6 element layer construction. The orange meshes represent the aluminum base plate, and the red lines represent the cohesive zone interfaces.

The aluminum based plate was fixed in displacement. The loading plate from the experiment was not included in the model. Instead, a roller constraint and a compressive displacement were applied to the top of the shear tie flange to achieve the same boundary and loading effect. The displacement was applied at a dynamic rate of 0.5 m/sec, same as the loading rate of the dynamic impact experiments. A mesh size of 1.27 mm was used in the curved corner region of the shear tie; while a mesh size of 3.3 mm was used in the
flange of the shear tie. A mesh size of 2 mm was used for the aluminum base plate to properly simulate the contact between the shear tie corner and the plate.

The deformation plots of this model are shown in Figure 3.31. Key failure events are shown in an order analogous to Figure 3.27. Figure 3.32 shows the FE model's force vs. applied displacement results, plotted along with the shear tie compression experimental data. The force and displacement corresponding to each image in Figure 3.31 are indicated in Figure 3.32. The elastic uploading curve (Figure 3.31a), the peak force at shear tie delamination (Figure 3.31b), and the crushing failures (Figure 3.31c) at the curved corner were very well captured.

Although this model produced accurate predictions of the delamination and crushing failures observed in the experiments, crushing of the elements at the shear tie corner eventually led to instability issues in the simulation. Element instability was the most prominently after significant crushing and during post-buckling failure, as shown in Figure 3.31d. The instability was the result of stiffness degradation in the shear tie corner elements upon damage initiation, which allowed large element distortions to develop within these elements. Since each element consisted of a sublaminate of two fabric plies (as shown in Figure 3.18), element removal from the simulation was not possible until both plies in the sublaminate group were damaged. The continuum shell elements could not retain their aspect ratio. Excessively distortion of the elements invalidated the model's stiffness matrix and led to the termination of the simulation. This failure occurred before the elements were fully degraded. The stability issue was aggravated
when the six layer shear tie model was implemented into the large FrameXX fuselage specimen model because the increase in the number of shears ties.

Figure 3.31. (a) Unloaded shear tie compression model with 6 layers of continuum shell elements through the thickness, (b) initial delamination at shear tie corner, (c) crushing failure and flattening of the corner, (d) buckling failure of the flange with large distortion of the corner elements.
To overcome the element instability issue, the shear tie model was updated by further increasing the number of elements through the thickness of the shear tie to twelve layers. Figure 3.33 shows the model's construction through the thickness. Each sublaminate group was now represented with two element layers. As such, each laminate ply was represented with one layer of elements stacked in the thickness direction. The nodes were shared between the plies in each sublaminate group. For example, plies #1 and #2 shared nodes at their contacting faces, as did plies #3 and #4, and so on. Cohesive surface interactions were applied at contacting surfaces of sublaminate groups, as shown in Figure 3.33. Since the plies were no longer grouped, fiber failure of the ply and the removal of its elements were independent on the other plies. A shear tie corner element was deleted when its respective ply sustained damage, and its stiffness was reduced by
95%. Both the continuum shell elements SC8R and the 8-node brick element C3D8R were tested within this approach to determine their feasibility for modeling fiber crushing. Although the Abaqus continuum shell element was more computational efficient for with few elements required in the thickness direction to capture bending, the solid element was more suitable for simulating crushing damage. Element distortion control and enhanced hourglass control were enabled to improve the stability of the 8-node solid element (C3D8R). Element distortion control prevented nodal movement past the center of the element. The enhanced hourglass resisted zero energy displacement modes caused by reduce integrated of the element. Both the distortion control and enhanced hourglass control increased the element's stability by adding penalty stiffnesses to the element.

![Updated shear tie model configuration, the red lines are cohesive zone interfaces.](image-url)
Figure 3.34 shows a series of deformation plots for the twelve layer solid element shear tie FE model. The updated model's force vs. applied displacement results have been plotted in Figure 3.35 along with the experimental data. The force and displacement corresponding to the images in Figure 3.34 are indicated in Figure 3.35. Figures 3.34a, b, and c show that the general failure behaviors were captured at the update solid model. More importantly, as shown in Figure 3.34d, the solid element shear tie model was much more stable and was able to simulate fracture at the shear tie curved corner without erroneous element distortion modes. However, as a result of the distortion control scheme and the added penalty stiffening to the solid elements, the initial uploading curve, the delamination load, and the post-buckling fracture failure of the shear tie were not accurately captured. Figure 3.35 shows that the initial elastic curve appeared to be stiffer compared to the experimental data. The peak force at delamination was higher at 7.39 kN, compared to the experimental average of 6.63 kN. Finally, the post-buckling bending fracture was not predicted. The shear tie flange was too stiff to bend and fail by buckling. Instead, fracture occurred at the shear tie's curved corner. After the ply element failure initiated, the contact force oscillated due to the high contact stresses experienced by the elements located at the shear tie corner. The failure criteria were triggered at the shear tie corner, and elements were degraded and removed from the simulation. Thus, the shear tie model severed at the shear tie corner.
Figure 3.34. (a) Unloaded shear tie compression model with 12 layers of solid elements through the thickness, (b) close up view of the initial delamination at shear tie corner, (c) flattening of the shear tie corner, and (d) crushing failure and severing at the shear tie corner.
Mesh sensitivity of the shear tie compression was examined in both the through thickness direction and in-plane direction of the laminate. Figure 3.36 shows the compression force vs. flange tip displacement plot for the shear tie model with 4, 6, 12 continuum shell model up to 8 mm of flange tip displacement. The initial failure of these models was delamination at around 1.7 mm of displacement. All three models predicted this failure; however, the model with 4 layers of shells overpredicted the failure load at 9.9 kN. As the number of elements through the thickness increased to 6 and 12 layers, the peak force at delamination converged to approximately 6.2 kN. Crushing failure of the shear tie curved corner was also different in the 4 layer model, which did not show decreasing compression forces after the peak load was reached until 7.4 mm.
Figure 3.36. Compression force vs. flange tip displacement plot for the shear tie model with different number of element layers through the thickness.

Figure 3.37 shows the model's sensitivity to the element size along the curvature direction of the 6-layered continuum shell and the 12-layered solid element shear tie model. The peak force for delamination failure of the 6-layered shell model, and fiber fracture failure of the 12-layered solid model were examined. The range of element lengths examined was between 0.5 to 3 mm. Larger element sizes were not considered because the outer radius of curvature at the shear tie corner was 10.16 mm. At least 4 elements along the curvature were required to model the bending of the curved section. Also, element lengths smaller than 0.5 mm were not practical for the application in large fuselage panel model. The plot shows that the peak forces at delamination and fiber fracture did not vary beyond 10% for the range of element lengths between 0.5 to 3 mm. The solutions seemed to be convergent with reducing element sizes.
3.5 D-Shaped Bumper Compression Experiments and Modeling

FE simulation of the StringerXX specimens required the accurate modeling of the D-shaped rubber bumper used in these experiments. In the impact experiments, the rubber bumper and the fuselage panel were both deformable, and their deformations were interdependent. Therefore, it was necessary to model the rubber bumper material behavior, deformed geometry, and contact area in order to model the panel stresses in the impact zone. A D-shaped rubber bumper is shown in Figure 3.38. The bumper's base width and depth were 198 mm and 76 mm, respectively. However, its wall thickness and outer radius of the D-shaped bumper were variable. In addition, the rubber material used to manufacture the bumper exhibited hyperelastic behavior, with increasing compressive
stiffness as it deformed. As a result, the contact pressure between the bumper and the impacted structure varied with the change in bumper thickness.

Figure 3.38. A D-shaped rubber bumper. The bumper's base width and depth were 198 mm and 76 mm, respectively.

To examine the D-shaped bumper compression behavior, a series of isolated bumper experiment were conducted, in which the bumper was quasi-statically compressed between two parallel steel plates. The experimental setup is shown in Figure 3.49. The two steel plates were attached to the test fixtures in an MTS uniaxial testing machine. They were 304.8 mm (length) x 101.6 mm (width) x 38.1 mm (thickness) in dimension, and they remained rigid throughout the compression experiment. The flat base of the bumper then glued to top steel plate with Polyepoxy resin. The bottom steel plate made contact with the bumper during testing. The four bumper compression tests were ran at a rate of 6 mm/sec until a contact force of between 60 to 90 kN was developed. The contact force vs. displacement plots are shown in Figure 3.40. The four bumper compression experiments were very consistent. The contact force was initially very low before the hollow gape in the bumper's D-shape was closed. At approximately
45 mm of actuator displacement, the D-shaped bumper collapsed, and the bumper's stiffness and load increased rapidly to the target force. Note that the bumper compression experiments were purely hyperelastic and the bumpers rebounded after the experiment.

Figure 3.39. D-shaped bumper compression test setup (left) and fully collapsed bumper (right).

Figure 3.40. D-shaped bumper compression test data.
A half model of the rubber bumper compression experiment is shown in Figure 3.41. The bumper model's dimensions were determined by tracing the physical bumper's edges into a piece of engineering paper, and then input into the model. The two steel plates were also included in the model (orange meshes in Figure 3.41). The half bumper was attached to the top steel plate by displacement tied constraint. Reduced integrated solid elements C3D8R were used for aluminum plate, while solid elements with incompatible mode formulation (C3D8I) were used for the rubber bumper. Contact between the bottom plate and the bumper was accounted for by a "hard," penalty contact interaction. On the other hand, the Lagrangian contact interaction was assigned to the bumper's self-contact during its collapse. A friction coefficient of 0.5 was assigned to contacts.

Figure 3.41. Half model of the bumper compression experiment.
Symmetry boundary conditions were applied to the symmetry plane of the model at both the bumper and the aluminum plates. Also, the bottom plate was fixed in displacement, while a constant, downward velocity of 0.5 m/sec was applied to the top plate. Rectangular mesh with aspect ratios close to 1 was applied to the bumper (see Figure 3.41) to help minimize element distortion errors. The bumper's element size was 6 mm, while the aluminum plates' element size was 12.7 mm.

Due to its irregular shape and collapsing deformation during impact, modeling of the rubber bumper presented a major challenge. Simulation of the large deformations of nearly incompressible rubbers was well known to introduce numerical instabilities by excessive element distortion. This problem was overcame by adding element distortion controls and enhanced hourglass controls to the bumper model, similar to in the shear tie compression model. In addition, the Ogden hyperelastic material model was used to define the rubber's constitutive behavior. The Ogden model is a highly stable material model that can achieve simulation results up to 700% compressive strain [Kim B. et al. 2012]. It is written as a strain energy potential function in terms of the deviator principle stretches and two material parameters.

\[
U = \sum_{i=1}^{N} \frac{2\mu_i}{\alpha_i^2} \left( \bar{\lambda}_1^{\alpha_1} + \bar{\lambda}_2^{\alpha_2} + \bar{\lambda}_3^{\alpha_3} - 3 \right) \tag{3.10}
\]

where \(\mu_i\) are related to the shear modulus, and \(\alpha_i\) are dimensionless real numbers. \(\bar{\lambda}_i\) are the deviatoric principal stretches components and \(N\) is the strain energy potential order.
The Ogden parameters used in study experiment were determined from fitting the model to a test data set for a similar chloroprene rubber material presented by Bergstrom [Bergstrom 1999]. The Ogden parameters are summarized in Table 3.5.

Table 3.5. Ogden material parameters N = 2.

<table>
<thead>
<tr>
<th>i</th>
<th>( \mu_i ) (MPa)</th>
<th>( \alpha_i )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>66.59</td>
<td>3.564</td>
</tr>
<tr>
<td>2</td>
<td>494.43</td>
<td>-0.1485</td>
</tr>
</tbody>
</table>

To assess the similarity between Bergstrom's rubber material and the rubber material in GSE bumpers, simulation was conducted to compare the overall deformed bumper shape and the contact force data to the experimental results. The fully collapsed bumper model is shown in Figure 3.42, which matches the deformed shape of the physical bumper shown in Figure 3.39. Elements at the inner corner of the D-shape experienced high shear and compressive deformation upon bumper. However, they did not show signs of distortion, which allowed the model to continue to completion. In addition, a nearly exact match for contact force vs. displacement plot was found between the model and the experiment, as shown in Figure 3.43. The bumper model showed the load increase upon collapsing, at the same displacement. The simulation was ran well beyond the 90 kN level of the experiment, proving that the bumper model could remain stable at much higher loads and would not lead to stability errors when implemented in the StringerXX panel models.
Figure 3.42. The fully collapsed rubber bumper model (with the Ogden material definition).

Figure 3.43. D-shaped bumper compression test data and FE result.
3.6 Stringer Element Experiments and Modeling

3.6.1 Stringer Element Experiments

The final set of small scale experiments was the stringer element indentation experiments. The goal of this experiment was to examine the formation of skin damage during the impact experiments. As discussed in the previous chapter, GSE impact damage detection was an issue because the composite fuselage skin may not leave visible signs of damage after impact. The thin composite fuselage skin did not show global deformation, and was inclined to rebound even after fuselage panel sustained significant internal damage. Thus, the minor skin damage found in the StringerXX and FrameXX specimen tests could serve as the only indicator of the impact. Also, it was beneficial to establish the accuracy of using finite element model to predict the degree of skin crack damage due bending. For these reasons, the skin-stringer element indentation test was designed based on StringerXX specimens that were loaded on the panel skin directly over a stringer.

Each stringer element specimen consisted of a section of the stringer co-cured to the curved skin. They were cut from a pristine StringerXX panel specimen. Figure 3.44 shows the dimensions of the stringer element cut. Additional cross-sectional details can be found in Figure 2.2. As shown in Figure 3.44, the width of the specimen skin was 200
mm, and its depth was 76.2 mm. The hat-shaped stringer was co-cured to the skin at its flanges. The span of unreinforced skin between the skin-stringer joints was 101.6 mm.

Figure 3.44. Dimetric view of the stringer element specimens cut from a pristine StringerXX panel. Its detailed cross-sectional dimensions are shown in Figure 2.2.

The stringer element specimens were quasi-statically indented at the external skin to excite skin crack failures. A MTS uniaxial testing machine was used to test the specimen. In order to mount the specimens, the specimens' stringers were partially inserted into Polyepoxy resin inside an aluminum C-channel. When the Polyepoxy hardened, a fixed boundary condition was created at the bottom of the stringer. Figure 3.45 shows the cross-section view of the specimens with the region of the stringer that was fixed in hardened epoxy. The vertical distance between the specimen's outer skin surface and the epoxy was 34 mm. A potted test specimen and its quasi-static indentation test setup are shown in Figure 3.46. The aluminum C-channel was bolted to test fixtures. The indentation loading was applied via a 76.2 mm wide D-shaped rubber bumper clamped to the opposing cross-head of the test machine. The bumper indentation location
was centered over the stringer, similar to the Stringer00, 03, and 06 experiments. In the stringer element tests, the sides of the specimen were painted white to enable better visual observation of crack formation.

![Cross-sectional view of the stringer element specimen showing its potted region.](image)

**Figure 3.45.** Cross-sectional view of the stringer element specimen showing its potted region.

![Stringer element specimen potted in Polyepoxy in an aluminum C-channel, and Quasi-static indentation test setup of the stringer element.](image)

**Figure 3.46.** (a) The stringer element specimen potted in Polyepoxy in an aluminum C-channel, and (b) Quasi-static indentation test setup of the stringer element.

A free body diagram of the stringer element indentation test is shown in Figure 3.48. During contact, the bumper deformed to apply a non-uniformly distributed pressure load over the skin surface of the specimen. The skin was supported by the diagonal walls
of the stringer, which were fixed in the Polyepoxy resin. Based on beam theory, the maximum moment, $M$, and shear force, $V$, were expected to occur at the joining locations between the skin and the stringer.

![Free body diagram of the stringer element subjected to quasi-static indentation against the bumper. The locations of maximum bending and shear stress are marked with red squares.](image)

Figure 3.47. Free body diagram of the stringer element subjected to quasi-static indentation against the bumper. The locations of maximum bending and shear stress are marked with red squares.

The skin stress state could then be found by assuming a linear moment stress distribution and parabolic shear stress distribution. The in-plane moment stresses and the out of shear stresses are:

\[
\sigma = \frac{Mz}{I} \tag{3.11}
\]

\[
\tau = \frac{3V}{2t} \left(1 - \left(\frac{2z}{t}\right)^2\right) \tag{3.12}
\]

where $z$ is the position along the out-of-plane direction of the skin, with the origin on the mid-plane of the skin, $t$ is the skin thickness, $I$ is the skin's second moment of area. Figure 3.48 shows the bending and stress profiles through the laminate skin thickness.
The peak in-plane tension and compression stresses were found at the exterior and interior surfaces of the skin, respectively. Thus, skin cracks were expected to initiate at the plies on the surfaces of the skin. The stringer and skin joining locations also represented points of geometrical and stiffness discontinuity. Therefore, concentrated stresses higher than those predicted with beam bending equation may also developed at these locations.

Two stringer element specimens, SE01 and SE02, were tested. The first specimen, SE01, was quasi-statically indented until barely-visible skin damage was generated. The second specimen, SE02, was indented until through-thickness skin cracks were found. Figure 3.49 shows the load vs. displacement plots of the two specimens. The initial loading curves were elastic deformations. As the rubber bumper's hollow cavity collapsed, the contact force increased rapidly due to increased stiffness of the bumper. At the maximum contact force, the specimen's skin bent locally at the joining
location between the skin and the stringer. The stress concentration on the skin near the joint location led to the desired skin failure.

During the SE01 indentation test, matrix level damage initiated at a loading of 14 kN and continued to accumulate thereafter. Low energy clicking noises were heard at this time. At 19.04 kN of loading, the specimen experienced initial skin failures, accompanied by higher energy clicking noises. Skin surface cracks were found to coincide at geometrical discontinuity of the skin where the stringer joined. These surface cracks were only visible when examined under a glancing light source, as shown in Figure 3.50. Surface undulation around the crack locations also indicated residual skin deformation. Like the skin cracks, however, the residual deformation was only visible with a glancing light source.

![Figure 3.49. Load vs. displacement plot of the shear tie Compression experimental results.](image-url)
Figure 3.50. SE01 after skin crack failure; with a glancing light source, two visible skin cracks were found at the skin-stringer joining locations.

Figures 3.51 shows photomicrographs of the SE01 specimen at the one of the geometrical discontinuity locations before and after the indentation experiment. Figure 3.51a shows the skin and stringer's laminate buildup before the specimen was painted white, and Figure 3.51b shows the fiber and matrix crack formations after being indented accentuated by the white paint. In-plane ply failure was be found on the surface plies of the skin, along with some delamination between the plies. The skin's laminate buildup consisted of 18 plies: a [0/45/90/-45]_{2S} unidirectional tape layup and one additional 0° 3K fabric plies on each outer surfaces. Based on the position of the cracks, it was concluded that tension cracks were formed in the outer 0° fabric ply and two unidirectional plies (0° and 45°) on at top surface of the skin. On the other hand, diagonal cracks were found at the outer 0° fabric ply and three unidirectional plies (0°, 45°, and 90°) at the skins bottom surface. These diagonal cracks were created from the mixed shear and compression stresses that resulted in tension stress perpendicular to the crack. Additional
delamination cracks were also found to continue from the in-plane ply cracks. It was concluded from these images that when skin crack damage was initiated, the outer plies of the laminate experienced in-plane fracture, coupled with some inter-ply delamination.

![Photomicrographs of the SE01 specimen: (a) before the test, and (b) after the test, with ply cracks and delaminations at the skin-stringer joining location](image)

Figure 3.51. Photomicrographs of the SE01 specimen: (a) before the test, and (b) after the test, with ply cracks and delaminations at the skin-stringer joining location

After successfully capturing the initial skin cracks with SE02, the specimen SE02 was tested in two loading cycles in an attempt to examine damage visibility of failures before and after the initial skin cracks. The two loading cycles were combined to show only new loads and displacements in Figure 3.49. During the first loading, continuous low energy clicking sound initiated at 14.1 kN load and 56.6 mm displacement that indicated matrix failures. Loading cycle ended at this point to examine the damage. However, no externally visible damage or residual skin undulation was found. This reaffirmed that the initial matrix failures were externally invisible and did not contribute to stiffness change. During the second loading, the severe stiffness change occurred at approximately 18.0 kN and 58.8 mm, the failures were likely fiber failures and delaminations, similar to those observed in SE01. A lower failure initiation load was
recorded compared to SE01 (fiber failure at 21.55 kN). Afterwards, the loading curved entered a detour. Between 58.8 mm and 69.6 mm of cross-head displacement, cumulative fiber failures occurred that caused small load drops. The final failure at 20.1 kN and 69.6 mm was through-thickness skin crack. Figure 3.52 shows the damage formation in SE02 after the test. The final skin crack was found at the skin-stringer joint location where the shear and bending stresses were the highest due to geometrical discontinuity and stiffness change.

![Figure 3.52. SE02 with through-thickness skin crack at one of the skin-stringer joint location.](image)

### 3.6.2. Stringer Element Indentation FE Model

A half-symmetry finite element model of the stringer element indentation experiments was created in Abaqus/Explicit to access the model's ability to predict the formation of surface visible cracks. The half-symmetry approximation was valid because the laminate layup was quasi-isotropic. The half model is shown in Figure 3.53. The
stringer and skin components were created separately, each with its own composite laminate layup definition. The intra-skin delamination damage was not a major failure mode in these specimens. Thus, the skin and stringer where each represented with a single layer of continuum shells in the through thickness direction. In addition, displacement tied constraint was used to represent the joint between the skin and stringer flange. To model the fracture failures due to bending and shear stresses, the Hashin-Rotem failure criterion was implemented to model failure within the skin.

![Finite element model of the stringer element indentation test.](image)

The D-shape rubber bumper model was imported from the previous section into the current model to apply the pressure load. The modeling definition of the bumper was detailed in the previous section. Similarly, contact between the model's skin and the bumper, and potential contact between the skin and stringer was accounted for "hard," penalty contact interaction. The Lagrangian contact interaction was assigned to the bumper's self-contact during its collapse. Contacts involving the rubber bumper were
given a friction coefficient of 0.5, while contacts between the composite shells were defined as frictionless. Symmetry boundary conditions were applied at the symmetry plane of the model at both the bumper and the stringer element. The potted region of the stringer was fixed in displacement. A constant, downward velocity of 0.5 m/sec was applied to the top of the bumper throughout the simulation. A fine mesh size of 1.27 mm was applied to the critical region of the skin where it joined with the stringer. A coarse mesh size of 5 mm was applied elsewhere in the model.

The force vs. actuator displacement curve of the test and FE model is shown in Figure 3.54. A nearly one-to-one matchup was found between these curves in the elastic region. In the experiment, failure initiation was found at 21.55 and 20.33 kN, whereas the FE model failed at 19.04 kN. However, the predicted failure load was still accurate within 10% of the experimental data. At failure, the stringer element model's skin bent and the model showed a large load drop to 16.3 kN (see Figure 3.54 and 3.55). While the delamination failure was not captured in the model due to the lack of cohesive zone modeling, the in-plane ply failures of the FE model matched well with the failures found in the experiments. The model results were interrogated on a ply-by-ply basis to determine the level of failure in each ply. The Hashin-Rotem damage variables for fiber tension, fiber compression, matrix tension, and matrix compression were examined. Figure 3.36 shows the color contour plot for ply damage on the model's skin surface at the time of the load drop. The fiber tension failure on the top ply of the skin was found (as indicated by the grey area) on the skin at the location where it was found in the experiments. The investigation revealed bending induced failures of the top and bottom
six plies in the model. The extent of the model-predicted damage was between plies failures of SE01 (2 to 3 external plies per surface) and SE02 (through-thickness skin crack). Thus, the model results were in good agreement.

Figure 3.54. Load vs. displacement plot of the stringer element experimental results and FE Model.

Figure 3.55. Stringer element model's deformed shape at failure.
3.7 Chapter Conclusions

Analysis and simulation of the large-scale panels were broken down into simpler, small-size parts, and the individual failures were examined in small scale setups that replicated those failure modes observed in the large-scale tests. The structural elements were taken from the fuselage panels to create coupons and element specimens. The loading and boundary conditions for the small-scale experiments were defined in a manner that replicated conditions of the structural element from the impact experiment. The stresses states in these specimens leading up to failure were explored. Furthermore, the elements and coupon level experiments supported finite element model development of a FEA modeling methodology that accurately captures the key physics and failure modes. Their simple geometry and boundary conditions helped to increase the efficiency.
of the model development. The resulting models showed very good correlation across all of the coupons and elements tested. The modeling definitions were next transferred to the large-scale panel models.
Chapter 4. Modeling of Large-Scale Experiments

This chapter addresses the final stage of the analysis approach outlined in Figure 1.2, the numerical analysis of the dynamically impacted fuselage panels based on models developed at small-scale level. Nonlinear finite element models were created for the large-scale Stringer05 and Frame03/04 specimens. One of the major obstacles in modeling cumulative small-scale damaging events occurring in such large specimens was the high computational cost required to run the models. Therefore, the choices for analysis methods were limited. First of all, modeling nonlinear events with implicit integration scheme was very costly due to force equilibrium requirement at the end of each integration step. Balancing the internal force in the structure and the externally applied load could require many additional small times steps. This was especially true for large models that used cohesive surface interactions extensively, such as in the multi-layered shear tie models. In this case, the simulation could stall because of the large number of interactions (internal forces) that had to be solved for and balanced against the load. Furthermore, convergence issues would arise when delamination or fiber failure was triggered because of the change in stiffness. Stiffness loss often resulted in instability of the model, and prevented the completion of the simulation. Hence, the explicit integration scheme was used exclusively for the large-scale models as it did not enforce force equilibrium.

Secondly, the quasi-static experiments had long durations and thus, from modeling perspective, overly time-consuming to simulate. Although the implicit scheme
could be useful for modeling quasi-static processes, it was not viable for failure analysis. On the other hand, the explicit integration scheme was designed for short, dynamic events, with durations in the order of a few seconds or less. This was because the model's complexity controlled its time step size for a simulation. To simulate a panel with a large number of failure events, the explicit integration scheme would require extremely small time steps to avoid errors and to stabilize the solution. Therefore, the number of time steps and the simulation time increased drastically compared to simple, linear elastic models. Explicit simulation was only possible when the impact event occurred over a short duration, such as in the dynamic impact experiments. The quasi-static indentation experiments were conducted at a rate of approximately 6 mm/min, and the duration of such experiments could last up to several minutes of actual time. These were very long events to model, and the simulation would require several months of physical time to complete. For these reasons, nonlinear models were developed only for the dynamically impacted specimens ran at 0.5 m/sec, where the experiments completed in less than 0.3 second.

4.1 Modeling of the Stringer05 Experiment

A finite element model of the Stringer05 dynamically impacted panel test was developed in Abaqus/Explicit. This panel was reinforced with two stringers and impacted on the skin between the stringers. A cross-section view of the model is shown in Figure 4.1, while a side view of the model is shown in Figure 4.2. The dimensions of
the model were based on the specimen's drawings. The panel model was a half model, due to the quasi-isotropic laminate layup and the geometrical symmetry. The meshes in Figure 4.1 and 4.2 are color coded for each component in the model. The half model of the panel consisted of the external skin with a reinforcing stringer (green mesh), as well as the four supporting shear ties (blue mesh). A half model of the D-shaped rubber bumper (grey mesh) and its supporting steel plate (orange mesh) form the impactor. Displacement tie constraint was used to connect the bumper and the plate. Symmetry boundary conditions were applied at the plane of symmetry of the model. Fixed boundary conditions were applied at the shear tie flanges, where they were bolted to the steel fixture. A constant velocity of 0.5 m/sec was applied to the back of the steel plate to generate the impact load on the panel. Its total displacement was 113 mm (with an initial 2.5 mm gap), representing the experimental conditions.

Figure 4.1. Cross-section view of the Stringer05 half model.
As an additional boundary condition, the steel brackets bolted to the shear ties were also modeled as vertical steel plates. The close up view of the shear tie in Figure 4.2 shows that vertical steel plates were placed behind the shear ties. This was done in order to drive the opening rotation of the shear ties and the lateral movement of the panel during impact, as described in the previous chapter (see Figure 3.11). On the other hand, the bolt fasteners used to connect the shear ties to the panel skin were not included in the model. Instead, displacement tie constraints were applied to the contacting surfaces between the shear ties and the skin. This was a valid simplification because the bolts did not experience failure or induce failure in the composite. Contact between the bottom
the bumper and the panel was achieved by assigning a hard, penalty contact interaction to these surfaces. On the other hand, the Lagrangian contact interaction was assigned to the bumpers self-contact during its collapse, and the composite panel's self-contact during impact. A friction coefficient of 0.5 was assigned to contacts involving the rubber bumper, whereas contacts that did not involve the rubber bumper were approximated as frictionless.

The 8-node continuum solid elements were used to model the steel fixtures (C3D8R) and the bumper (C3D8I). The steel fixtures were given elastic material properties, while the hyperelastic Ogden material model was used to describe the D-shaped bumper, as shown in the previous chapter. Mesh sizes of 12.5 and 5 mm were assigned to the steel fixtures and the bumper, respectively. The 8-node continuum shell (SC8R) elements were used for the carbon fiber panel. The Abaqus built-in composite layup was used to define the sectional properties of the skin, stringers, shims, and shear ties. A global mesh size of 10 mm was assigned to the panel; however, the mesh was refined at locations of potential failure. These locations were the skin-stringer joint (with a 3 mm mesh) and the shear tie's curved corner (with a 1.5 mm mesh). To model the in-plane ply failure and softening behaviors seen in the test, the Hashin-Rotem failure criterion has been implemented to all of the composite sections to simulate in-plane ply failure, and cohesive surface interactions have been implemented to simulate delaminations (skin-stinger interface and in shear tie radius region). The carbon/epoxy properties and the cohesive properties used in this model were summarized in the previous chapter.
The Stringer05 impact experiment resulted in both skin-stringer delamination and delamination within the shear ties' curved corners. These failures modes were simulated with cohesive surface interactions. Figure 4.3 and 4.4 show the cohesive zone interface locations in the skin-stringer joint and the shear tie's curved corner, respectively. As shown in Figure 4.3, details of the skin-stringer step-joint were included in the model by modeling the changes in the stringer's thickness at the joint location. This was necessary in order to predict the panel’s stiffness since the stringer flange was within the impact zone and a part of the primary load path. Earlier models created with a non-stepped joint configuration overestimate the panel’s stiffness. Also, the step joint affects the transfer of shear stresses between the skin and stringer flange, so having the correct geometry was critical to predicting the stress state in the skin-stringer connection. The skin and the stringer parts had separate meshes to allow for the use of cohesive surface interactions between their contacting surfaces. The shear ties were modeled with four layers of elements in the thickness direction, as in Figure 4.4. Cohesive surface interactions were then applied between the layers to simulate shear tie delamination caused by radial tension stress from the opening moment. As previously discussed in the test results section, radial delamination of the shear ties have been observed to reduce the shear tie rotational stiffness and the panel's global stiffness. The shear ties were separated into only four individual layers in the model, as opposed to six individual plies, to lower computation cost.
Figure 4.3. Cross-section view of the skin-stringer joint showing the cohesive zone interface area between the skin and stringer (red line).

Figure 4.4. Side view of the 4-layer shear tie showing the cohesive zone interface area between the element layers.
The deformation plots of the half model before loading and at maximum bumper displacement are shown in Figure 4.5. As the transverse load was applied, the panel skin was subjected to a biaxial bending. The deformed plot shown in Figure 4.5b correctly predicted the mode shape of the panel. Figure 4.6 shows a side view of the panel at maximum bumper displacement, with the skin and stringer bending to absorb the impact load. The moments from the skin were transferred to the shear ties, causing opening rotation of the shear ties, as well as lateral (sideways) deflection of the panel.

Figure 4.5. Stringer05 model: (a) unloaded, and (b) at peak bumper displacement. The steel plate above the bumper was omitted from the view.
A comparison of the load vs. skin indentation plots between the experimental data and the FEA model is shown in Figure 4.7. The figure shows that there is good matchup between finite element model and the test panel in terms of their responses up to 36.7 mm displacement corresponding to the failure load. The FEA model’s peak load was 68.6 kN, which was 2.3% higher than the experimental peak load at 67.0 kN. During the Striner05 experiment, local skin-stringer delamination, stringer and skin fiber failures, and shear tie radius delaminations were observed. The FE model captured each of these failure events with varying degrees of success. The model prediction for each failure event along the force plot is indicated in Figure 4.7 with a letter annotation. Crack formation was determined in the model by examining the Hashin-Rotem ply damage parameters for fiber tension (HSNFTCRT), fiber compression (HSNFCCRT), matrix tension (HSNMTCRT), and matrix compression (HSNMCCRT) failures at each composite laminate ply through the element thickness. When all of the integration points through the thickness of an element have experienced failure in one of the criteria, then the element would be deleted. Likewise, delamination in the model was determined by
examining the cohesive surface damage parameter (CSDMG) on the relevant joined surfaces.

Figure 4.7. Comparison of Stringer05 experimental data and FE model result.

Figures 4.8 to 4.11 show the occurrence of failure events in the model with the color contours of the damage parameter values described above. A damage parameter value of 1 (grey color in the contour) would indicated failure in its respective mode. These events correspond to the loads indicated in Figure 4.7. The sequence and indentation depth corresponding to each of these events in the simulation were similar to those found in the experiment. For comparison, the damage states of the Stringer05 panel after impact are also included with these figures. Figure 4.8a shows the radial delamination damage at an internal cohesive interface of the half shear tie shown at the symmetry plane in Figure 4.1. The damage occurred at 31 kN of impact load. It initiated
at the edge of the shear tie due to the proximity to the stringer (i.e., the primary load path). This was similar to the damage shown in Figure 4.8b from a shear tie in the experiment. Although the actual delamination is no visible in Figure 4.8b, the surface distortion at the shear tie corner was a sign of internal ply separation. Pinpointing when shear tie radial delamination occurred during the Stringer05 dynamic test was difficult. However, it was observed from Stringer02 quasi-static testing that shear tie failures occurred at approximately 35 kN. Thus, the predicted delamination damage was in the correct range of loading.

Figure 4.8. Stringer05 (a): cohesive surface damage in the shear tie, and (b): the exterior view of the same shear tie in the experiment showing surface undulations (circled). The symmetry plane coincides with the red line.
The model-predicted skin and stringer cracks are shown in Figures 4.9a and 4.10a, respectively. The rubber bumper was omitted from these figures. In the figures, the color contour displayed the external ply failure by tension in the skin, and by buckling induced compression in the stringer. In the simulation, this damage occurred at the peak contact force of 68.6 kN (see Figure 4.3). The cracks were formed beneath the bumper, due to the high localized bending stresses at geometrical discontinuity at the skin and stringer joint. For comparison, the locations of the skin and stringer cracks from the experiment are shown in Figures 4.5b and 4.6b, respectively. The model successfully predicted the location of these cracks.

Figure 4.9. Stringer05 (a): Hashin ply failure by fiber tension at the skin's external surface, and (b): the exterior view of panel skin with the though-thickness skin crack circled. The symmetry plane coincides with the red line.
Finally, the model-predicted skin-stringer delamination damage is shown in Figure 4.11a. The color contour here is a plot of the cohesive surface damage parameter on skin of the specimen. Similar to the physical experiment, the skin-stringer delamination damage initiated underneath the bumper and grew towards the shear tie region. As shown in Figure 4.11b, the size of the predicted delamination zone was nearly an exact match to the experimental result. In the simulation, the skin-to-stringer delamination was found to develop during the load drop, as opposed to occurring at the peak load. Also, there is discrepancy in the predicted degree of skin crack formation. At the end of the loading, only the outermost plies have failed in the model, whereas a
through-thickness crack was found on the panel skin in the experiment. These discrepancies can be attributed to the fact that the formation of skin cracks, stringer cracks, and delamination were all competing failure modes, reflecting different ways in which impact energy can be absorbed by the panel. The extent of the skin crack was affected by the delamination failure. Since the delamination damage was delayed in the model, the development of skin cracks was thus also different.

Figure 4.11. Stringer05 (a) cohesive surface damage in the skin-stringer joint, and (b) the exterior view of panel skin with the skin-stringer delamination (white hatched zone). The symmetry plane coincides with the red line.
4.2 Modeling of the Frame03 Experiment

The Frame03 and Frame04-1 experiments represented impact scenarios closest to ground service vehicle collision against a fuselage section. These were larger fuselage panel specimen with additional reinforcing C-frames. The impact responses of these panels were more complex due to the increase in the number of elements. Of critical importance were the failures in the shear ties and their interactions with the C-frames, which drove the subsequent failures. The analysis work presented in this thesis was built upon a pre-existing baseline model developed by DeFrancisci [DeFrancisci 2013], as the continuation of the composite aircraft blunt impact project. An internal view of this baseline model is shown in Figure 4.12.

The baseline model was a half-symmetry model due to the geometric and material symmetry of the panel, half-way along the length of the C-Frames (see Figure 4.13). The Frame03 half model consisted of the skin with reinforcing two stringers (green mesh), fifteen shear ties (blue mesh), and five frames (yellow mesh). The rubber impactor (grey mesh) is shown on the external side of the panel. The middle three frames in the impact zone were composite C-frames, while the outer frames outside of the impact zone were aluminum frames. The panel specimen was modeled using continuum shell elements (SC8R) for all of its carbon fiber parts, including the skin, stringers, shear ties, and C-frames. Composite layup definition was used for the carbon fiber, and only a single layer of continuum shell elements was created in the through-thickness direction. The delamination damage was not simulated with this baseline model. However, the Hashin-
Rotem failure criteria were used to model in-plane ply failures in the shear ties and the C-frames. Due to the size of the panel, a global mesh of 19 mm was assigned to the panel, with a locally refined mesh of 6 mm being applied to the shear ties. The cylindrical rubber bumper impactor was idealized as a rectangular rubber pad in the model. This was because the cylindrical bumper had a uniform wall thickness of 25.4 mm which collapsed at relatively low loading level into a configuration that could be reasonably approximately as a 50.8 mm-thick rectangular pad. Solid elements (C3D8I) were used to model the bumper with a purely elastic material definition.

Figure 4.12. Frame03/04 baseline FE model.

Figure 4.13 shows the cross-section view of the half model with its boundary conditions, which were chosen to reflect the experimental setup. Symmetry boundary condition was applied at the symmetry plane of the model. The ends of the C-frames were reinforced with stiffness material and section properties to represent the steel
fixtures clamped to the C-frames. The C-frames were simply supported at the ends, with added rotational springs to resist the bending deformation of the panel from the impact loading. The Abaqus spring elements were used to attach the C-frame's ends to a fixed plate with a spring stiffness of $1.13 \times 10^6$ N-m/rad. Twisting deformation of the C-frames at the boundary, however, was prevented due to the attached steel blocks that constrained this degree of freedom. A dynamic impact load was applied to the backside of the rubber pad at a rate of 0.5 m/sec. Since the bumper model was to assumed to be collapsed before the simulation, the total displacement bumper displacement was 78.7 mm

In terms of interactions, displacement tie constrains were used to connect the skin structure, shear ties, and C-frames, instead of bolt connections. To account for panel self-contact during crushing deformation, normal and tangential contact properties were defined between the composite parts. Hard contact with Lagrangian contact constrains
was used in the normal direction, and frictionless contact was used in the tangential direction. For the bumper to skin structure contact, hard normal contact with the penalty contact constraint and a 0.5 coefficient friction was assigned to the tangential direction.

Figure 4.14 shows the contact force history of the Frame03 and Frame04-1 specimens in comparison to the FE baseline model [DeFrancisci 2013]. The Frame03 and Frame04-1 results were combined in this plot. The initial load up response of the experiments was correctly predicted by the baseline model up to 10 mm skin displacement (indentation). At this displacement, shear tie corner delamination initiated in the experiment, followed by fiber fracture due to crushing of the shear ties. These failures produced the initial load drops in the experiments. However, the baseline model not successful in predicting these failures due to the lack of delamination representation via cohesive zone modeling and the lack of transverse shear failure criteria. As a result, the delamination failures in the shear tie's curved corners were not captured. Instead, the initial shear tie failure mode was buckling at the vertical flange. Figure 4.15 shows this failure mode in the baseline model. The shear tie flange fracture in post-buckling were delayed until a much higher contact force was developed because the Hashin-Rotem in-plane failure criteria could only account for bending stress induced failures, while neglecting the transverse shear stresses. Subsequent failures such as bending failures at the fastener lines were also delayed. The initial shear tie delamination failure modes need to be predicted to accurately capture the subsequent responses.
Figure 4.14. Load history of the combined Frame03 and Frame04-1 tests and baseline FEA Model.

Figure 4.15. Close-up view of Frame03 shear tie buckling failure as predicted by the baseline model.
In order to improve the model predictions, the multilayered shear tie modeling approaches were incorporated into this baseline model. The improved shear tie models were imported directly from the shear tie compression coupon model described in Chapter 3. The updated half-symmetric model of Frame03 specimen is shown in Figure 4.16, with six new shear ties models applied in the impact zone and adjacent to the impact zone (densely-meshed shear ties). Except for the shear ties, all the composite parts were modeled using continuum shell elements (SC8R) with composite layup definition. Meanwhile, the impactor was modeled using the continuum solid elements (C3D8I) with hyperelastic rubber material definition. The updated shear ties were multilayer models and each consisted of twelve layers of elements through the thickness. Both the SC8R and C3D8R elements were used for the shear ties in two separate simulations. Their results are presented below.

Figure 4.16. The updated Frame03/04 model with the six center shear ties replaced by refined models.
As discussed in the previous chapter, modeling fiber crushing at the shear ties during impact presented a major challenge. Excessive element distortions at the shear tie corner often led to numerical instability issues. Simulation was especially difficult for a large panel model with multiple shear ties that simultaneously experienced varying degrees of crushing damage. Therefore, different elements and material failure models were used in an attempt to simulate this behavior. Both the continuum shell (SC8R) and the continuum solid (C3D8) elements were used in the shear ties models to determine the best approach.

The SC8R element performed better in the bending, and better predicted the initial delamination damage. Figure 4.17 shows the contact force vs. indentation plot for the Frame03 model with continuum shell shear ties. The experimental result is also shown in this figure. There were good predictions for the initial panel stiffness, shear tie delamination load, load drop at crushing failure, and crushing of the curved corner. However, with fiber crushing damage initiation, the shell elements at the shear tie corner became highly distorted, as shown in Figure 4.18. The distorted element shape became unacceptable for the simulation to continue, and fracturing of the shear ties was not achieved in this model. As a result, the simulation crashed before the time of the shear tie fracture failure in the impact zone.
Figure 4.17. Contact force vs. indentation plot of the Frame03 test and FE model with shell element shear ties.

Figure 4.18. Close-up view of the impact shear tie in the FrameXX model. Shear tie crushing damage modeled with continuum shell elements, before the simulation crashed.
To alleviate the element instability problem, the simulation was modified with the continuum solid C3D8R element shear tie model and the number of element layers was increased to twelve. Element section controls were also enabled. The spurious deformation modes in each solid element were prevented by the added penalty stiffness. Although the delamination load prediction was slightly overestimated due to the element stiffening, the solid element was much more stable, and allowed the simulation to run to completion. Figure 4.19 shows the load vs. indentation plot of the Frame03 test compared with the updated solid element FE model results. The solid element shear tie crushing failure is shown in Figure 4.20. Compared to the continuum shell element model in Figure 4.18, the solid elements remained rectangular throughout the simulation. Hence, the shear tie fracture failure was successfully simulated.
The subsequent shear tie fracture failure modes were also captured with the solid element shear tie model. Figure 4.21 and 4.22 show the key panel failures at various time after impact from the experimental results and modeling results, respectively. In the updated Frame03 model, the impacted shear ties behaved similarly as the shear tie compression simulation, as shown in Figure 3.34. Delamination of the impacted shear ties occurred at 65 kN (see point a in Figure 10). Then, crushing of the impacted shear ties at the curved corners initiated at 79.5 kN (see point b) to cause an initial load drop, which occurred at a much higher load than in the experiment. Between 18.6 and 28.0 mm of displacement, the shear tie corner crushing failure occurred (between points b and c), producing some oscillations in the contact force. After the shear tie corners were crushed, the shear flange acted as a plate in direct compression, thus allowing the load to
build up again. At point c in Figure 10, the center shear tie flange fractured at the corner or at halfway between the corner and bolt line. Figures 4.21a and 4.22a show the model and experimental shear tie fracture failure in the impact zone. This corresponded to the second load drop at 91.60 kN in the experiment, and 89.7 kN in the model. The load path was then redirected to the shear ties adjacent to the impact zone, as well as via direct contact between the stringer and C-frame, as shown in Figures 4.21b and 4.22b. C-frame twisting was driven by the contact force being applied offset from the C-shape section's shear center, so large C-frame rotation caused the shear ties adjacent to the impact zone to fail in bending. Figures 4.21c and 4.22c show the shear ties adjacent to the impact zone fracture with large C-frame rotation. As shown in Figure 4.19, this model was mostly accurate for up to 70 mm of indentation, at which time the C-frames were predicted to fail near the outer boundary conditions.

Despite the overall good correlation between the model and the experiments shown in Figure 4.17, some details such as the panel skin indentation level and failure loads did not match with the experiments. After the initial shear tie delamination, shear tie fracture failure (point occurred in at lower skin indentation and higher load compared with the experiments. This could be due to the fact that the solid element shear ties were stiffened by the element section controls, so the contact forces built up more quickly for all the subsequent failures compared to the experiment.
Figure 4.21. Updated Frame03 half model: (a) impacted shear tie corner fracture damage, (b) c-frame twisting, and (c) adjacent shear tie fracture at the corner.
Figure 4.22. Frame03 experiment: (a) impacted shear tie corner fracture damage, (b) C-frame twisting, and (c) adjacent shear tie bending failure at the bolt line.

Figure 4.23 shows the cross-section views of the Frame03/04 model as the bumper displacement increased. These events correspond to those annotated in Figure 4.12. As described in the experimental results in Chapter 2, twisting of the C-frames in
the impact zone was tied to the degree of damage in the shear ties. As the shear ties experience damage, the connection between the skin and the stiffening C-frame weakened. The C-frames, which were loaded away from their shear centers, were allowed to twist more each time after the impacted shear ties delaminate (Figure 4.13a), crush (Figure 4.13b), and fracture (Figure 4.14c). Once fractured, the C-frames came into contact with the stringers, which drove further twisting of the C-fractures. Large C-frame deformation eventually lead to damage in the shear ties adjacent to the impact zone and C-frame fracture failures (Figure 4.14d).

Figure 4.23. Side view of the Frame03/04 model showing increasing degrees of C-frames twisting during: (a) impacted shear tie corner delamination, (b) impacted shear tie crushing, (c) impact center shear tie fractures, (d) adjacent center shear tie and C-frame fractures.
Figure 4.24a shows the final damage state of the model along a C-frame. The impacted shear tie directly beneath the bumper suffered fracturing along its curved corner. This damage state was shared by the shear tie adjacent to the impact zone (not directly loaded). The C-frame also suffered crack damage next to the undamaged shear tie close to the reinforced boundary condition. This damage state was similar to the damage seen on along some of the C-frames (Figure 4.24b).

Figure 4.24. (a) A cross-section view of the Frame03/04 model summarizing all the predicted damage along a C-frame, and (b) a photograph of the Frame03 specimen that shows similar failures along a C-frame.
4.4 Chapter Conclusions

In the previous chapter, element-level FE models were then created to accurately predict these experiments which help to validate choices made in defining the models and demonstrate the capability to predict these failure modes. In this chapter, the validated element-level finite element models were used to build the large-scale panel impact models. These physics-based models could predict the key failures events with a small amount of error. The stringer elements were able to predict skin cracking initiation within a 10% load error. In the case of the shear tie compression coupons, the results were in accurate agreement with the experiments after refining the mesh through the thickness of the shear tie so that there were twelve layers of elements through the thickness (ply-by-ply modeling). The modeling methodologies were then directly transferred to simulating large-scale StringerXX and FrameXX experiments.

The Stringer05 model gave accurate prediction of the failure modes and failure loads. Also, by updating the shear tie model in the FrameXX models, the progression of damage following failure of the shear ties was better captured. These results could be subsequently applied to investigating various new structural configurations that have not been experimentally tested. However, future improvements could still be made. For example, stress singularities issues could be better addressed during the element failure process in order to overcome the force oscillation created when ply failures occur. Also, the anisotropic Hill failure criterion was not perfect and could not be uniformly applied to
all composite structures. It was suitable for modeling the shear tie failures after some assumptions were made because of the material's anisotropy. Furthermore, failure parameters such as the fiber shear failure strength and fracture energy were approximated.

Chapter 4, in part, is a reprint of the materials appearing in Experimental and Modeling Investigation of Blunt Impact to Stringer-Reinforced Composite Panels 2013, Chen, Zhi Ming; Kim, Hyonny; Defrancisci, Gabriela, Destech Publications 2013. The dissertation author was the primary investigator and author of this paper.
Chapter 5. Conclusions

A series of large scale impact blunt experiments were conducted on carbon fiber composite fuselage panel specimens to assess the damage tolerance and detectability of such structures in GSE collision events. The specimen material, geometry, and layup were similar to the new generation of composite commercial aircrafts. Impact testing involved constraining the panels at the shear ties or at the ends of the C-frames, and applying an out-of-plane load via rubber bumpers. The damage sustained included delaminations at the shear ties, shear tie crushing and fracturing, stringer-to-skin delamination, skin cracking, and C-frame fractures. Based on loading cases conducted, GSE collisions with higher than 64 kN of force would always produce significant damage to the internal components, such as stringers and C-frames. These severe damage modes could drastically reduce the load carrying capability of the fuselage, and require immediate repair. However, depending on the impact locations, speed, impactor type, the damage may not be readily visible from the exterior of the fuselage. The external fuselage skin is very flexible, and would rebound and hide the damage incurred by the internal components. Thus, it was imperative to gain full understanding of the damage process to aid the development of preventive strategies and damage detection measures.

An inverted pyramid approach was taken during the analytical methodology development process. Damage modes sustained by the fuselage panel specimens were isolated and studied individually via focused small-scale experiments. These small-scale
coupon experiments aimed to recreate the loading and failures of the individual fuselage components during the large-scale impact experiments. Examining the coupon failures allowed for in-depth stress analysis and facilitated the efficient building of finite element models. The finite element analyses were conducted from the small to large-scale, in a reverse order compared to the experiments. Due to their simplified loading conditions and small size, coupon level experiments were possible to simulate without the complications of having multiple failure modes occurring in a full-panel simulation. This opened up the way to fine tune the modeling methodology. Realistic material property data were used in the simulations to ensure that the models could be universally applicable to larger scale models. The coupon model successfully captured the force-deformation behavior, small-scale failure initiation, and post-failure behavior in a near exact manner. At which point the modeling parameters and methodologies were transferred to the large scale model. The end result was comprehensive fuselage panel models capable of predicting damage initiation, growth, and post-failure behavior of the experimental panels.

Large-Scale Fuselage Impact Experiment Conclusions

Large-scale fuselage panel failure modes were dependent on the impact location, speed, impactor type. For the StringerXX specimen, these factors influenced the amount of local versus global deformations. Localized failures include skin and stringer cracking at the impact zone, as well local skin-stringer delamination. Global failures include shear
tie delamination, crushing, and widespread skin-to-stringer delamination. Impacts conducted against stiffener locations, at dynamic speed, or with a rigid impactor all contributed to localized panel deformation responses. Conversely, impacts conducted against non-stiffened skin locations, at quasi-static speeds, or with a soft impactor contributed to more global panel response.

For the FrameXX specimens that were impacted with long cylindrical bumpers. It was found that varying the impact speed altered the response of the C-frames. Impacting at quasi-static speeds allowed contact locking to develop between the C-frames and stringers that prevented C-frame rotation, thereby allowing high contact interaction stresses to build up causing stringer or C-frame penetration damage. Whereas impacts conducted at dynamic speeds had continuous sliding contact between the stringers and C-frames and did not allow concentrated local stress buildup. The C-frames experienced large rotation and caused wide-spread failures in the adjacent shear ties and the C-frames themselves.

**Small-Scale Experiment Conclusions**

The small-scale experiments paved the way to in-depth failure analysis and finite element models. The coupon experiments replicated the loads and boundary conditions experienced by fuselage elements during impact. Thus, the deformation and failure modes of the fuselage elements were reproduced. These data allowed for focused
mathematical and numerical analysis. With realistic material parameters applied consistently to all of the small scale models, the models were shown to achieve very high accuracy with respect to load, deformation, and failure responses observed during the experiments.

**Finite Element Modeling Conclusions**

Large scale composite fuselage panels were modeled with continuum shell elements to increase computational efficiency. Shell elements were generally very robust for modeling bending behavior. They were used in conjunction with the composite material layup manager to define the laminate composition. The composite could be modeled with a single element layer, or multiple element layers though the thickness of the composite depending on whether delamination was a relevant failure mode at the specific component. Also, continuum shell elements could approximate transverse shear responses; therefore they worked well with cohesive elements to predict delamination failures. However, special attention was required when the elements were subjected to compression loading such as at the shear tie corner and rubber bumper elements. In these cases, the compressive stress could readily lead to element distortion errors and early determination of the model simulation. The continuum solid elements worked better under compression as they supported enhanced section control.
Simplifications were made to the boundary conditions and bolt connections to reduce simulation time. Displacement and rotational boundary conditions were applied directly to the composite parts in lieu of modeling the metal fixtures and fixture interactions. Furthermore, some of the bolt connections were simplified to tie interactions between the contacting surfaces of the bolted parts because they were not critical failure locations. A comprehensive treatment of bolt connection would be required whenever bolt failures occurs, such as fracturing of the shear ties at the bolt lines. The resulting models were highly accurate, and only deviate from the experimental results at the post-failure regime.

**Benefits to Aviation**

Many aspects of this project could be beneficial to the safety of aviation. First of all, the experimental results could be used for educating and training GSE operators and aircraft pilots. The widespread internal damage seen in the FrameXX specimens could contribute to aircraft failure during failure and would require immediate attention. However, the general lack of external damage visibility caused by high energy blunt impacts severely hinders the damage detectability during pre-flight visual inspection. Thus, the experimental results will help to generate awareness for the blunt impact damage, so aircraft designs can be improved for these damage states. Also this research will help to reinforce proper education of aircraft and GSE operators, and promote thorough inspection of the aircraft at locations away from the impact site should GSE collisions occur. Secondly, the FE model results help predict the most probable locations
of damage after a collision, and improve active and passive damage detection systems. Identifying the critical cases that can generate large internal damage without external visibility can also improve GSE operation procedures. The ability to simulate impact events with FE models could help to improve future fuselage designs to prevent catastrophic damage, or to encourage externally visible damage.

**Future Works**

A major issue found with the current composite fuselage design is that impact induced internal failures may not leave externally visible signs. As a future extension of this work, finite element models could be used optimize fuselage design either to strengthen the fuselage against high energy blunt impacts, or to improve damage visibility. Statistical data of ground service vehicle collision could be incorporated to determine the most likely impact location and energy levels. Altering the geometry of the fuselage assembly or the individual components could help to redirect the impact load paths and encourage external failure modes, while discouraging visible, internal failures modes.
References


Appendix. Additional Data from Large-Scale Fuselage Impact Experiments

This appendix presents additional data for the large-scale fuselage impact experiments. Specifically, the test data from Stringer03 (quasi-static) and 06 (dynamic) experiments are presented below. Both panels were had three stringers and were loaded directly on the middle stringer. Stringer03 was loaded five times until extensive stringer-to-skin delamination failure occurred. Figure A.1 shows the contact force vs. actuator displacement plots for Stringer03. During the 1st loading, the panel was loaded to 39.78 kN, and two surface level cracks were found on the panel skin under the indented area. Figure A.2 shows that this damage mode was visible only during close-up examination under glancing light source, similar to those found in the Stringer Element compression experiments. Shear tie corner cracks were also found in all four shear ties adjacent to the indented stringer. During the 2nd and 3rd loadings, bending moment at the indentation location caused cracks to form at the stringer’s curved corner between the stringer flanges and diagonal walls. Figure A.3 shows the stringer crack at its curved corner. These failures hindered the moment and shear force transfer from the skin to the stringer. During the 4th loading, the panel was loaded up to 56.98 kN, ultrasound scan of the shear tie radius regions showed that all four of the main shear ties suffered crushing and radial delamination. Also, the stringer corner cracks have propagated along the length of the stringer. During 5th loading, the panel was loaded up to 61.64 kN. Extensive delamination of the loaded stringer near the shear ties locations occurred due to high interlaminar shear stresses. The exterior of the panel after final failure is shown in Figure
A.4. The white hatched zone indicates the area of skin-stringer delamination. Many surface level skin cracks were found, similar to the one shown in Figure A.2. However, no through-thickness skin cracks were observed. This level of external damage would be very difficult to detect visually. Crushing and radial delaminations were found in all shear ties attached to the panel.

Figure A.1. Stringer03 contact force vs. actuator displacement plots.

Figure A.2. Barely visible skin cracks (between the two white lines).
Test specimen Stringer06 was tested dynamically at 0.5 m/sec using an MTS 220 Kip actuator. The actuator displaced 4.25 inches into the panel. The Stringer06 load vs. actuator displacement plot is shown in Figure A.5, along with the Stringer03 data. The figure shows that the dynamic loading had minimal effect on the initial stiffness of the panel. However, major failures initiated at different load levels (48.5 kN for Stringer03,
and 57.4 kN for Stringer06). It was found post-test that similar damage modes were generated compared to Stringer03. In addition, localized skin cracks were observed in the Stringer06 dynamic test, as shown in Figure A.6.

Figure A.5. Stringer03 and 06 contact force vs. actuator displacement plots.

Figure A.6. Through-thickness skin cracks of the Stringer06 specimen.
The fuselage panel loading details, max load, and failure modes were summarized in Tables A1 to A3.

**Table A.1. StringerXX panels result summary.**

<table>
<thead>
<tr>
<th>Specimen ID</th>
<th>Loading Details</th>
<th>Max Load (kN)</th>
<th>Failure Modes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stringer00</td>
<td>Aluminum on Stringer <em>Quasi-static</em></td>
<td>30.7</td>
<td>Localized Penetration; Highly Visible</td>
</tr>
<tr>
<td>Stringer01</td>
<td>Aluminum on Skin Between Stringers <em>Quasi-static</em></td>
<td>26.7</td>
<td>Localized Penetration; Highly Visible</td>
</tr>
<tr>
<td>Stringer02</td>
<td>Bumper on Skin Between Stringers <em>Quasi-static</em></td>
<td>61.7</td>
<td>Skin-Stringer Delam., Shear Tie Damage, No Skin Cracks; Non-Visible</td>
</tr>
<tr>
<td>Stringer03</td>
<td>Bumper on Stringer <em>Quasi-static</em></td>
<td>61.6</td>
<td>Stringer Cracking; Skin-Stringer Delam., Shear Tie Damage; Surface-Level Skin Cracks</td>
</tr>
<tr>
<td>Stringer04</td>
<td>Bumper Centered on Stringer Flange <em>Quasi-static</em></td>
<td>78.2</td>
<td>Stringer Cracking; Skin-Stringer Delam., Shear Tie Damage; Skin Cracks</td>
</tr>
<tr>
<td>Stringer05</td>
<td>Bumper on Skin Between Stringers, <em>Dynamic</em></td>
<td>67.0</td>
<td>Stringer Cracking; Skin-Stringer Delam., Shear Tie Damage; Skin Cracks</td>
</tr>
<tr>
<td>Stringer06</td>
<td>Bumper at Stringer, <em>Dynamic</em></td>
<td>57.4</td>
<td>Stringer Cracking; Skin-Stringer Delam., Shear Tie Damage; Skin Cracks</td>
</tr>
</tbody>
</table>
Table A.2. FrameXX quasi-static, 3-frame panels result summary.

<table>
<thead>
<tr>
<th>Specimen ID</th>
<th>Loading Details</th>
<th>Max Load (kN)</th>
<th>Failure Modes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frame01</td>
<td>Bumper on Skin Between Stringers</td>
<td>57.4</td>
<td>Shear Ties Crush, Frame Crack, Stringer Sever &amp; Flange Delam.; Non-Visible</td>
</tr>
<tr>
<td>Frame02</td>
<td>Bumper at Stringer</td>
<td>71.0</td>
<td>Shear Ties Crush, Frame Crack, Stringer Sever &amp; Flange Delam.; Visible Skin Crack Due to free edge effect</td>
</tr>
</tbody>
</table>

Table A.3. FrameXX dynamic (0.5 m/sec), 5-frame panels result summary.

<table>
<thead>
<tr>
<th>Specimen ID</th>
<th>Loading Details</th>
<th>Max Load (kN)</th>
<th>Failure Modes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frame03 &amp; Frame04-1</td>
<td>Bumper on Skin Between Stringers</td>
<td>57.4</td>
<td>Shear Ties Crush and Fracture, Frame Crack; Non-Visible</td>
</tr>
<tr>
<td>Frame04-2</td>
<td>Bumper on Skin Between Stringers,</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td><em>Aluminum Shear Ties</em></td>
<td>126.5</td>
<td>Frame Shear Fracture; Non-Visible</td>
</tr>
</tbody>
</table>