Title
High Energy Wide Area Blunt Impact on Composite Aircraft Structures /

Permalink
https://escholarship.org/uc/item/7fq9827j

Author
DeFrancisci, Gabriela K.

Publication Date
2013

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

High Energy Wide Area Blunt Impact on Composite Aircraft Structures

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

in

Structural Engineering

by

Gabriela K. DeFrancisci

Committee in charge:

Professor Hyonny Kim, Chair
Professor Chiara Bisagni
Professor Francesco Lanza Di Scalea
Professor Vlado A. Lubarda
Professor Vitali F. Nesterenko

2013
The dissertation of Gabriela K. DeFrancisci is approved, and it is acceptable in quality and form for publication on microfilm and electronically:

__________________________________________

__________________________________________

__________________________________________

__________________________________________

__________________________________________

Chair

University of California, San Diego

2013
DEDICATION

My family: Sheilah, Christina, Steven and Laura.
# TABLE OF CONTENTS

Signature Page ................................................................. iii
Dedication ................................................................. iv
Table of Contents .............................................................. v
List of Figures ............................................................... viii
List of Tables ................................................................. xiv
List of Abbreviations ........................................................ xv
Acknowledgements .......................................................... xvi
Vita and Publications ....................................................... xvii
Abstract of the Dissertation ................................................ xx

1 Introduction ............................................................... 1
1.1 Motivation ............................................................... 1
1.2 Objectives ............................................................... 7
1.3 Approach ............................................................... 8
1.4 Novel Contributions .................................................... 9

2 Background .............................................................. 12
2.1 High Velocity Versus Low Velocity Impacts ......................... 12
2.2 Quasi-Static Equivalence ............................................ 14
2.3 Typical Low Velocity Experimental Setup ........................... 15
2.4 Impactor Geometry and Material .................................... 17
    2.4.1 Impactor Bluntness Effects ................................... 17
    2.4.2 Rubber Material Modeling .................................... 18
2.5 Panel Size and Scaling Effects ..................................... 19
2.6 Summary ............................................................... 20

3 Large Scale Experiments ................................................. 21
3.1 Overview ............................................................. 21
3.2 Blunt Impact Threat Characterization ................................ 22
3.3 Panel Description ..................................................... 30
    3.3.1 Composite Material ........................................... 35
    3.3.2 Manufacturing ............................................... 36
3.4 Stringer Panels ....................................................... 43
    3.4.1 Test Setup ..................................................... 46
LIST OF FIGURES

Figure 1.1.1: Aircraft Surrounded by GSE Between Flights ............... 2
Figure 1.1.2: Schematic Diagram of Design Load Levels Versus Categories of Damage Severity as Defined by the FAA [5] ................. 3
Figure 1.1.3: Patch Away from Door Indicating Accidental Damage ........ 5
Figure 1.1.4: Cylindrical Bumper Typically Found on GSE .................. 6
Figure 1.1.5: Monolithic Composite Structure .............................. 6
Figure 1.3.1: Building Block Approach to Understanding Damage to Monolithic Composite Structures .............................. 8
Figure 1.4.1: High Amount of Skin Deformation in Panel With a Soft Impactor 11

Figure 2.1.1: Comparison Between (a) Large Mass and (b) Small Mass Impact Response [8] ........................................... 13
Figure 2.2.1: Force-Deflection Histories for Concave Shell Specimen Impacted at 3.1 m/s and Tested Quasi-Statically [23] ............... 15
Figure 2.4.1: High Deformation of Rubber Material During Impact on an Aluminum Plate [31] ................................. 19

Figure 3.2.1: Patches Surrounding Door on Boeing 747 ...................... 24
Figure 3.2.2: Boeing 737 and Surrounding GSE Between Flights .......... 24
Figure 3.2.3: Velocity of a Belt Loader Approaching a Boeing 757 Aircraft 25
Figure 3.2.4: Difficult Approach Angle of Belt Loader .................... 26
Figure 3.2.5: Potential Scraping on Fairing as Bumper is Raised to Door Level 27
Figure 3.2.6: Rolling Staircase Contacting Aircraft at an Angle .......... 28
Figure 3.2.7: Cylindrical Bumper and “D” Shaped Bumper ............... 29
Figure 3.2.8: Cylindrical Bumper and “D” Shaped Bumper Used in Experiments 30
Figure 3.3.1: Composite Fuselage and Acreage Internal Structure [40, 41] 31
Figure 3.3.2: Typical StringerXX and FrameXX Test Panels ............. 32
Figure 3.3.3: Shear Tie and Frame Cross Section Dimensions (all units in mm) 33
Figure 3.3.4: Shear Tie Dimensions ......................................... 34
Figure 3.3.5: Stringer Dimensions (all units are in mm) .................. 34
Figure 3.3.6: Frame Side View with Material Orientation for Shear Ties and Frames .................................................. 34
Figure 3.3.7: 1.8 m Diameter Autoclave at San Diego Composites ....... 37
Figure 3.3.8: Manufacturing Cure Cycle .................................... 38
Figure 3.3.9: Stringer and Skin Material Orientation ....................... 38
Figure 3.3.10: Skin Outer Mold Line (OML) Tool ......................... 39
Figure 3.3.11: Stringer Inner Mold Line (IML) Tool ..................... 40
Figure 3.3.12: Corner Detail of Stringer-Skin Connection ................. 41
Figure 3.3.13: Frame (OML) Tool ......................................... 42
Figure 3.3.14: Shear Tie (OML) Tool ...................................... 43
Figure 3.4.1: Stringer Panel .................................................. 44
Figure 3.4.2: Stringer Impact Locations ............................................. 45
Figure 3.4.3: Rigid 76.2 mm Radius Indentor (left) and OEM Rubber Bumper
(right) ................................................................. 45
Figure 3.4.4: Panel Stringer02 Test Setup in SATEC Machine ................. 46
Figure 3.4.5: Stringer Test Fixtures (Indentation Location 2, Dimensions are
mm) ................................................................. 47
Figure 3.4.6: Stringer00 Force vs. Indentation Curves ......................... 48
Figure 3.4.7: Localized Deformation of Panel Caused by 76.2 mm Radius
Rigid Indentor During First Loading of Stringer00. ....................... 49
Figure 3.4.8: No Visible Permanent Deformation After First Loading of Stringer00
Despite Formation of Delamination .............................................. 49
Figure 3.4.9: Stringer00 Peanut Shaped Delamination After First Loading .... 50
Figure 3.4.10:Stringer00 After Final Failure Following Second Loading .... 51
Figure 3.4.11: Stringer00 After Final Failure Following Second Loading; Panel
Rebounded to Original Overall Shape Despite Penetration Damage ........ 51
Figure 3.4.12: Stringer01 Final Failure Following Fourth Loading, Penetration
of Indentor Through Skin ...................................................... 52
Figure 3.4.13: Stringer01 Force vs. Indentation Curves ......................... 53
Figure 3.4.14: Deformation of Highly Compliant Rubber Bumper Indentor . 55
Figure 3.4.15: Stringer02 Force vs. Indentation Curves ......................... 56
Figure 3.4.16: Stringer02 Delamination Schematic ............................... 57
Figure 3.4.17: Stringer02 Delamination Between the Skin and Stringer Follow-
ing Fourth Loading .......................................................... 58
Figure 3.4.18: Stringer02 Following Fifth Loading. Complete Delamination of
the Right-Side Stringer (S2) Out to the Free Edge was Observed .......... 59
Figure 3.5.1: Phase I Test Panel Design Concept .................................. 61
Figure 3.5.2: Deformation State Dependent on Boundary Conditions ......... 62
Figure 3.5.3: Hoop and Rotations Degree of Freedom Stiffness at Boundaries 63
Figure 3.5.4: Full Barrel Quarter Symmetric Model with Equivalent Elastic
Constants, 76.2 mm Radius Indentor; Model Represents a Full
Barrel With Fixed Outer Boundaries – Only \( \frac{1}{4} \) of Geometry is
Modeled Due to Quarter Symmetry ............................................. 64
Figure 3.5.5: Comparison Metrics Between the Full Barrel Model and Frame
Panel Model .......................................................... 65
Figure 3.5.6: Hoop Displacement for Various Rotational and Translational
Stiffness .......................................................... 66
Figure 3.5.7: Boundary Rotation for Various Rotational and Translational
Stiffness .......................................................... 67
Figure 3.5.8: Phase I Frame Panel Experimental Setup .......................... 68
Figure 3.5.9: Frame01 Loading History ........................................... 70
Figure 3.5.10: Frame01 Component Locations ..................................... 71
Figure 3.5.11: Frame01 During Loading L3 at 44.48 kN .......................... 72
### LIST OF TABLES

<table>
<thead>
<tr>
<th>Table Number</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.3.1</td>
<td>Cytec X840/Z60 12k Tape Lamina Material Properties</td>
<td>35</td>
</tr>
<tr>
<td>3.3.2</td>
<td>Cytec X840/Z60 Plain 6k Weave Fabric Lamina Material Properties</td>
<td>36</td>
</tr>
<tr>
<td>3.4.1</td>
<td>Stringer Panels Loading Details</td>
<td>44</td>
</tr>
<tr>
<td>4.2.1</td>
<td>X840/Z60 6K Plain Woven Fabric Lamina Elastic Properties</td>
<td>121</td>
</tr>
<tr>
<td>4.2.2</td>
<td>X840/Z60 12k Unidirectional Tape Lamina Elastic Properties</td>
<td>121</td>
</tr>
<tr>
<td>4.2.3</td>
<td>Aluminum 2024 Material Properties</td>
<td>122</td>
</tr>
<tr>
<td>4.2.4</td>
<td>X840/Z60 12k Unidirectional Tape Lamina Hashin-Rotem Failure Properties</td>
<td>126</td>
</tr>
<tr>
<td>4.2.5</td>
<td>X840/Z60 6K Plain Woven Fabric Hashin-Rotem Failure Properties</td>
<td>127</td>
</tr>
<tr>
<td>4.4.1</td>
<td>Baseline FEA Model vs. Experimental Results</td>
<td>140</td>
</tr>
<tr>
<td>4.4.2</td>
<td>Adjacent Shear Tie Failure History, Baseline FEA Model vs. Experimental</td>
<td>145</td>
</tr>
<tr>
<td></td>
<td>Results</td>
<td></td>
</tr>
<tr>
<td>4.4.3</td>
<td>Load and Displacement Values Corresponding to Figure 4.4.6</td>
<td>145</td>
</tr>
<tr>
<td></td>
<td>Labels</td>
<td></td>
</tr>
<tr>
<td>4.5.1</td>
<td>Panel Geometry and Material Effects Model Summary</td>
<td>153</td>
</tr>
<tr>
<td>4.5.2</td>
<td>Three vs. Five Frame Panel Impacted Over One Frame Summary</td>
<td>156</td>
</tr>
<tr>
<td>4.5.3</td>
<td>Number of Contacted Frames Summary</td>
<td>159</td>
</tr>
<tr>
<td>4.5.4</td>
<td>Contact Pressure and Peak Bending Stresses in the Bottom of Ply 2 at a</td>
<td>161</td>
</tr>
<tr>
<td></td>
<td>Back-Side Bumper Displacement of 31.8 mm</td>
<td></td>
</tr>
<tr>
<td>4.5.5</td>
<td>Thick Stringers FEA Model vs. Baseline FEA Model</td>
<td>166</td>
</tr>
<tr>
<td>4.5.6</td>
<td>Thin Skin FEA Model vs. Baseline FEA Model</td>
<td>168</td>
</tr>
<tr>
<td>4.5.7</td>
<td>All Aluminum 2024 Compared to the Baseline FEA Model</td>
<td>174</td>
</tr>
<tr>
<td>4.5.8</td>
<td>FEA Model With Aluminum Shear Ties and Frames Compared to the Baseline</td>
<td>182</td>
</tr>
<tr>
<td></td>
<td>FEA Model and All Aluminum 2024 FEA Model</td>
<td></td>
</tr>
<tr>
<td>4.5.9</td>
<td>FEA Model Results Summary</td>
<td>186</td>
</tr>
<tr>
<td>A.1.1</td>
<td>Cohesive Elements vs. Cohesive Surfaces</td>
<td>202</td>
</tr>
</tbody>
</table>
# LIST OF ABBREVIATIONS

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Full Form</th>
</tr>
</thead>
<tbody>
<tr>
<td>HEWABI</td>
<td>High Energy, Wide Area, Blunt Impact</td>
</tr>
<tr>
<td>FAA</td>
<td>Federal Aviation Administration</td>
</tr>
<tr>
<td>EASA</td>
<td>European Aviation Safety Agency</td>
</tr>
<tr>
<td>JAMS</td>
<td>Joint Advanced Materials and Structures Center of Excellence</td>
</tr>
<tr>
<td>OEM</td>
<td>Original Equipment Manufacturer</td>
</tr>
<tr>
<td>GSE</td>
<td>Ground Service Equipment</td>
</tr>
<tr>
<td>FEA</td>
<td>Finite Element Analysis</td>
</tr>
<tr>
<td>VID</td>
<td>Visible Impact Damage</td>
</tr>
<tr>
<td>BVID</td>
<td>Barely Visible Impact Damage</td>
</tr>
</tbody>
</table>
ACKNOWLEDGEMENTS

First and foremost, I would like to acknowledge Professor Hyonny Kim for his support and guidance during this project. I am truly grateful for the opportunity to work on this research project. I would like to acknowledge that the experimental portion of this dissertation was a joint effort with another PhD student, Zhi Ming Chen.

I would like to acknowledge the Federal Aviation Administration Joint Advanced Materials and Structures Center of Excellence for sponsoring this research. I would specifically like to thank Dr. Larry Ilcewicz, Curtis Davies, and Lynn Pham from the FAA for their technical support and input.

Thank you Cytec Engineered Materials for donating the materials used to manufacture the test panels and San Diego Composites for their manufacturing support and expertise.

I would like to thank the Boeing Company for supporting this project and for providing me with a very valuable co-op experience. Specifically, I would like to thank my mentors who have participated in various workshops for this project: Al Fawcett, Kevin Davis, Arne Lewis, David Polland, and Steven Chisholm.

Had it not been for Professor Russ Westmann, I would have attended a different university for my graduate studies. Thank you so much for steering me towards the composites field and UCSD.

Several undergraduate research assistants (Isabel Cole, Mac Delaney, Sean Luong, Jon Hughes, Sho Funai and Chris Horiuchi) made the manufacturing and testing of these large scale panels possible, and for that I am very grateful. I would also like to thank my other labmates: Sara White, Jen Rhymer and Daniel Whisler.

Thank you DM Hoyt of NSE Composites for guidance in the initial panel design. Thank you United Airlines, specifically Eric Chesmar, for allowing us to quantify the GSE threat at LAX.

Finally, I would like to thank my friends and family for their support during the dissertation writing process.
EDUCATION

2008 B. S. in Civil Engineering *magnum cum laude*, California Polytechic State University, San Luis Obispo

2011 Graduate Teaching Assistant, Department of Structural Engineering, University of California, San Diego

2013 Ph. D. in Structural Engineering, University of California, San Diego

REGISTRATION

E.I.T. State of California, 2006

MEMBERSHIPS

American Institute of Aeronautics and Astronautics

PUBLICATIONS


TECHNICAL PAPERS


PRESENTATIONS


EXPERIENCE

2008-2013 Graduate Research Assistant, Department of Structural Engineering, University of California, San Diego

Spring 2011  Graduate Teaching Assistant, *Repair of Composite Structures*, Department of Structural Engineering, University of California, San Diego

**AWARDS**

2013  UCSD Structural Engineering Department Dissertation Fellowship

2012  Federal Aviation Administration, Joint Advanced Materials and Structures Center of Excellence, *Student of the Year*

2011  UCSD Jacobs School of Engineering Research Expo, Semi-Finalist

2010  California Space Grant Fellowship

2009-2010  Transportation Research Board Fellowship on Public Sector Aviation Issues

2008-2009  UCSD Structural Engineering Department Fellowship
ABSTRACT OF THE DISSERTATION

High Energy Wide Area Blunt Impact on Composite Aircraft Structures

by

Gabriela K. DeFrancisci

Doctor of Philosophy in Structural Engineering

University of California, San Diego, 2013

Professor Hyonny Kim, Chair

The largest source of damage to commercial aircraft is caused by accidental contact with ground service equipment (GSE). The cylindrical bumper typically found on GSE distributes the impact load over a large contact area, possibly spanning multiple internal structural elements (frame bays) of a stiffened-skin fuselage. This type of impact can lead to damage that is widespread and difficult to detect visually. To address this problem, monolithic composite panels of various size and complexity have been modeled and tested quasi-statically and dynamically. The experimental observations have established that detectability is dependent on the impact location and immediately-adjacent internal structure of the panel, as well as the impactor geometry and total deformation of the panel. A methodology to
model and predict damage caused by wide area blunt impact events was estab-
ished, which was then applied to more general cases that were not tested in order
to better understand the nature of this type of impact event and how it relates to
the final damage state and visual detectability.
1 Introduction

1.1 Motivation

The use of composites in airframe primary structural components (e.g., fuselage and wing) is drastically increasing. For example, the Boeing 787, the Airbus A350, and the Bombardier C-series all contain primary structural components manufactured using composite materials. This drives a need to better understand the damage mechanisms caused by accidental impact loading.

Accidental impact loading can be caused by a wide range of threats including foreign object damage, hail and ground service equipment. Foreign object damage (FOD) can be the result of debris such as rocks, runway/pavement fragments, dropped tools, fasteners, parts, etc. [1]. These impact sources can lead to significant damage that is prospectively difficult to detect, including damage to the internal structure, that reduces the strength and stability of the composite structure [2, 3]. However, the largest source of damage to a commercial aircraft is caused by accidental contact with ground service equipment (GSE). Specifically, 50% of major damage was recorded to be caused by baggage vehicles and 60% of minor damage was caused by collision with ground vehicles and equipment [4]. An additional factor contributing to this is that flight turn-around times are short and the area surrounding an aircraft can be crowded with multiple vehicles, thereby increasing the probability of accidental contact. Figure 1.1.1 shows an example of an aircraft surrounded by several GSE between flights.
The Federal Aviation Administration (FAA) has defined five categories of damage that correspond to the load capability of the damaged structure. Category 1 damage is defined as allowable damage that could potentially go undetected, even during scheduled inspection, or allowable manufacturing defects. The structure must be capable of sustaining ultimate load capability for the duration of service life of the aircraft with Category 1 damage present. Some examples of Category 1 damage are barely visible impact damage (BVID) and small manufacturing defects. Category 2 damage would be reliably detected during scheduled maintenance. Examples of Category 2 damage include visible impact damage (VID), deep gouges or scratches, manufacturing errors that were not detected in the factory, and any local heat or environmental degradation. The aircraft structure can sustain a limit load capability until the Category 2 damage is detected in the scheduled maintenance cycle, in which the damage must be repaired to restore ultimate load capability. Category 3 damage would reliably be visually detected within a few flights by ramp personnel or be apparent by obvious signs of damage such as loss
Category 3 damage has shorter duration than Category 2 damage in which the structure can sustain at or near limit load. Some examples of Category 3 damage are large VID, fuel leaks, or cabin noise. Category 3 damage would typically be larger and more detectable than that of Category 2. Category 4 damage is discrete source damage from a specific known incident. Some examples are bird strike, tire burst, rotor burst, and severe in-flight hail. Finally, Category 5 damage is defined as severe damage created by ground or flight events that is not accounted for in design criteria. Some examples of Category 5 damage are severe GSE collision with aircraft, flight overload conditions, abnormally hard landings, and maintenance jacking errors. A key aspect of Category 5 damage is the visual detectability, some impact events involving composites will not have clear visual indications of damage [5]. For this reason, self reporting of GSE personnel is a key aspect to detectability and safety. The damage versus load capability for the five categories is shown in Figure 1.1.2.

GSE impact events are classified by the Federal Aviation Administration.
(FAA) as Category 5 damage [5]. The impact scenarios addressed in this dissertation are focused on GSE impact in the acreage area of the fuselage, i.e., at locations away from the door surround area. Figure 1.1.3 shows a patch away from the door, indicating accidental damage is possible in the acreage vicinity. This type of damage is very different from barely visible impact damage (BVID), which falls under Category 1 damage (small scratches, gouges, minor damage) where the aircraft can still maintain ultimate load [5]. Several researchers have conducted lab scale impact tests to simulate FOD and have defined damage thresholds. The energy levels required to produce delamination or other damage was on the order of 10 J, with a few exceptions of thicker laminates (e.g., a Graphite/Epoxy laminate with 96 plies) requiring energy levels up to 100 J [1, 2, 6, 7]. Olsson compared the response of a plate to a “small mass” and “large mass” for an impact event with an energy of 10 J [6]. High energy, blunt impact damage caused by GSE impact is rare and an unanticipated event, that is orders of magnitude higher in energy than FOD. For example, a belt loader with a mass of 3030 kg, traveling at 0.75 m/s has a kinetic energy of 852 J. Some cargo loaders have a mass over 15,000 kg. Contact events caused by GSE is often produced by a driver or operator error, and therefore, self reporting from the ground service personnel plays a critical role in detection.

A key aspect of this research is understanding the detectability of a GSE impact event on a composite aircraft structure. There is a major difference in the permanent deformation after impact of an aircraft manufactured from aluminum versus composite materials. Metallic structures are subject to plastic deformation when damaging events occur. However, with carbon fiber composites, the structure can undergo significant internal damage (broken frame, shear tie or delaminated stringer to skin connection) that is not necessarily visually detectable. Furthermore, the rubber bumper typically found on GSE softens the contact between the GSE and the aircraft, increasing the contact area and decreasing the contact stresses on the skin. An example of a cylindrical bumper typically found on GSE is shown in Figure 1.1.4. The bumper on the GSE allows the load to be transferred to a wider area and possibly more structural components. This research focuses on
Figure 1.1.3: Patch Away from Door Indicating Accidental Damage

damage that is widespread but not visually detectable from the exterior (impact side).

The panels evaluated in this dissertation are all monolithic construction (as opposed to sandwich panels). Monolithic aircraft structures include hoop stiffeners (referred to as frames in this dissertation), brackets connecting the frames to the skin, also known as shear ties, longitudinal stiffeners or stringers, and a skin. Figure 1.1.5 shows a drawing of a monolithic panel with each component labeled, as well as an image of a test panel before the frames were installed. The skin and stringers were co-cured, meaning that they were manufactured as one part. The shear ties and frames were manufactured individually and assembled to the skin by mechanical fasteners.
Figure 1.1.4: Cylindrical Bumper Typically Found on GSE

Figure 1.1.5: Monolithic Composite Structure
1.2 Objectives

The first objective of this research was to develop a methodology to predict damage, including the initiation and development of major failure of a composite aircraft structure impacted by GSE. This objective was achieved through a series of large scale experiments and numerical modeling. The methodology, developed in conjunction with the experimental activities, can be applied to other general impact scenarios that were not evaluated experimentally to understand the relationship between the structural configuration and resulting blunt impact damage. The methodology was validated by comparing model results with experimental data.

The second major objective was to provide a large set of experimental observations in order to develop a better understanding of the key failure modes and how they influence the global structural response and detectability of the impact event. This includes understanding internal stress states and the relationship between loading situations and impactor parameters. The second objective was achieved through experimental observation of the progressive failure history for a series of large-sized test panels, the determination of the spatial extent of damage possible within these panels, and detailed finite element analysis (FEA) of these tests. A combination of instrumentation and videos were used to collect strain, load and deformation data to clearly document the failure initiation and evolution for each experiment.

The third objective was to apply the analysis methodology to general impact cases to determine what parameters are important in the development of widespread damage that is difficult to detect. Specifically, FEA models were used to explore how the individual component material, namely aluminum vs. composite, and internal structural configuration affects the detectability of the impact event.
1.3 Approach

The investigation of damage in a monolithic composite structure caused by accidental contact with GSE was accomplished through a series of large scale experiments and FEA studies. There were multiple phases of testing, each increasing in size and complexity. Figure 1.3.1 depicts the building block approach used to understand the basic key failure modes on a smaller scale before moving to larger, more complex structures with competing failure mechanisms. The initial smaller scale test panels evaluated damage between the skin and stringers caused by a rigid, discrete indentor. Various impact locations were evaluated in order to determine the critical impact location to cause significant levels of internal damage that is not visually detectable. The test data and experimental observations were used to validate the finite element analysis methodology. Once the finite element models were validated by comparing key metrics to the experimental results, FEA was used to evaluate more general cases, develop an understanding of how the various parameters influence the structural response, and determine what parameters were more prone to creating widespread internal damage that is difficult to visually detect.

Figure 1.3.1: Building Block Approach to Understanding Damage to Monolithic Composite Structures
1.4 Novel Contributions

This research provides three major contributions to the engineering community.

• **Experimental Observations.** The first major contribution is a deep-level understanding of blunt impacts achieved via a series of experimental observations for various impact scenarios between a monolithic (stringer and frame stiffened) composite panel manufactured from aerospace-grade carbon/epoxy composite material. Specific interest was to find conditions that directly relate to widespread structural damage with little to no visual detectability. A total of ten impact scenarios are described in this dissertation. The scale of the test panels and energy of the impact events are orders of magnitudes larger than other low-velocity, high mass impact scenarios described in the existing literature. Most “low velocity” impact cases are evaluated at the lab scale on small simple panels, often 127 x 127 mm in size [1, 6, 7]. The soft impactor material and wide contact area of these impact events are also unexplored in the current literature.

• **Prediction Methodology.** A methodology to analytically predict damage caused by accidental contact between GSE and an aircraft is included in this dissertation. The methodology uses existing commercial FEA code (Abaqus) to predict damage initiation, evolution, and detectability. In order to accurately develop a modeling methodology, a experimental methodology for conducting blunt impact testing on large panels (i.e., not full aircraft) was also required and its development is included in this dissertation. Special interest was given to the importance of accurately representing the boundary conditions. Without correct boundary conditions, the deformation and stress state of the panel are completely different than what would occur in a full structure. The analysis methodology is generally applicable and can be used to evaluate other impact scenarios that were not addressed in this dissertation experimentally or analytically. A program to automatically create FEA models within Abaqus/CAE was developed that allowed various im-
impact scenarios to be evaluated efficiently and consistently. The program is an extensive Python script, that was used within the Abaqus/CAE program to define the model parameters. The program can be used as a preliminary design tool to evaluate the sensitivity of various structural geometries to impact damage.

- **General Understanding of Structural Response to a High Energy, Wide Area, Blunt Impact Event.** There are many parameters that could be explored using the FEA modeling methodology described above. This dissertation addresses the following studies of how the various parameters relate to the detectability of damage caused by a high energy, wide area, blunt impact (HEWABI) event.

  - **Soft Material vs. Hard Material Contact.** The effects of material stiffness of the impactor on the damage detectability was explored experimentally and through a series of FEA models. There was competition of initial failure modes between the bending and interlaminar shear stress developing in the skin and the contact stress between the skin and the impactor. Softer impactor material allows for more deformation in the skin, leading to higher bending stresses as the skin conforms around locations of sharp stiffness discontinuities, e.g., where the skin joins internal stiffeners. Alternatively, rigid impactors created much higher contact stresses as well as bending and interlaminar shear stress at the impactor edges that led to detectability in the form of surface-visible cracks. Figure 1.4.1 is a sketch of the frame rotation and high deformation the skin experienced when impacted with a soft impactor. The high degree of frame rotation was very influential on the global stiffness and damage evolution during the impact event.
- **Thickness of Individual Components.** The FEA modeling methodology was used to evaluate how the stiffness of various components changed the global structural response, damage initiation and failure evolution of the panel. The effects of thicker stringers, thicker frames and shear ties, and a thinner skin (all relative to the experimental panel configuration) were individually evaluated within FEA.

- **Material of Individual Components.** Hybrid structures composed of composite with aluminum skin, composite with aluminum shear ties and frames, and composite with aluminum stringers were all evaluated to understand the permanent plastic deformation of the aluminum components and how it related to detectability of the impact event. This supports the notion that composites are more likely to experience damage that is more difficult to detect visually in comparison to aluminum structures.
2 Background

This chapter provides a literature review of the topics that are directly related to this research. The following topics are addressed:

- The difference between high and low velocity impact events, highlighting the importance of correctly representing the boundary conditions
- The typical “high mass, low velocity” experimental setup
- Bluntness effects of the impactor, including current work that evaluates rubber impact on composites
- Scaling of boundary conditions from small experiments to larger panels and structures

2.1 High Velocity Versus Low Velocity Impacts

The current literature explores impacts on composite structures for a wide range of velocities, from ballistics to low velocity tool drop events. A critical aspect of any impact event is its velocity regime (low vs. high) which controls the global deformation of the impacted structure and to what degree that deformation is influenced by boundary conditions.

Figure 2.1.1 [8] shows the response of a plate for high velocity and low velocity impact events. The time during the impact event is indicated by labels (1, 2, 3) in each sketch, which corresponds to when in the force and deflection plots the indicated deformation occurs. For a high velocity event, the peak force occurs before the peak deflection and elastic wave propagation dominates the structural
response, as shown in Figure 2.1.1b. High velocity events are wave dominated because as the duration of contact force history decreases, the plate cannot respond quickly enough to develop a global response [8, 9].

**Figure 2.1.1:** Comparison Between (a) Large Mass and (b) Small Mass Impact Response [8]

For a low velocity impact event, as shown in Figure 2.1.1a, the peak load and deflection occur at the same time in the loading history. With a low velocity event, the influence of boundary conditions on the overall structural response is significant because there is sufficient time for the impact wave to travel to the boundary and return to the impact location, reaching a force - displacement equilibrium. Therefore, for low velocity tests (typical of drop weight and pendulum), a static-like structural response develops and therefore boundary conditions become crucial when attempting to accurately correlate the response of test panels to the actual response in the structure of interest. Low velocity impacts representing tool drop can be characterized as exciting deformation modes in the target that are primarily described by the first mode of vibration, or the “static mode” [10]. Higher modes can be neglected because the contact force duration is much longer than the time required for the impact wave to reach the boundaries and return. The low velocity impact phenomenon can be approximated by an energy balance model where the total energy of the system is conserved, neglecting energy of higher vibration modes, friction, and other losses [11]. Damage created from a low velocity
impact event is often more strongly influenced by the boundary conditions and the test fixture than the material properties [12].

### 2.2 Quasi-Static Equivalence

For low velocity events, dynamic impact scenarios can usually be experimentally represented using equivalent quasi-static tests. Many researchers have shown equivalence between quasi-static and low velocity impact tests [13, 14, 15, 16]. Equivalence for flat composite plates [10, 12, 17, 18, 19, 20, 21] and composite shells [22, 23] has been established. For example, Tan and Sun [20] found correlation between the strain response predicted from finite element models with quasi-static indentation and low velocity impact tests.

Wardle and Lagace [23] found that composite structures impacted at 3 m/s resulted in the same damage behavior as when they were quasi-statically loaded, as shown by the contact force versus deflection plot in Figure 2.2.1. The figure shows that the quasi-static indentation and dynamic impact tests followed the same load path up to failure. Vibrations that occurred in the dynamic tests were found to have negligible effects on the damage produced when a composite shell was impacted with a low velocity high mass object. It was concluded that damage states created during quasi-static tests and dynamic tests are equivalent if the peak loads are the same [21, 23, 24]. Due to their more controlled nature, quasi-static indentation tests can provide more consistent insight to the damage progression and interaction of damage modes than impact tests, allowing damage mechanisms to be compared between specimens [23].
Figure 2.2.1: Force-Deflection Histories for Concave Shell Specimen Impacted at 3.1 m/s and Tested Quasi-Statically [23]

2.3 Typical Low Velocity Experimental Setup

Current literature that addresses high mass, low velocity impact on composite structures is geared towards understanding the failure mechanisms caused by tool drop and foreign object damage (FOD). Damage from these sources can occur during routine operation and while the aircraft is receiving maintenance. FOD can lift up off the ground and impact an aircraft while taking off or landing. Inadvertent impact damage that can occur during maintenance consists of tools dropped, personnel stepping on delicate locations, part handling, and storage accidents [1].

Typically, tool drop and FOD are experimentally handled by drop weight or swinging pendulum impact tests. These test methods have traditionally been used to produce damage in the low velocity range, using a heavy mass to produce a kinetic energy level of interest. The basic principle of these low velocity test setups is to apply an impact threat at a known energy level and observe what damage, if
any, develops. The impact energy level can be determined by the known mass and drop height, based on energy balance (i.e., initial potential energy = kinetic energy at moment of impact) or can more accurately be known by measuring velocity of the impactor just prior to the impact event, from which the projectile kinetic energy can be calculated. The drop weight and pendulum impact setups are also amenable to the measurement of contact force history that develops between the impactor and target, by use of a dynamic load cell [25].

A pendulum impact system typically consists of a rotating rigid arm with an impactor head, or an impactor mass swinging on cables. The impactor swings into a test specimen that is typically mounted in the vertical plane. The impactor is released from a predetermined height corresponding to a desired impact energy level [26]. Similarly, a drop test system consists of vertically dropping impactors from a predetermined height onto a panel that is typically mounted in the horizontal plane [27]. The falling weight is usually guided by a tube or rail system so as to achieve better targeting accuracy. The energy of the event is determined by the mass of the impactor and the height from which the impactor is dropped.

Some advantages of the drop weight and pendulum impactors are: fairly simple setup of test apparatus, impact energy levels can be easily calculated by energy balance, impactor tip can be instrumented allowing the energy level to be correlated to the developing impact forces, and the ability to easily test a variety of tip geometries. For drop weight and pendulum impact systems, the impactor should be caught on the first rebound, eliminating multiple impacts [25, 26].

The GSE threat is several magnitudes higher in energy and contact area than tool drop or FOD. The length of a cylindrical bumper typically found on GSE (e.g., the bumper on the platform of a catering truck) can be up to ~2.4 m in length. The typical mass for a GSE vehicle ranges from 3000 kg to over 15,000 kg, and a corresponding kinetic energy of 843 J to 4200 J when traveling at 0.75 m/s. Typical pendulum and drop tests evaluate impact events on the order of 10 to 100 J in energy. Therefore, drop tests and pendulum impact tests are not an appropriate representation of wide area blunt impact caused by accidental contact with GSE.
2.4 Impactor Geometry and Material

Kinetic energy, peak force, and force history, are common parameters used to classify impact. Other considerations are the specimen thickness and layup, impactor geometry, material, and velocity of the impactor just prior to impact, and the shape and boundary conditions of the impacted structure [7].

2.4.1 Impactor Bluntness Effects

The impactor geometry plays a significant role in the damage produced [28, 29]. Poe evaluated hemispherical indentors that ranged from 12.7 to 50.8 mm in diameter, and found that thick composites indented with blunt indentors (50.8 mm diameter) reduced the tensile strength of the laminate without producing visible damage, while sharp impactors (12.7 mm diameter) caused similar reductions in post-impact tensile strength but with visible damage [28]. Elber used a 25mm diameter steel ball with a pendulum impactor to classify tool drop [10]. Typically, metal (often hardened steel) with hemispherical tips are used as the impactor and are essentially non-deforming relative to the target. The pendulum impactor and drop test allow for a variety of tip geometries, ranging in bluntness, to be tested [27, 30].

Whisler and Kim used a pendulum impactor to evaluate the effects of impactor bluntness (tip geometry ranging from 12.7 mm to 152.4 mm in diameter) on the damage threshold of thin and thick composite panels. As the radius of the impactor (i.e., bluntness) increased, so did the energy required to cause damage. The higher impact energy led to a higher peak contact force, which in turn increased the likelihood that damage would occur at locations away from the impact site due to the higher loads and lower local stresses produced by the blunter radius tip. Therefore, as the impactor bluntness increased, the contact pressure decreased and there was a less localized response. Finally, they performed similar impact studies with a 70A durometer rubber pad between the impactor and panel. The results showed that the rubber pad distributed the contact pressure and eliminated any impactor radius effects. In essence, for a given energy, the rubber pad yielded a
similar panel deformation state regardless of the tip radius. The rubber pad served to further distribute the load and create an even more blunt impact event than the largest tip radius that was evaluated [29].

### 2.4.2 Rubber Material Modeling

Currently, the literature for rubber projectile impact on composite structures focuses primarily on tire burst fragments [31, 32, 33, 34, 35]. Johnson et al. [32] simulated a high velocity tire impact event using a rubber ball with Blatz-Ko finite elastic model in the FEA code PAM-CRASH to aid in the design of composite CF/PEEK wing access cover panels. Other researchers used the hyperelastic Mooney-Rivlin material law (excluding strain rate effects) to model tire impacts on aluminum plates and access panels [33, 34]. Heimbs [31] conducted an extensive literature review on the material models used to represent the rubber in the tire impact simulations and determined that the hyperelastic Mooney-Rivlin model is the state of the art approach for a high velocity tire impact event.

The rubber material exhibited a highly deformable elastic response in the projectile under load in comparison to an impact event with a rigid projectile or impactor. This led to a more distributed loading of the target structure [31]. The high deformation of the rubber material in a drop test is shown in Figure 2.4.1. In the figure, a rigid impactor with a 40 mm diameter rubber tip is shown as it contacted a rigid steel plate for two impact energy levels, 50 J and 100 J. For both impact cases, the rubber material deformed around the rigid impactor, which created a larger contact area, i.e., a more blunt impact event, which would induce a more global response. While tire fragment impact studies do not give direct insight into the damage produced by GSE, what can be learned and associated from this is the highly deformable rubber material creates a flattened geometry during impact. Therefore, the actual geometry of the rubber bumper is less important than the structure supporting it. This soft impactor topic is revisited in the the FEA modeling portion of this dissertation in Chapter 4.
2.5 Panel Size and Scaling Effects

Large scale experiments are time consuming and very expensive, especially for composites due to the involved manufacturing process. Therefore, there is an ongoing effort to develop methods to scale lab size (i.e., pendulum and drop test) experiments to be applicable to larger full-scale structures [36, 37, 38]. Analytically derived scaling laws can predict the elastic response of a scaled structure (plates and cylinders), however, the same scaling laws do not apply to the damage formation [36, 38]. Damage formation is more complicated than predicting the elastic response, and is dependent on scale, which is consistent with fracture mechanics [36]. Other researchers have extended boundary condition scaling to other simple structures, such as beams, rods and plates. It was found that if impact cases have the same non-dimensional parameters, but different boundary conditions, materials, or size, the normalized impact response will be the same [37].

Crashworthiness testing is another closely related topic where scaling is beneficial. Good correlation between a full fuselage (1.5 m diameter), including sub floor, and a 1/5 scale model drop test was demonstrated in order to assess scaling procedures [39]. However, the structure was fairly simple, and did not include longitudinal or hoop stiffening elements.
2.6 Summary

Low velocity impact events are significantly influenced by the boundary conditions of the tested structure because the response is dominated by the first mode of vibration (i.e., quasi-static mode). Equivalence between quasi-static indentation and low velocity impact events has been established by a number of researchers. Therefore, it is acceptable to run quasi-static indentation tests in place of low velocity tests, which allows for a more detailed observation of the damage initiation and evolution.

The magnitude of a GSE impact event is significantly higher in energy and contact area than the current literature available on typical “low velocity, high mass” impact. Therefore, the standard drop tests and pendulum impactor tests are not a sufficient representation of a GSE impact event.

Another key aspect of this research is the bluntness of the impactor and how this influences the damage and visual detectability of an impact event. As the bluntness of the impactor increases, a more global structural response is induced, and the detectability of the impact event is reduced. Furthermore, a rubber pad on an impactor tip serves to increase the contact area and reduce the contact pressures. The high compliance of the rubber material impactor tip leads to flattening of the rubber under load, so the geometry of the rubber tip is less important than the geometry of a rigid tipped impactor.
3 Large Scale Experiments

3.1 Overview

The experimental work included in this dissertation can be described as large-scale blunt impact tests and was conducted over multiple phases, as described by the “building block” pyramid shown in Figure 1.3.1 in Chapter 1. A building block approach typically begins by evaluating small panels or coupons before continuing to larger, more complex structures. For this research, initially the basic key failure modes and bluntness effects of the impactor were evaluated on several small panels quasi-statically. Once a general understanding of those parameters was established, the next set of experiments quasi-statically explored fewer panels that were larger and more complex (i.e., panels included additional structural components). Phase I of this activity focused on establishing a basic understanding of key failure modes, how they were excited in relationship to bluntness parameters and impact location, and the establishment of a database measuring structural response and failure development. In addition to assessing the mechanisms of how blunt impact damage forms, the experimental results were critical to the development of modeling methodology and simulation tools for predicting damage. This is described by the bottom tier on the pyramid in Figure 1.3.1.

The following Phase II involved larger-sized test panels (and accompanying models) and accounted for dynamic effects, geometry scaling, and ever-important boundary condition effects. Because the test panels were relatively small in comparison to a full size aircraft, boundary conditions played a critical role in creating a similar deformation state between the experiments and a larger structure. At each level, high fidelity finite element modeling and correlation was used to estab-
lish damage prediction capability. This dissertation describes tests conducted for Phase I and II, but not Phase III. Phase III would be a significantly larger panel, either a quarter or half barrel size, impacted dynamically with a larger impactor or actual vehicle.

The manufacturing and testing process for composites is extremely involved and expensive. Cost and time required for fabrication and testing increased with size and complexity of the test panel. This work was a joint effort between the author and another PhD graduate student researcher, Zhi Ming Chen. There were also a number of undergraduate research assistants that contributed to the manufacturing and assembling of the panels.

3.2 Blunt Impact Threat Characterization

In order to first develop a broad understanding of blunt impacts to aircraft, the threat was characterized and quantified. This was achieved through a series of workshop meetings at UCSD as well as a GSE observation activity. At the first workshop on January 23, 2009, industry partners from Boeing, Airbus, United Airlines, Cytec, and the FAA gave presentations that identified GSE as a key threat to aircraft safety. On March 19, 2009, GSE activity at LAX was observed and documented in order to quantify a high energy, blunt impact event. Panel geometry and layup was designed and reviewed at a second working meeting in June 2009. The following findings describe the observations made at LAX that helped determine the final experimental design.

The purpose of the LAX observation activity was to make first-hand observations and document the interactions between the ground service equipment (GSE) and the aircraft between flights. Participants of the activity included Professor Hyonny Kim, Gabriela DeFrancisci, and Eric Chesmar of United Airlines (SFO), escorted by Frank Brown, a personnel safety training specialist for United Airlines at LAX, and Anthony Chin of United Airlines (LAX). Measurements, photos, and videos were taken of ground service equipment, typical bumpers, and potential contact surfaces and locations in order to scale speeds and dimensions of
the GSE. Docking practices for a variety of aircraft were documented. Approach
speeds and general near-aircraft movement of ground vehicles were also observed.

Damage to the aircraft often results from cargo movement while loading the
aircraft, or docking of the GSE and passenger bridge around the aircraft doors.
These areas are in close vicinity of the doorway openings and have a more ro-
 bust structure, due to other load requirements, than the the structure found in
the acreage area. The acreage area of an aircraft is the area between the wing
and forward or aft doors, and is simply the fuselage structure with the hoop and
longitudinal stiffeners. Other events can occur (less common) such as maintenance
equipment or other unattended GSE blown into the aircraft, or the aircraft settling
down onto equipment during the fueling and passenger loading processes, which
could cause damage further away from the doors in unprotected areas. Addition-
ally, damage far away (i.e., several feet) from the doors can commonly be observed
(as evidenced by existence of patches applied to these locations) to exist as well,
created by undefined sources, as shown in Figure 3.2.1.

The aircraft observed at LAX were the Boeing 737, 747, 757, 767, and the
Airbus 320. Different aircraft geometries and sizes influence the possible impact
sources. Smaller aircraft, such as the B737, have a shorter turn-around time be-
tween flights, often as short as 35 minutes, and therefore operations around these
aircraft can be more rushed, thereby increasing chance of GSE contact. These
aircraft are lower and there is limited space around the smaller aircraft, leading
to a higher concentration of GSE traffic levels in tight proximity of each other.
These factors also contribute to increased likelihood of impact by GSE. Figure
3.2.2 shows the underside of a Boeing 737 surrounded by several ground service
vehicles. During the LAX observation activity, the B737 was serviced by GSE for
fuel, luggage, catering, and lavatory simultaneously. As seen in the Figure 3.2.2,
the 737 fuselage is at a high risk of direct contact with the GSE due to its low
height from the ground.
Figure 3.2.1: Patches Surrounding Door on Boeing 747

Figure 3.2.2: Boeing 737 and Surrounding GSE Between Flights
Figure 3.2.3 shows the speed of a TUG 660E Belt Loader as it approached a B757, with the loading belt in the “down position”. A still image of the belt loader used to calculate the velocity is shown above the plot. Within close vicinity of the aircraft (approximately 10 cm from coming to a complete stop), the belt loader was traveling at 0.5 m/s. For a belt loader with a typical mass of 3030 kg, the energy of the vehicle traveling at 0.5 m/s would be 379 J. Another possible source of impact described is a luggage-towing vehicle driving too fast as it approaches the aircraft to unload, and accidentally colliding with the belt loader, which could then contact the aircraft.

Figure 3.2.3: Velocity of a Belt Loader Approaching a Boeing 757 Aircraft
There is a different set of threats associated with larger aircraft such as the B747 and B767. The angle of approach for docking (e.g., for a catering truck) is more difficult towards the tail of the aircraft than for the smaller aircraft due to curvature of the aircraft towards the tail and the height of the door relative to the vehicle. Figure 3.2.4 shows a catering truck with the platform raised near the tail of an aircraft. The bottom of the fuselage of the large aircraft is much higher, thereby eliminating contact from sources such as vehicles and luggage carts driving by.

![Difficult Approach Angle of Belt Loader](image)

**Figure 3.2.4**: Difficult Approach Angle of Belt Loader

Protruding features, such as the body fairing or wing, are at risk from scraping damage as the platform of a GSE vehicle is lifted to access the nearby doors. In Figure 3.2.5, the platform on a catering truck is lifting to the door forward of the body fairing on a Boeing 747. The fairing is the curved portion of the aircraft where the wing connects to the fuselage. The red arrow in the figure indicates the direction of movement of the platform.
Figure 3.2.5: Potential Scraping on Fairing as Bumper is Raised to Door Level

Another possible impact case would be if a GSE protected with a long cylindrical bumper contacted the aircraft at an angle, where only a small portion of the bumper contacted the aircraft. This impact case is shown by the rolling stairway with a cylindrical bumper touching an aircraft in Figure 3.2.6. Note, this image is not damaging the aircraft, but barely touching it. The purpose of Figure 3.2.6 is to illustrate the possible incidence angle of contact.
Based on the LAX observation, there are several possible test cases to represent accidental contact between GSE and an aircraft. It was decided in the workshop meetings with Boeing, Airbus, EASA and the FAA, that this research effort should focus on impact in the acreage area of the fuselage. By focusing on the acreage area, the test panel structure was simplified (in comparison to a door surround area) and allowed a general understanding of the key failure mechanisms between the components (e.g., frames, stringers). Two types of impactors were evaluated in the experiments, a single “point” load in the center of the frame bay to understand the damage that would occur to the stringers and skin, and a long cylindrical bumper to evaluate damage to the frames.
Impactor Description

The two most common bumpers found on the GSE were a long cylindrical bumper and a short “D” shaped bumper, as shown in Figure 3.2.7. These bumpers would produce a distributed point load or a wide line load. To match these airport-observed bumper types, similar bumpers were purchased and are shown in Figure 3.2.8. The rubber material of the “D” bumper was either ethylene propylene diene monomer rubber (EDPM) and or special process recycled (SPR) rubber polymer. The “D” bumper shown in Figure 3.2.7 was 83 mm wide, 190 mm tall, with an approximate radius of 102 mm. The material for the cylindrical bumper used in all wide line load tests was a natural rubber of approximately 60 durometer. The cylindrical rubber bumper shown in Figure 3.2.7 was approximately 178 mm in diameter.

Figure 3.2.7: Cylindrical Bumper and “D” Shaped Bumper
3.3 Panel Description

This section describes the test panels that were designed for this research. The intention was that the test panel would have a similar response under impact as if the actual acreage structure was impacted by GSE with a rubber bumper. Figure 3.3.1 shows part of a commercial fuselage barrel before final assembly and a close up view of a partial composite monolithic fuselage structure. The fuselage acreage area and wing box area are also depicted in Figure 3.3.1. In the close up view of the fuselage structure, only two frames are shown, in an actual aircraft the frames are spaced along the length of the fuselage barrel. The dashed line in the close up view shows the section of the full barrel the test panels were designed to simulate.

Three types of panels were experimentally evaluated in the Powell Laboratory at UCSD. Each subsequent phase of activity increased in size and complexity. Figure 3.3.2 shows the three types of panels and corresponding loading areas. In this dissertation, results will be presented for three StringerXX panels and four FrameXX panels. The panels were composed of four main elements: stringers, frames, skin and shear ties. The stringers are the longitudinal stiffeners that run the entire length of each test panel. The frames are hoop-wise stiffeners that exist...
to maintain the fuselage cross-section in a circular shape, and the shear ties are the brackets that connect the frames to the skin.

All three types of panels had identical component geometry, which are shown in Figures 3.3.3, 3.3.4, and 3.3.5. Figure 3.3.3 shows the cross section of the frame and shear tie connection, as well as component dimensions and fastener locations. There was a shim between the shear ties and skin that was the same layup and thickness of the stringers. This provided a uniform surface to bolt the shear ties to the skin. More details about the panel manufacturing are described in Section 3.3.2. The side view and dimensions of the shear ties, as well as the fastener locations and spacing, are shown in Figure 3.3.4. Figure 3.3.5 shows a cross section view of the stringer with dimensions. The stringer spacing for all test panels was 305 mm, measured along the arc length of the outer skin surface. The radius of curvature of the outer surface of the skin was 2794 mm for all test panels.
Figure 3.3.2: Typical StringerXX and FrameXX Test Panels

The StringerXX test panels had an overall dimension of 914 mm by 914 mm. Each test panel had either two or three stringers, depending on whether the impact location was centered directly on a stringer or between stringers. The shear ties on the StringerXX panels were attached directly to the test fixtures via 6.35 mm diameter mechanical fasteners, with a spacing of 508 mm between the shear tie rows (i.e., support to support length). More details about the StringerXX test panels are described in Section 3.4.

The FrameXX test panels were attached to the boundary conditions at each end of the frames, and the chord length of the frames was 1524 mm support to support. The width of the skin for the Phase I FrameXX test panels was 1168 mm, with an arc length of 1625 mm. The frame spacing for the Phase I FrameXX panels was 508 mm. A side view of the FrameXX panels with the stringer spacing and support length of the frames is shown in Figure 3.3.6. The only differences between the Phase I and Phase II FrameXX panels were the number of frames,
frame spacing, and overall width of the panel in the stringer direction. The Phase II FrameXX panels had five frames, spaced at 457 mm, and a panel width of 1930 mm.

The StringerXX panels were loaded quasi-statically with a discrete indentor and were intended to understand the failure to the skin and stringers caused by a more localized impact between frames. The Phase I Frame01 and Frame02 panels had three frames and were loaded quasi-statically with a long cylindrical bumper across two frames. Finally, the largest Phase II Frame03, Frame04-1 and Frame04-2 panels were five frame specimens dynamically loaded across the center three frames. The loading area for each type of test panel is depicted in Figure 3.3.2.

**Figure 3.3.3**: Shear Tie and Frame Cross Section Dimensions (all units in mm)
Figure 3.3.4: Shear Tie Dimensions

Figure 3.3.5: Stringer Dimensions (all units are in mm)

Figure 3.3.6: Frame Side View with Material Orientation for Shear Ties and Frames
3.3.1 Composite Material

Specimen materials are 177-179 °C (350-355 °F) autoclave-cured carbon fiber and toughened epoxy matrix (similar to current composite aerospace fuselage materials) provided by Cytec Engineered Materials. Specifically, these materials are X840/Z60 12k unidirectional tape, and X840/Z60 6k woven fabrics. The material properties, provided by Cytec Engineered Materials and supplemented with existing literature [42], are shown in Tables 3.3.1 and 3.3.2.

Table 3.3.1: Cytec X840/Z60 12k Tape Lamina Material Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s Modulus:</td>
<td></td>
</tr>
<tr>
<td>( E_{11} ) GPa</td>
<td>168.2</td>
</tr>
<tr>
<td>( E_{22} ) GPa</td>
<td>10.3</td>
</tr>
<tr>
<td>( E_{33} ) GPa</td>
<td>10.3</td>
</tr>
<tr>
<td>Poisson’s Ratio:</td>
<td></td>
</tr>
<tr>
<td>( \nu_{12} )</td>
<td>0.27</td>
</tr>
<tr>
<td>( \nu_{23} )</td>
<td>0.54</td>
</tr>
<tr>
<td>( \nu_{13} )</td>
<td>0.27</td>
</tr>
<tr>
<td>Shear Modulus:</td>
<td></td>
</tr>
<tr>
<td>( G_{12} ) GPa</td>
<td>7.0</td>
</tr>
<tr>
<td>( G_{23} ) GPa</td>
<td>3.7</td>
</tr>
<tr>
<td>( G_{13} ) GPa</td>
<td>7.0</td>
</tr>
<tr>
<td>Lamina Thickness:</td>
<td></td>
</tr>
<tr>
<td>Ply Thickness (mm)</td>
<td>0.142</td>
</tr>
<tr>
<td>Density:</td>
<td></td>
</tr>
<tr>
<td>Density (g/cm³)</td>
<td>1.6</td>
</tr>
<tr>
<td>Strength:</td>
<td></td>
</tr>
<tr>
<td>Longitudinal Tensile Strength (MPa)</td>
<td>2800</td>
</tr>
<tr>
<td>Longitudinal Compressive Strength (MPa)</td>
<td>1620</td>
</tr>
<tr>
<td>Transverse Tensile Strength (MPa)</td>
<td>57.2</td>
</tr>
<tr>
<td>Transverse Compressive Strength (MPa)</td>
<td>227.5</td>
</tr>
<tr>
<td>Longitudinal Shear Strength (MPa)</td>
<td>75.8</td>
</tr>
<tr>
<td>Transverse Shear Strength (MPa)</td>
<td>75.8</td>
</tr>
<tr>
<td>Failure Evolution Energy:</td>
<td></td>
</tr>
<tr>
<td>Longitudinal Tensile Fracture Energy (m-kN/m²)</td>
<td>100.0</td>
</tr>
<tr>
<td>Longitudinal Compressive Fracture Energy (m-kN/m²)</td>
<td>100.0</td>
</tr>
<tr>
<td>Transverse Tensile Fracture Energy (m-kN/m²)</td>
<td>0.2</td>
</tr>
<tr>
<td>Transverse Compressive Fracture Energy (m-kN/m²)</td>
<td>0.2</td>
</tr>
</tbody>
</table>
### Table 3.3.2: Cytec X840/Z60 Plain 6k Weave Fabric Lamina Material Properties

<table>
<thead>
<tr>
<th><strong>Young’s Modulus:</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$(E_{11})$ GPa</td>
<td>80.0</td>
</tr>
<tr>
<td>$(E_{22})$ GPa</td>
<td>80.0</td>
</tr>
<tr>
<td>$(E_{33})$ GPa</td>
<td>13.8</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Poisson’s Ratio:</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$\nu_{12}$</td>
<td>0.06</td>
</tr>
<tr>
<td>$\nu_{23}$</td>
<td>0.37</td>
</tr>
<tr>
<td>$\nu_{13}$</td>
<td>0.50</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Shear Modulus:</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$(G_{12})$ GPa</td>
<td>6.5</td>
</tr>
<tr>
<td>$(G_{23})$ GPa</td>
<td>4.1</td>
</tr>
<tr>
<td>$(G_{13})$ GPa</td>
<td>5.1</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Lamina Thickness:</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Ply Thickness (mm)</td>
<td>0.208</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Density:</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (g/cm$^3$)</td>
<td>1.6</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Strength:</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal Tensile Strength (MPa)</td>
<td>992.8</td>
</tr>
<tr>
<td>Longitudinal Compressive Strength (MPa)</td>
<td>772.2</td>
</tr>
<tr>
<td>Transverse Tensile Strength (MPa)</td>
<td>992.8</td>
</tr>
<tr>
<td>Transverse Compressive Strength (MPa)</td>
<td>772.2</td>
</tr>
<tr>
<td>Longitudinal Shear Strength (MPa)</td>
<td>77.9</td>
</tr>
<tr>
<td>Transverse Shear Strength (MPa)</td>
<td>77.9</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th><strong>Failure Evolution Energy:</strong></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal Tensile Fracture Energy (m-kN/m$^2$)</td>
<td>100.0</td>
</tr>
<tr>
<td>Longitudinal Compressive Fracture Energy (m-kN/m$^2$)</td>
<td>100.0</td>
</tr>
<tr>
<td>Transverse Tensile Fracture Energy (m-kN/m$^2$)</td>
<td>100.0</td>
</tr>
<tr>
<td>Transverse Compressive Fracture Energy (m-kN/m$^2$)</td>
<td>100.0</td>
</tr>
</tbody>
</table>

### 3.3.2 Manufacturing

All test panels were fabricated by the University of California at San Diego (UCSD) with consultation and autoclave access from San Diego Composites. This included panel design, tooling design and manufacture, fabrication and evaluation of trial parts, and fabrication of the panels. A documentation system of material out time tracking and ply layup tables with check-off sheets was used to control quality of parts production.
For Phase I, layup for all parts was done by hand at UCSD then transported to San Diego Composites to cure. San Diego Composites manufactured the Phase II skin and stringer panels. All C-frames and shear ties were manufactured by UCSD. Figure 3.3.7 shows the 1.8 m diameter autoclave at San Diego Composites with Phase I frame panel components loaded in prior to cure. The pressure, vacuum, and temperature of the cure cycle is depicted in Figure 3.3.8 as a function of time.

![Figure 3.3.7: 1.8 m Diameter Autoclave at San Diego Composites](image)

The stringers (hat stiffeners) and shims (spacers between the shear ties and the skin) were co-cured to the skin. The skin layup was $[0/45/90/-45]_2$ tape plies with an additional single layer of 6k fabric over the top and bottom, with the $0^\circ$ direction orienting parallel to the stingers. The stringers had a layup of $[0/45/-45/90/45/-45/0]_s$ with the $0^\circ$ direction orienting along the stringer main axis direction (parallel to the stringers), as shown by the top view sketch of a test panel in Figure 3.3.9. The skin and stringers had the same material orientation (depicted in Figure 3.3.9). Each stringer had a single 6k layer of $0^\circ$ fabric over the top of the hat stiffener and a single fabric layer along the skin below the stringer mold.
The skin’s curved geometry was achieved by the outer mold liner (i.e., panel
outer surface) tool shown in Figure 3.3.10. This was a 9.5 mm thick Aluminum 6061 T6 sheet which was rolled to the desired radius of 2794 mm. The rolled sheet was then bolted to an aluminum support structure with tapped holes drilled from the tool bottom side in a manner which did not penetrate through to the tool surface.

![Skin Tool with Cured Part](image)

![2794 mm Radius](image)

![1219 mm](image)

![Aluminum 6061 T6 Support Structure Bolted to Rolled Sheet](image)

![9.5 mm Thick Rolled Aluminum 6061 T6 Sheet](image)

**Figure 3.3.10:** Skin Outer Mold Line (OML) Tool

Each stringer was individually layed up by hand on a silicone insert mold. The silicone mold was wrapped in a release film prior to layup. This allowed the molds to be reused when manufacturing following test panels. The stringer lay up on the mold is shown in Figure 3.3.11. The mold was then placed on a single 6k ply of fabric that had spanned the distance from the outer edge of one flange to the other (i.e., the stringer footprint on the skin). A folded tape 0° “noodle” (originally 6.35 mm wide, then folded along the length as shown in the Noodle Detail of Figure 3.3.12) was inserted at the edge of the silicone mold before the stringer plies were layed up on the tool. The stringer to skin corner detail, as viewed by a microscope, and a sketch of the noodle and fabric ply on the mold is shown in Figure 3.3.12.
A critical detail was the stringer to skin corner detail. This was important because one of the key failure modes was delamination of the stringer flanges from the panel skin. Manufacturing trial studies were used to determine the best way to control this geometry and insure the defined radius was maintained in this location, and that the geometry was consistently produced for each panel in all stringers without defects such as wrinkles, voids, or resin rich pockets.

Each stringer mold had a three piece Aluminum 6061 insert, as shown in Figure 3.3.11. After the cure was finished, the center rectangular piece was removed first, followed by the outer two pieces. Once the aluminum inserts were completely removed, the silicone could collapse (assisted by pulling vacuum) into the empty cavity and be removed from the part.

**Figure 3.3.11**: Stringer Inner Mold Line (IML) Tool
The shear ties and frames were manufactured using only fabric material for increased drapability into compound curved regions of the tools. The shear ties had a layup sequence of $[\pm 45/0]_3S$. The frames had a layup sequence of $[\pm 45/0]_3S$ with two additional 0° plies in the flanges for additional bending rigidity. The principle direction of the shear ties and frames for all test panels was parallel to the frames. The dimension schematic drawings show the material orientation for both the shear ties (Figure 3.3.4) and the frames (Figure 3.3.6). The shear ties were fastened to the skin and shims using 6.35 mm HiLok HL19 PB8-5 countersunk shear head alloy fasteners and fastened to the frames using 6.35 mm HiLok HL18
PB8-5 protruding shear (non-countersunk) head alloy bolts, and HL70-8 aluminum collars.

The C-frames were layed up in a outer mold line (OML) defining tool. The frame tool was manufactured as four individual machined parts, which were then bolted to an Aluminum 6061 T6 12.7 mm thick plate. The four parts were welded together then machined smooth. The frame tool with a cured C-frame is shown in Figure 3.3.13. The shear ties were layed up two at a time on the machined aluminum OML tool, shown in Figure 3.3.14. By using an OML tool for the frames and shear ties smooth and well controlled mating surfaces between the parts can be achieved. Also, the geometry of the shorter section of the shear tie that fastened to the skin was also controlled by using an OML tool, thereby achieving consistent fit-up geometry. These mating surfaces can be seen in Figure 3.3.3.

Figure 3.3.13: Frame (OML) Tool
Manufacturing Challenges

It should be noted that the manufacturing activity was greatly challenging, with many setbacks in tooling acquisition (supplier manufacturing errors), as well as the large number of person-hours of labor required to cut plies and conduct layup for each panel. The labor hours devoted to this aspect of the project can not be emphasized enough. The benefit, however, was that the test panels, being 100% designed and manufactured by UCSD, has no associated restrictions affecting the public dissemination of the test results.

3.4 Stringer Panels

This set of panels was designed specifically to investigate the damage formation of the skin-stringer panels in the bays between the frames, as well as the connection to the skin, caused by a distributed area "point" indentor. Therefore, they are designated as “Stringer Panels,” with the series ID StringerXX, as shown by the drawing in Figure 3.4.1. A total of seven panels were manufactured and tested, but only three are being reported on within this dissertation. The panels discussed in this dissertation were quasi-statically indented using either a 76.2 mm radius rigid indentor or an original equipment manufacturer (OEM) rubber
bumper at two different locations relative to the stringers. The bumper material was ethylene propylene diene (M-class) rubber (EDPM) and or special process recycle (SPR) rubber polymer. The panel test matrix is shown in Table 3.4.1, and the locations of indentation are shown in Figure 3.4.2. The panels were fabricated so that the indentation location was centered on each panel, which determined if the panel had two or three stringers. A photograph of the “rigid” aluminum indentor and the OEM rubber bumper purchased from TUG, glued to a steel plate, is shown in Figure 3.4.3. The bumper was very compliant, as shown by the images of the bumper before and during indentation in Figure 3.4.3. The bumper conformed to the panel surface and led to an approximate contact area of 155 cm$^2$, in comparison to the approximate contact area of 38 cm$^2$ developed by the rigid indentor.

<table>
<thead>
<tr>
<th>Panel ID</th>
<th>Rigid 76.2mmR</th>
<th>Bumper</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stringer00</td>
<td>Location 1</td>
<td></td>
</tr>
<tr>
<td>Stringer01</td>
<td>Location 2</td>
<td></td>
</tr>
<tr>
<td>Stringer02</td>
<td></td>
<td>Location 2</td>
</tr>
</tbody>
</table>

Figure 3.4.1: Stringer Panel
Figure 3.4.2: Stringer Impact Locations

Figure 3.4.3: Rigid 76.2 mm Radius Indentor (left) and OEM Rubber Bumper (right)
3.4.1 Test Setup

The StringerXX panels, shown in Figure 3.4.1, were tested in the Powell Structural Research Lab at UCSD using a 2669 kN SATEC uni-axial tension/compression test machine. All StringerXX panels were loaded in the center of the panel with a discrete indentor. To accurately capture the force applied to the panel, the indentor was mounted directly to a 222 kN load cell, which was in turn mounted to the test machine. Indentation of the panel was measured by axial displacement of the test machine. The shear ties were bolted to heavy 12.7 mm thick L-angle steel fixtures that were water jetted to the same dimensions and curvature as the composite frames found on the larger FrameXX panels. A dimensioned side view of the StringerXX test fixture for Impact Location 2 is shown in Figure 3.4.5. The L-angles were bolted to the SATEC machine table. The typical test setup for the stringer panel is shown in Figure 3.4.4.

![Panel Stringer02 Test Setup in SATEC Machine](image_url)
3.4.2 Procedure for Test

Each panel was indented quasi-statically in multiple loading sequences. Once audible damage occurred, the panel was unloaded and evaluated for delamination in the skin with the use of a Physical Acoustics Pocket UT portable ultrasonic inspection system. A 5 MHz transducer operated in pulse-echo mode and Ultragel II® couplant were used for the scans. The pulse-echo transducer measured the time of flight of the back-face reflected pulse, which was converted into a thickness measurement of the panel. Any internal delaminations thus showed up as a “thinner” region due to the earlier reflection of the ultrasonic signal. Each panel was “A scanned”, or evaluated at one location at a time, then manually documented. The panel damage state was then documented after each loading sequence.

3.4.3 Stringer00 Results

Stringer00 was quasi-statically indented with the 76.2 mm radius rigid indenter at Location 1 (centered over stringer). Figure 3.4.6 shows the force vs. indentation curves for panel Stringer00. The plot with the markers indicates the first loading sequence (Loading L1) and the solid black plot shows the second loading sequence (Loading L2) leading up to the final load drop, at which penetration of the indentor through the skin occurred in a sudden manner. For the 76.2 mm
radius rigid indentor bearing into this short-span space between the two stringer walls, high bending stresses developed close to the location of indentor contact which is clearly visible in Figure 3.4.7.

During the first loading, the panel experienced an initial failure that consisted of a peanut shaped delamination in the skin, directly under the indentor at a load of 12.12 kN. The resulting peanut shaped delamination was caused by the high interlaminar shear stress developed at the hard edges of the 76.2 mm wide indentor at the locations circled in Figure 3.4.7. The delamination was localized to these locations. After development of this initial delamination, the test machine was stopped and unloaded, and the panel rebounded to the original undeformed shape with no visible signs of damage from the outer surface, as shown by the view of the skin after the first loading in Figure 3.4.8. The peanut shaped delamination is shown in Figure 3.4.9.

Figure 3.4.6: Stringer00 Force vs. Indentation Curves
**Figure 3.4.7**: Localized Deformation of Panel Caused by 76.2 mm Radius Rigid Indentor During First Loading of Stringer00.

**Figure 3.4.8**: No Visible Permanent Deformation After First Loading of Stringer00 Despite Formation of Delamination
The walls of the stringer, oriented out-of-plane with respect to the panel skin, provided bending rigidity in the region near the contact location. Because the indentation was at the center of the 101.6 mm spacing between the two walls of the stringer, it created a small-span stiff region directly under the indentor, resulting in the high initial portion in the stiffness as seen in Figure 3.4.6. At a load of approximately 14.68 kN during the second loading, softening was observed when the load level increased past the point of initial failure observed in the first loading. This could have been caused by compliance in the delaminating skin or softening of the shear ties under deformation. Final failure of the panel occurred at 30.68 kN in the mode of penetration of the indentor through the panel skin (see Figure 3.4.10). The penetration damage was a localized failure with no delamination between the skin and the stringer flanges.
Figure 3.4.10 shows that the skin cracked at the abrupt thickness change where the stringer was co-cured to the skin. Once completely unloaded, the panel again returned to its overall original shape, as shown by the side view of the panel after the second loading in Figure 3.4.11, with the exception of the hole produced by the penetration. Crushing of the shear tie corners closest to the center (directly loaded) stringer was also observed following the second loading.

Figure 3.4.10: Stringer00 After Final Failure Following Second Loading

Figure 3.4.11: Stringer00 After Final Failure Following Second Loading; Panel Rebounded to Original Overall Shape Despite Penetration Damage
3.4.4 Stringer01 Results

Stringer01 was loaded at a location between stringers (see Location 2 in Figure 3.4.2) by the 76.2 mm radius rigid indentor. While the observation of loud popping sounds defined the end of the first loading, no damage was detectable by visual and A-scan examination. However, since the A-scan was only applied in the central region of the panel, near the loading contact zone, it was surmised that damage might have occurred in the shear tie radius locations, which could not be inspected by A-scan due to the curvature of the shear tie. Initial (detectable) failure occurred at a load of 13.34 kN and at an indentation displacement of 14.73 mm during the second loading. When the panel was unloaded and inspected by A-scan, two small delaminations in the skin, directly under the indentor edges were observed. This is shown by the close up view of Figure 3.4.12. Note that this is roughly the same load range at which similarly-located delaminations (under indentor edges) formed in panel Stringer00, thereby confirming that this delamination was formed as a result of high local interlaminar shear stress developed at the hard edges of the indentor.

Figure 3.4.12: Stringer01 Final Failure Following Fourth Loading, Penetration of Indentor Through Skin
During the third loading, the next intermediate failure occurred at 22.83 kN and an indentation displacement of 20.57 mm. A-scan revealed that the delamination under the indentor increased in size. The panel was then reloaded for a fourth loading, during which a loud crack was heard and slight load drop observed at a load of 26.71 kN and an indentation displacement of 21.84 mm. The machine was stopped and held at a constant actuator position, corresponding to a load of 26.43 kN, to observe and photograph the damage state. However, the panel could not support this sustained load, and soon after stopping the displacement of the machine, full penetration of the indentor through the panel skin occurred.

Figure 3.4.13: Stringer01 Force vs. Indentation Curves

Figure 3.4.13 shows the force vs. indentation curves for Stringer01. Indentation of the panel was defined by the measured displacement of a potentiometer on the skin directly under the indentor. Since the location of indentation was further away from the stringer walls, the panel skin initially deformed more than Stringer00 (more compliant) under the same load. As large deformations devel-
oped and the contact area of the indentor increased, a more even distribution of the load made it difficult for additional indentation (stiffening). Thus, the curves appear to stiffen with increasing indentation level in Figure 3.4.13.

### 3.4.5 Stringer02 Results

Stringer02 was indented at the same location as Stringer01, on the panel skin between stringers, but with the deformable rubber bumper instead of the rigid indentor. Figure 3.4.14 shows the bumper during loading to convey the degree of deformation the bumper experienced as it conformed to the panel surface, which created a larger contact area (approximately 155 cm$^2$) and resulted in lower contact pressures. The interlaminar shear stress around the periphery of the contact area was also reduced due to the soft transition at the edge of contact. This made it possible to develop failure in structural features located away from the location of indentation.

Several separate loading events were applied, and are plotted in Figure 3.4.15. Indentation of the panel was measured by a displacement potentiometer placed on the underside of the panel, directly under the indentor. While the load-indentation curves for Stringer02 are similar to Stringer01, there are some inflections in the data observed at low-level loads due to the behavior of the rubber bumper as its “D” shape cavity deformed and collapsed at a load of approximately 2.89 kN.

Significant loss in stiffness can be observed with each successive loading in Figure 3.4.15 due to accumulated damage including delamination and crushing developing in the shear ties, thereby affecting the rotational stiffness boundary condition (in earlier load cycles). Delamination growth in the stringers, to the degree that global stiffness is affected, is observable as a departure from the linear force vs. indentation trend (indicating stiffness loss).
During the first two loading sequences, the damage was limited to the visible radial delamination in the radius region of the shear ties. No delamination was observed in the panel skin or between the stringers and skin. During the third loading, a large load drop occurred at 57.96 kN and an indentation displacement of 32.3 mm. The location where the panel was bolted to the shear ties experienced delamination between the skin and the stringer (i.e., at an internal structural

**Figure 3.4.14:** Deformation of Highly Compliant Rubber Bumper Indentor
There was no delamination observed between the panel skin and stringer close to the bumper contact area. Figure 3.4.16 shows a sketch of the delamination between the skin and stringers or shims that occurred in the Stringer02 test panel for the third, fourth and fifth loading sequences.

![Stringer02 Force vs. Indentation Curves](image)

**Figure 3.4.15**: Stringer02 Force vs. Indentation Curves

During the fourth loading, delamination occurred between one of the stringer inner flanges (location ID: S1F2 in Figure 3.4.16) and the panel skin close to the location of indentation, as well as at locations adjacent to the shear ties. The failure occurred at a load of 61.33 kN and a displacement of 34.5 mm. No damage was visible on the exterior surface following unloading, even after the development of significant stringer to skin delaminations. Figure 3.4.17 shows a view of the panel skin, with no visible damage, after the fourth loading. There was also delamination and crushing/bending failure in the curved section of the shear ties. It should be noted that after the load drop at 61.33 kN, the panel still held a load
of 47.06 kN, i.e., it had not been penetrated, and still had load bearing capability. However, due to the extensive damage observed, the panel was removed from the test machine for a more thorough nondestructive inspection to understand and document the damage incurred.

**Figure 3.4.16**: Stringer02 Delamination Schematic
As a result of the high residual load in Stringer02, a fifth loading was performed. During the fifth loading, the primary load path was redirected towards the intact stringer on the right-side of the panel shown in Figure 3.4.18. The panel held a load of 55.34 kN before the second stringer’s inner flange (location ID: S2F1) delaminated as well. Figure 3.4.18 shows that the final failure was a delamination of the second stringer that ran all the way to the free edge of the panel. Significant time passed between the fourth and fifth loadings to allow for a thorough examination and non-destructive mapping of the damage state. Even though the shear ties were removed and reassembled with new HiLok fasteners, and the test fixtures and panel were reinstalled on the SATEC machine, the fifth loading curve passed through the ending level of the fourth loading curve (see Figure 3.4.15). This indicates that no new damage was accumulated during the inspection.
Stringer00 and Stringer01 were both indented with an aluminum indentor. Both panels exhibited delamination within the panel skin as intermediate failures, and eventually skin penetration at approximately 30 kN. Stringer02 was indented with the rubber bumper centered between stringers, and underwent significant internal damage in the form of extensive stringer (and shim) to skin delamination that was not visually detectable from the outer surface. The rubber bumper did not produce penetration failure and thus was able to apply over 60 kN loading.

The StringerXX tests support a major conclusion, which was the indentor material significantly contributed to the detectability of the impact event. The rigid indentor created very visually detectable damage in the form of a large hole in the panel. The rubber material increased the contact area (approximately 155 cm$^2$ for the rubber bumper in comparison to 38 cm$^2$ for the rigid indentor), and significantly reduced the contact pressures and bending and interlaminar shear
stresses at the impact location. The rubber D-shaped bumper had a similar width relative to the rigid indentor (the width and length parameters are labeled in Figure 3.4.3), but the overall length was longer. Because the rubber conformed to the surface of the panel, the entire length essentially made contact with the panel. The larger contact area, combined with the soft rubber material, led to higher applied loading with lower local stresses, thereby driving formation of wide spread damage away from the impact location that was not visually detectable. The rigid indentor did not deform, so only a small portion of the surface area actually contacted the panel, developing highly localized stresses near this contact zone. Also, there was an effect of the corner of the rubber bumper being much softer than the rigid indentor, thereby decreasing interlaminar shear stress at the edges of the contact boundary.

3.5 Phase I Frame Panels

The FrameXX panels were primarily focused on understanding damage development to the circumferential frame members and their connection to the skins. Originally this panel configuration was to represent a “Steady State Zone”, as depicted in Figure 3.5.1, and be “line loaded” across two frames. However, during the industry partner Working Meeting at UCSD held on June 30 to July 1, 2009, it was agreed, based on input from Boeing and Airbus, that the critical location was where the indentor terminates, i.e., where bi-axial bending in the skin develops. This is the “Transition Zone” condition indicated in Figure 3.5.1. A third frame was therefore added to the UCSD Phase I FrameXX panels, and the end of the long indentor terminated directly under the center frame. In this manner, this configuration includes one frame in the "Transition Zone" and one in the "Steady State Zone," and the whole panel can be thought of as a pseudo half-symmetric representation of a six-frame panel (the term "pseudo" acknowledges that the symmetry is not perfect since skin at the center has been trimmed close to the frame, and non-rotating boundary conditions were not enforced at that edge).
3.5.1 Representation of Boundary Conditions

The boundary conditions for the frame panels were critical to ensure that the panel response was similar to the behavior of a full barrel. Due to the low speeds of the ground service equipment represented by these experiments, a pseudo quasi-static response would be excited in the panel structure, and therefore boundary conditions play an important role in the panel response. Boundary conditions were carefully designed to allow correlation of data from a substructure panel to the full barrel. Figure 3.5.2 is a sketch of the response of a simply supported panel with moving boundary and a fully clamped boundary condition panel. It can be seen that the deformation of the panel under load is significantly influenced by the choice of boundary conditions, particularly for this "arched" curved geometry. In reality, the full structural response is somewhere between the two cases.
Employing finite element analysis, an iterative approach was taken to determine the boundary conditions required in order for the frame panel specimen to have a similar response as if it were part of a larger structure (e.g., a full barrel). To ensure proper boundary conditions of the frame panels, spring stiffness was implemented in the hoop direction $K_H$ (i.e., direction tangential to panel surface) and the boundary support rotational degree of freedom $K_\theta$ of the test panels. Figure 3.5.3 is a schematic of the boundary conditions that were applied to the panel model to determine the appropriate frame end boundary conditions for the experiments.
The determination of the appropriate value of spring stiffness was achieved via a set of finite element models of the full barrel (see Figure 3.5.4 for the quarter symmetric full barrel model), and the Phase I FrameXX test panel (see the “Frame Specimen Model” in Figure 3.5.5). The full barrel model was a quarter symmetric model, fixed at each end of a 6.1 m long barrel. The test panel model was supported at the frame ends and was used to evaluate different hoop and rotational frame boundary stiffness. The comparison metrics were local indentor displacement, hoop displacement and frame rotation, which are shown in Figure 3.5.5.

In Figure 3.5.5, the local indentor displacement $\delta$ was used as the common “loading” metric for comparison between the frame panel and the full barrel models. In the frame model, there was no global displacement due to the boundary conditions. In the full barrel model, the local indentor displacement was the total indentor displacement minus the global displacement at the frame location indicated in Figure 3.5.5. In the “Full Barrel Model” of Figure 3.5.5, the global displacement is labeled as $\delta_G$ and the hoop displacement ($\delta_H$) and frame rotation ($\delta_\theta$)
are described in the close up view labeled “hoop displacement and rotation”. An iterative process was used to determine the correct spring stiffness needed in the frame panel boundaries such that boundary displacements were equivalent to the displacements in the full barrel model. Hoop-direction and rotation displacements must have the correct stiffness to match values between the frame panel and the full barrel. Doing so would achieve a similar deformation state, thereby making results measured from the frame panel similar to the full barrel. The displacements and rotations at the boundary were matched based on data from the second frame in the full barrel model and the outside frame (fully under indentor) of the frame panel (see “Frame of Interest” in Figures 3.5.4 and 3.5.5).

**Figure 3.5.4**: Full Barrel Quarter Symmetric Model with Equivalent Elastic Constants, 76.2 mm Radius Indentor; Model Represents a Full Barrel With Fixed Outer Boundaries – Only $\frac{1}{4}$ of Geometry is Modeled Due to Quarter Symmetry
Figure 3.5.5 shows the local indentor displacement and the hoop displacement ($\delta_H$) for the full barrel model and several test panel models with varying rotational ($K_\theta$) and translational stiffness ($K_H$). A good match would be a panel model that produced the same hoop displacement for a given local indentor displacement when compared to the full barrel model (i.e., the same slope is desired). In Figure 3.5.6, the panel model with no added rotational or translational stiffness produced a very close match to the full barrel hoop displacement plot. The solid line plot signifies the full barrel model and the simply supported model is indicated by the dashed line (labeled Panel Model $K_H=0$ kN-m/rad, $K_\theta=0$ kN/m), which match each other closely. From this comparison, it can be deduced that the translational stiffness ($K_H$) is inconsequential to creating an equivalent hoop displacement between a panel model and the full barrel model.
While the simply supported case matched well with the hoop displacement, the frame rotations did not match as well. The same approach that was taken to compare the hoop displacements was repeated to compare the frame rotations at the boundaries. Figure 3.5.7 shows a comparison for the frame rotation at the panel frame end boundary location of the full barrel model to the panel models with various stiffness’s. The rotational stiffness of a model with 1129.8 kN m/radian and no translational stiffness (depicted by the solid line with the triangle markers shown in Figure 3.5.7) correlated well the rotational stiffness of the full barrel model. For this given boundary condition, the rotational stiffness reduced the hoop displacement. However, at 30 mm of local indentor displacement, the hoop displacement for the panel with a rotational stiffness of 1129.8 kN m/radian (and no translational stiffness) was still within 18% of the hoop displacement of the full barrel model for the same amount of local indentor displacement.
The peak bending stresses in the top of the frame were compared between models. At an indentor displacement of 12.2 mm, the peak bending stresses in the full barrel model was 27.65 MPa. For the model with no added rotational or translational stiffness the peak bending stress was 39.93 MPa, and the panel with a high rotational stiffness but no translational stiffness gave a peak bending stress of 31.30 MPa. It was determined that a high rotational stiffness was critical to create a similar deformation state between the full barrel and panel model. The model with no stiffness added allowed the panel to deform too easily, which created very high bending stress in the frame.

### 3.5.2 Phase I FrameXX Test Setup

The Frame01 and Frame02 panels were mounted vertically on a strong wall in the Powell South Research Lab. Each frame was supported with a pin-roller
setup, with controlled rotational stiffness (1,130 kN-m/rad) at each frame end. The rotational stiffness was achieved through the use of stiffness plates. The panels were indented with a rubber cylindrical original equipment manufacturer (OEM) bumper (purchased from SAGE Parts, model number 660301201, natural rubber - 60 durometer) with an outer diameter of 178 mm, inner diameter of 127 mm, and length of 572 mm (cut to length at UCSD). The bumper was mounted to a 152 mm x 152 mm x 6.25 mm wall square box beam 762 mm in length, supported by two 222 kN load cells. The indentor and supporting fixture assembly were mounted on a 1-D shake table in the UCSD Powell Structural Research Lab, with the table used as a 1-D actuator applying quasi-static displacement-controlled loading. The experimental setup is shown in Figure 3.5.8.

![Figure 3.5.8: Phase I Frame Panel Experimental Setup](image)

### 3.5.3 Procedure for Test

In the Phase I tests, the 1-D shake table was used in a quasi-static displacement control mode to apply indentation slowly to the panel at a rate of 3 mm/min. The testing process consisted of multiple loading sequences, with unloading after
significant damage events occurred (shear tie crushing, stringer severing and delamination, cracked frame). Frame01 was tested in a series of four loading events before one of the frames was severely damaged. Frame 02 provided a direct load path (through the stringer to frame contact interaction) to the internal structure, and frame cracking occurred after the second loading.

### 3.5.4 Frame01 Results

Panel Frame01 was indented at the center of the panel between stringers. With each successive loading (referred to as Loading L1 to L4), the panel lost overall stiffness due to the development of damage, as shown by the change in slope for each successive loading sequence. Figure 3.5.9 shows the total load as a function of the skin displacement, measured on the skin centered between frames directly under the bumper. Figure 3.5.10 shows a view of the panel mounted to the strong wall and identifies each component discussed in this section (e.g., Frame #1, shear tie F01H). The rotational stiffness was achieved via combination of shaft+bearings and 25.4 mm thick steel flexure plates. The zero hoop stiffness was achieved by permitting translation of the lower set of end supports.

During Loading L1, audible damage at a load of 28.91 kN was followed by a load drop of 1.11 kN. No visible damage was observed, however, it was deduced that the center shear tie F01H on Frame #2 (see location F01H indicated in Figure 3.5.10) experienced delamination in the radius region. This assessment was supported by back to back strain gauge data on shear tie F01H. During Loading L2, both Frames #1 and #2 experienced rotation of the cross section (due to shear-center effect on open cross-section) at the loaded location. At 57.83 kN and 16.5 mm indentation, shear tie F01H completely failed and Frame #2 rebounded (less rotation of the frame), followed by a load drop of 11.12 kN. The damage to the panel was limited to shear tie F01H broken into multiple pieces and significant damage to the center shear tie F01C attached to Frame #1.
During Loading L3, the stringer contacted Frames #1 and #2, creating a direct load path to the frames. The vertical location of the bumper was designed so that the cylindrical bumper was centered between stringers upon initial contact. However, when the cylindrical bumper was fully compressed, the indentation area was more accurately defined by the 152.4 mm width of the steel box beam that supported the bumper. The steel support box with the fully compressed bumper and high deformation of Frame01 is shown Figure 3.5.11. During large deformations, the panel exhibited an overall elongation which caused a slightly eccentric loading location that was biased towards the upper stringer. In Figure 3.5.11 the steel box was directly in line with the upper stringer flange, therefore, the upper stringer was the primary load path after complete failure of the loaded shear ties.
Figure 3.5.10: Frame01 Component Locations

The first load drop in Loading L3 at 40.6 mm indentation (55.6 kN, load drop of 8.9 kN) was caused by a dramatic shift and increase in the rotation of Frame #2, which reduced the bending rigidity of the frame (no new fiber cracking observed). At 53.3 mm of indentation (55.6 kN), complete full-width fracture of shear tie F01C led to a load drop of 15.1 kN. Figure 3.5.12 shows the damage to the backside of test panel Frame01 after Loading L3. The bumper was positioned to load Frame #1 and Frame #2, which are labeled in the figure. Both shear ties (ID labels: F01C and F01H in Figure 3.5.10) are fully crushed in two locations and missing a large portion of the shear tie, which is shown in the close up view of the internal damage shown in Figure 3.5.13. The final damage during Loading L3
was severing (fiber failure) of the stringer above the indentor due to penetration of Frame #2 through the stringer at 62.2 mm indentation (44.5 kN, with a load drop of 1.3 kN), which is shown in Figure 3.5.12 and Figure 3.5.13. This was immediately followed by a load drop of 9.3 kN and delamination of the top stringer flange. Figure 3.5.14 shows the outer surface after Loading L3. The delamination between the stringer and skin is shown by the hatched regions and the location where the upper stringer severed is indicated in the figure.

Figure 3.5.15 shows the indented surface after Loading L3. On the left is a close up view of the skin and bumper after indentation. There is a faint outline on the skin where the rubber bumper contacted the skin surface. The full panel is shown on the right of Figure 3.5.15 after the bumper was removed. This figure shows there was not any significant visually detectable residual dent, and does not give any indication of the extensive internal damage shown in Figures 3.5.12 and 3.5.13.

Figure 3.5.11: Frame01 During Loading L3 at 44.48 kN
Figure 3.5.12: Frame01 Post L3 Major Damage: Backside Interior View

Figure 3.5.13: Frame01 Post L3 Damage, Closeup Interior View
Figure 3.5.14: Frame01 Outer Surface Showing Stringer to Skin Delamination Damage

Figure 3.5.15: Frame01 Outer Surface After Extensive Internal Damage
During the final loading sequence L4, through-thickness cracking in Frame #2 was observed where the stringer below the indentor contacted the frame. The Frame #2 crack is shown from both sides of the frame in Figure 3.5.16. With the reduced stiffness of the upper stringer, the lower stringer to frame interaction was the primary load path for Loading L4. The test was stopped at a load of 47.36 kN in order to prevent damage to Frame #3, which was reused in the following Frame02 test. It is hypothesized that if loading had continued, further damage to Frame #2 or damage to the lower stringer would have occurred. After loading L4, there was no clear exterior-visible damage to the panel except a few small cracks in the skin surface by the bolts on the center shear ties.

![Image showing Frame #2 cracking observed during Loading L4](image)

**Figure 3.5.16**: Frame01 Cracking Observed in Frame #2 During Loading L4

**Photogrammetry**

Photogrammetry is a technique that uses images to quantitatively determine geometric properties. For this study, the geometric property of interest was the outer surface profile of each large panel before and after damaging events, which was then used to assess whether there was any permanent deformation of the skin (i.e., a visually detectable dent). The photogrammetry analysis showed
permanent deformation of less than 20 mm over curved span of 1.27 m and a width of 1.0 m. Permanent deformation was confined to the location of applied loading (across Frames #1 to #2, under the Bumper Area), but there was almost no permanent deformation over Frame #3. Figure 3.5.17 shows four photogrammetry plots of Frame01 before loading, and after Loadings L2, L3, and L4. The scale on the right of the figure indicates the out of plane location (in the Z direction) of the skin surface. The Prescan Surface Profile shows the uniform curvature of the panel before any damage. After the subsequent loading sequences which caused extensive internal damage, the skin did not rebound to exactly the same geometry. However, the residual deformation was fairly subtle and difficult to detect visually.

**Figure 3.5.17**: Photogrammetry of Frame01 Showing Low Amount of Residual Deformation
3.5.5 Frame02 Results

The Frame02 panel was indented at a location directly centered on a stringer. The panel experienced significant skin deformation and extensive internal damage including a cracked frame and fractured stringer. The load vs. displacement plots for the two successive quasi-static loadings (referred to as Loading L1 and L2) applied to this panel are shown in Figure 3.5.18. The displacement was measured by a potentiometer centered on the top of the loaded stringer between Frame #1 and Frame #2. Figure 3.5.19 shows the skin deformation under the maximum bumper displacement during Loading L2.

![Figure 3.5.18: Force vs. Displacement of Frame02](image)

During Loading L1, the skin and shear ties deformed enough to allow the stringer to make contact with the two frames directly under the indentor. This provided a direct load path from the indentor to the frames before failure of the
shear ties. At a displacement of 32.5 mm (62.5 kN, with a load drop of 2.3 kN), a crack in the skin centered under the stringer originating from the edge of the panel developed and the panel was unloaded. The skin crack is shown in Figure 3.5.20. In addition to the 155 mm long skin crack, there was crushing damage at the corner of the four shear ties above and below the center stringer. There was no detectable delamination between the stringers and skin using ultrasonic A-scan.

![Frame02 Skin Deformation During Loading L2](image)

**Figure 3.5.19:** Frame02 Skin Deformation During Loading L2

During Loading L2, the contact between the center stringer and frame played a critical role in the behavior of the panel. Prior to penetration of the frame through the stringer, a series of small load drops occurred as the frames rotated open. This rotation was constrained by the frictional forces between the contacting frames and center stringer. When the friction forces were overcome, the frame incrementally rotated and in doing so, the bending rigidity of each frame was
reduced and the panel became more compliant. Loading L2 caused the skin crack to grow to a length of 290 mm, measured from the free edge. In Figure 3.5.21, the interaction between the frame and stringer caused a visible local deformation in the stringer that conformed to the shape of the corner of the rotated frame. At a displacement of 47.6 mm (70.9 kN), Frame #2 (center frame) penetrated through the center stringer. Further penetration of Frame #2 at a displacement of 52.7 mm caused additional stringer and shear tie damage.

![Image](image-url)

**Figure 3.5.20**: Frame02 Skin Crack Originating From Boundary, Post Loading L1

Finally, Frame #1 cracked just below the center stringer at a displacement of 56.3 mm and a load of 58.9 kN. Figure 3.5.22 shows the Frame#1 crack during the test (left) and the crack after the panel was unloaded (right). Scuff marks on the stringer indicate where the frame and stringer contacted (right image of Figure3.5.22). The damage progression during Loading L2 in panel Frame02, as well as the observations in L3 and L4 in panel Frame01, shows a key interaction between the contacting stringers and frames, where there was a competing failure between frame cracking and frame penetration into the stringer. The damage progression is dependent on the design details and layup (thickness) of the stringer and frames, as well as the amount of rotation the frames undergo, which affects the geometry of the contact.
Figure 3.5.21: Frame02 Stringer to Frame Contact Interaction

Figure 3.5.22: Frame02 Broken Frame #1 at 58.9 kN Experiencing Significant Rotation Under Load (left) and Damaged Frame After Unloading (right)

Figure 3.5.23 shows the damage that occurred to the shear ties on Frame
#1. Hinges formed just below the fastener line on the shear ties, which allowed additional frame rotation and eventually a crack formed in Frame #1 where the frame contacted the stringer. The rotation of the frames created a pull-off type loading on the shear ties. The crushing damage in the radius region of each loaded shear tie is shown in the close up view of the top right corner of Figure 3.5.23. Figure 3.5.24 shows the damage of the center stringer and Frame #2. Frame #2 penetrated through the stringer, which can be seen by the fiber damage in the close up side view of the broken stringer (upper right corner of Figure 3.5.24). There was also extensive shear tie crushing in the vicinity of the indented area.

![Figure 3.5.23: Damage and Fastener Line Hinges of Frame02](image-url)
Photogrammetry was used to quantify the residual deformation in the skin as a result of each loading sequence and the damage the panel incurred. Figure 3.5.25 shows the surface plot prior to indentation, and after Loadings L1 and L2. There was very little permanent deformation in the skin, on the order of ~10mm. Frame01 was indented at a more compliant location than Frame02, which led to a slightly larger permanent deformation. However, the permanent deformation of the skin for both Phase I panels was small, and would be difficult to visually detect.
3.5.6 Phase I FrameXX Discussion

The Frame01 specimen exhibited significant progressive damage. The shear ties were the first component to experience damage and complete failure. Contact between the stringer and frame provided a direct load path to the frames, even before the shear ties were completely severed. Increased frame rotation caused dramatic load drops as the bending rigidity of the individual frames decreased. In Figure 3.5.26, the combined loading history of Frame01 (gray plot) and Frame02 (black plot) test results are shown together for comparison.
It can be seen that the Frame02 had a higher load capacity than Frame01, but less progressive damage evolution leading up to the cracking of a frame. Frame02 was contacted in an area with higher local stiffness and provided a more direct load path to the frames than Frame01. The bumper loaded directly onto the stringer in the Frame02 panel test, which then contacted the frames. Therefore, damage to the internal structure in test panel Frame02 was more dependent on the interaction between the frame and center stringer than the progressive damage that developed in the shear ties. For both panels, the amount of damage to the panel directly correlated with the level of indentation of the skin. Bumper contact onto the skin spanning between stringers is more likely to lead to damage that is difficult to detect, as seen in the Frame01 results. There are no sharp stiffness transitions when indented between stringers in comparison to when the bumper contacted directly on top of a stringer wall-to-skin junction which created.
a bending stress concentration.

3.6 Phase II Frame Panels

The Phase II FrameXX panels were larger-sized panels composed of skin, four stringers, and five frames connected to the skin via mechanically-fastened shear ties. These panels were tested dynamically. Two of these panels were fabricated, referred to as Frame03 and Frame04. Frame03 and Frame04 had a slightly smaller frame spacing than the Phase I Frame01 and Frame02 panels due to autoclave size limitations defining the maximum panel length. The geometry of each component was identical to the previous test panels.

3.6.1 Test Setup

The same fixtures used for the boundary conditions in the Phase I FrameXX tests were reused on the boundary conditions of the center three frames of the larger Phase II FrameXX panels. The Phase II FrameXX panels were impacted across the center three frames, with a frame on each end of the panel outside the impact area, as shown in the sketch of the Phase II FrameXX panel in Figure 3.6.1. The impact area is outlined by the dashed box in the sketch and was centered on the panel between stringers. Loading was applied dynamically by a 1.0 m long bumper mounted on a dynamic servohydraulic actuator. Boundary conditions, as visible in the test setup in Figure 3.6.2, included rotating end supports for each loaded composite frame with controlled rotational stiffness achieved via flexure plates.
Figure 3.6.1: Phase II Impact Location

Figure 3.6.2 shows the Phase II FrameXX test set up in the Powell Laboratory at UCSD. The dynamic servohydraulic actuator was mounted to a steel I-beam frame test structure, shown in the sketch in the top right corner of Figure 3.6.2. Two 222 kN dynamic-rated load cells were mounted to the impactor fixture, which was mounted to the actuator. An aluminum 152.4 x 152.4 mm (6.35 mm wall) box beam, 1.0 m in length, was attached to the load cells. Supports with rollers prevented any out of plane displacement of the aluminum box beam. The 1.0 m long cylindrical rubber bumper, identical to the bumper used in the Phase I FrameXX tests (except for the length), was attached to the aluminum box beam via steel hose clamps.
Equivalent Aluminum Frames

In order to reduce the complexity of the experimental setup and to avoid manufacturing additional composite C-frames, the outer frames on the test panel were made of Aluminum 2024 water jetted plates. The boundary conditions on the aluminum frames had no rotational stiffness and were free to translate on one end. The cross section of the aluminum was calculated to produce a similar bending rigidity (EI) for the composite and aluminum frames. The Aluminum 2024 frames had a rectangular cross section with a width of 6.35 mm and the same height as the C-frames (108 mm).

A FEA study verified the material, cross section and boundary conditions of the outer frames did not significantly change the global stiffness of the panel under load. Three cases were evaluated, composite C-frames with rotational stiffness boundary conditions, aluminum boundary frames with rotational stiffness, and aluminum boundary frames that were simply supported with no rotational stiffness. It can be seen from Figure 3.6.3 that the overall stiffness was identical for the aluminum frames with simply supported boundary conditions and aluminum frames with rotational stiffness boundary conditions and very close (within

Figure 3.6.2: Test Set-up for Phase II Test Panels
5% at 10 mm of bumper displacement) for the model with composite boundary frames with rotational stiffness. Therefore, it was decided to use the more simple aluminum frames that were simply supported with no rotational stiffness for the outer boundary conditions.

![Figure 3.6.3: Loading History for Various Outer Frame Conditions](image)

### 3.6.2 Frame03 Results

**Test Procedure**

The first five-frame panel (Frame03) was tested in early March 2012. In Figure 3.6.4, the actuator displacement for both loading sequences is depicted as a function of time. The panel was loaded two times (referred to as L1 and L2) under displacement control, each with a constant velocity of 0.5 m/s, followed by a 0.5 s pause before unload. The actuator provided a constant velocity for the loading and unloading portion of the loading cycle, and did a good job of transitioning from a constant velocity of 0.5 m/s to 0 m/s. Loading L1 had a total actuator displacement of 159 mm, which included closing the initial gap (between the panel
and bumper) of 6.4 mm. The total displacement of the actuator in the second Loading L2 was 222 mm, which also includes closing the initial gap between the bumper and skin surface. Note that skin displacement will be much lower than the actuator stroke due to the deformation/flattening of the hollow bumper. Figure 3.6.5 shows the Frame03 panel during the peak impactor displacement of Loading L2.

![Actuator Displacement Profiles for Frame03](image)

**Figure 3.6.4:** Actuator Displacement Profiles for Frame03

**Observations**

During the first loading, there was moderate crushing damage in the radius region of the shear ties directly under the impactor, but there was no delamination between the skin and stringers or shims. The cylindrical bumper had a hollow inner diameter of 127 mm, so the expected displacement of the panel surface was on the order of 25 mm. A small load drop was observed at 18.3 mm of skin displacement and 62.1 kN in Loading L1, shown by the dashed plot of the displacement versus loading history in Figure 3.6.6.
The second loading sequence was intended to produce major failure, in the form of either frame or stringer damage. Figures 3.6.7 to 3.6.10 show a sequence of high speed video still captures that give insight into the failure process. Upon complete crushing of the directly loaded three shear ties (see Figure 3.6.6 at 34.3 mm, and visible in Figure 3.6.7), the load was then transferred from the stringer directly to the C-frames. This direct stringer-to-frame load path was confirmed by the scraping marks observed after impact, visible on the stringers and C-frames where contact occurred, as indicated in Figure 3.6.11. As the impactor displacement increased, the C-frames continued to rotate and hinges formed along the fastener line in the shear ties adjacent to the loaded shear ties on the center frame (Frame #3). Figures 3.6.7, 3.6.8 and 3.6.9 reveal that the shear ties on the center frame (Frame #3) formed a hinge along the fastener line before the adjacent shear ties on Frames # 2 and 4. After the adjacent shear ties on the center frame failed along the fastener line, the load carrying capability of the center frame was reduced. The failures along the fastener line in each shear tie occurred in a specific order, with the time difference between the initial hinge forming on the adjacent shear ties of the center frame (see Figure 3.6.7) and the final hinges forming on the adjacent shear ties of Frame #4 (see Figure 3.6.9) being just 11 ms. In the load history (Figure 3.6.6), this progression is depicted by the increasing loading
leading up to the major load drop at 65 mm of skin displacement.

Once all adjacent shear ties failed along the fastener line, there was little resistance to C-frame rotation. Figure 3.6.9 shows the failed shear ties and excessive frame rotation of each loaded C-frame just prior to frame failure. Final failure occurred in the C-frames away of the impact region (i.e., non-local failure) due to a combination of torsion, bending and shear. Figure 3.6.10 shows a still image just after the middle three loaded frames failed. The frame ends failed slightly differently based on the boundary conditions. The frame end that allowed translation had a failure line that was vertical and shear dominated because roller boundary condition alleviated some of the torsional load by allowing translation. The pin with rotational stiffness boundary condition led to a diagonal crack path, as shown by the images of the fractured frames in Figure 3.6.12.

**Figure 3.6.6**: Loading History for Frame03
**Figure 3.6.7**: Frame 03, Loading L2 Shear Tie Failure Progression 1

**Figure 3.6.8**: Frame 03, Loading L2 Shear Tie Failure Progression 2
Figure 3.6.9: Frame 03, Loading L2 Shear Tie Failure Progression 3

Figure 3.6.10: Frame 03, Loading L2 Shear Tie Failure Progression 4
Figure 3.6.11: Frame and Shear Tie Damage of Frame03 After Loading L2

Figure 3.6.12: Frame03 Frame Failure After Loading L2
While the second loading caused extensive damage to the internal structure, the presence of damage was not visually detectable from the skin side (no visible cracks formed and no gross dent was visible). The final internal damage state is shown in Figure 3.6.11 and by the frame failure near the boundaries in Figure 3.6.12. A view of the outside of the panel after impact is shown in Figure 3.6.13. It should be noted that even after the frames were severed, the panel still held a load of approximately 15 kN per frame before unloading. While no immediately surface crack or other surface damage was visible, a minor surface geometry change was measured. The photogrammetry method was used, as summarized in Figure 3.6.14, to measure a 4.5 mm permanent deformation. The visual detectability of 4.5 mm acting over a “dent span” of 1.0 m was found to be difficult to perceive by casual observation.

Figure 3.6.13: Frame03 After Loading L2
Figure 3.6.14: Photogrammetry of Frame03 After Impact Showing Residual Deformation

3.6.3 Frame04-1 Results

The second panel in the Phase II dynamic tests was identified as Frame04. This panel was tested in two separate configurations, Frame04-1 and Frame04-2. Test panel Frame04-1 was identical to Frame03. Due to the pre-existing damage (partial cracking in the radius-area) of the shear ties in Frame03 caused by Loading L1, there was no way to capture the peak load that would occur from an impact event for a pristine panel based only on the Frame03 test data. Therefore, for modeling correlation purposes it was desired to re-test this configuration and capture the first load drop of a pristine panel as the actuator loaded the panel through to initial shear tie failure (or some major failure mode).

Testing for Frame04-1 occurred in concurrence with a working meeting with industry partners. During the working meeting, it was determined that a stronger shear tie was desired to induce a different final failure state for the second panel. Thus, it was decided to load panel Frame04-1 to an initial moderate damage level with no failure of the frames, and to replace the shear ties to form panel config-
uration Frame04-2. In order to be able to reuse the Frame04 panel after the -1 loading sequence, the actuator displacement was chosen such that only the shear ties would fail without creating damage in the stringers or frames. Figure 3.6.15 shows Frame04-1 during impact subjected to 177 mm of actuator stroke (with 2.5 mm initial gap).

![Frame04-1 During Impact](image)

**Figure 3.6.15: Frame04-1 During Impact**

Figure 3.6.16 shows a load history comparison of Frame04-1 and the combined loading sequences of Frame03. This is shown clearly in the comparison in Figure 3.6.16. The first minor load drop at 16.1 mm of skin displacement and 63.5 kN was caused by crushing in the radius region of the loaded shear ties. This minor load drop and the load vs. displacement profile up from 0 to 22 mm was almost identical to the initial failure and loading profile observed in the first loading sequence of Frame03. As the impactor displacement increased, producing skin displacement beyond 22 mm, the panel supported a load of 91.6 kN. The sudden load drop at 33.7 mm of skin displacement was caused because the shear ties buckled and completely failed. No damage occurred in the frames, stringers, or skin in the Frame04-1 test.
Figure 3.6.17 shows a still from the high speed camera during the impact. The shear ties buckled and failed in two locations, near the radius region and at the bottom of the frame. Little frame rotation was observed during the loading. The damage was limited to the failed shear ties and the surrounding structure remained undamaged after impact. Figure 3.6.18 shows the inside view of the frames after impact and Figure 3.6.19 shows the skin from the impact side after the impact event. No indication of damage from the outer surface in the form of residual skin deformation or surface cracks.

Figure 3.6.16: Load History of Frame03 and Frame04-1
**Figure 3.6.17**: Frame04-1 Shear Tie Damage During Impact

**Figure 3.6.18**: Frame04-1 Shear Tie Damage After Impact
3.6.4 Frame04-2 Results

It was decided in an industry and FAA workshop before testing Frame04-1 that it would be beneficial to impact a large scale panel with stronger, aluminum shear ties. Therefore, the previously tested Frame04-1 specimen was then removed from the test fixtures and all shear ties on the three center frames (nine shear ties total) were replaced with more robust aluminum shear ties. The panel was then reinstalled in the test fixtures and tested to failure. The Frame04 panel with the robust aluminum shear ties is referred to as Frame04-2.

Figure 3.6.20 shows the larger Aluminum 7075 shear ties. The shear ties were composed of two parts, the flat web region which was water jetted to the desired specifications with undersized holes. The curved region that connected the shear tie to the skin was machined to the desired specifications, then connected to the web region by six 6.35 mm diameter standard fasteners. All composite to aluminum fastener connections were assembled with the same type of HiLok fasteners that were used in previous tests. The aluminum shear ties were significantly more robust than the composite shear ties, with a thickness of 3.175 mm (27% thicker...
than the composite shear ties on Frame03 and Frame04-1) and a width of 254 mm, with two additional fasteners between each shear tie and frame. The thicker shear ties not only prevented bucking in the shear tie web, but also resisted frame rotation.

![Frame04-2: Aluminum Shear Ties on Frame04-2](image)

**Figure 3.6.20:** Aluminum Shear Ties on Frame04-2

In the Frame03 and Frame04-1 tests, the crushing of the shear ties was the cause of the first major load drop. With the thicker shear ties, the shear ties were no longer the weakest component. This forced the initial failure to occur in the C-frames instead of the shear ties within close vicinity of the impact area, thereby leading to a more localized failure in the frames. There were two displacement potentiometers centered in each loaded frame bay. However, both detached from the panel during impact so the relationship between the load to the skin was only available for the first major load drop. Figure 3.6.21 shows the total load relative to the skin displacement for Frame03, Frame04-1 and Frame04-2. For the Frame03 and Frame04-1 experiments, the composite shear ties crushed in the radius region, as observed by the small load drop at approximately 16 mm of skin displacement (black and gray plots in Figure 3.6.21). With the thicker aluminum shear ties, no crushing occurred in the radius region and the overall load carrying capability was
significantly higher. Frame fracture was the initial and final failure mode, and thus there was not energy-absorbing progressive failure observed in the previous tests.

Due to the loss of the potentiometers to accurately capture the skin displacement, Figure 3.6.22 is included to show the failure history as a function of adjusted actuator displacement. The adjusted actuator displacement is the total actuator displacement minus the distance required to close the hollow cavity of the cylindrical bumper. The load increased elastically until it reached a peak load of 126.8 kN, followed by an abrupt load drop of approximately 26 kN. The initial load drop was caused by failure of the frame material surrounding the loaded shear tie on the center frame.

**Figure 3.6.21**: Phase II Loading Histories
Figure 3.6.22: Frame04-2 Failure During Load

Figure 3.6.23 shows the initial failure of the center frame. The failure initiated in the bottom part of the frame flange at the edge of the shear tie. The crack roughly outlined the shape of the shear tie passing through the outermost fastener holes, as shown in the close up view in the top left corner of Figure 3.6.23. The crack then propagated through the remainder of the frame, in a shear dominated failure mode.

Once the center frame failed in shear, the load was redistributed to the other loaded Frames #2 and #4. The next load drop occurred at 100 kN in Figure 3.6.22, which indicated that Frame #4 failed in shear. The Frame #4 shear failure is described in Figure 3.6.24, shown by two photos (from the East and West viewing angle).

The final load drop in Figure 3.6.22 was caused by the final loaded frame failure in Frame #2. Figure 3.6.25 shows all three loaded frames failed in shear within close vicinity of the impact area. Debris can be seen flying off the specimen, including the label that was originally on Frame #2 and a portion of Frame #3 that
broke off during impact. It should be noted that even with the frames completely severed, the panel still was capable of sustaining a total load of approximately 70 kN. In absence of the direct load path to the frames, the load was transferred through the skin and adjacent shear ties to the portion of the C-frames that were intact adjacent to the impact zone.

Figure 3.6.26 shows the frames after impact. Each frame was completely severed during the impact event, but the frames rebounded to roughly the original shape and damage was limited to the area of impact.

Figure 3.6.27 shows a view of outer surface of panel after impact. There were some minor cracks near the edge of the panel, but the general shape of the test panel quickly relaxed to the original shape. The degree of cracking on the outer skin, near Frames #1 and #5, could have been less severe (possibly non-existent) if less actuator stroke was applied during the test. This is evident in Figure 3.6.22 where much more actuator displacement was applied following the failure of the final loaded frame (Frame #2).

Figure 3.6.23: Frame04-2 Initial Failure in Center Frame
Figure 3.6.24: Frame04-2 Failure of Frames #2 and #4 During Load

Figure 3.6.25: Frame04-2 Complete Failure of Frames During Load
Figure 3.6.26: Frame04-2 Damaged Frames After Impacts

Figure 3.6.27: Frame04-2 Failure During Load
3.6.5 Frame by Frame Comparison of Quasi-Static and Dynamic Tests

Since the two phases of experiments involved loading across two (Phase I quasi-static) and three (Phase II dynamic) frames, it was decided to compare these results on a per frame basis. A force per frame-displacement comparison for quasi-static indentation and dynamic impact (Frame01 vs. Frame03, Frame04-1 and Frame04-2) is shown in Figure 3.6.28. It should be noted that all panels described in this section were loaded on the skin spanning between stringers and that the displacement metric refers to the skin displacement measured by a potentiometer. However, in the Frame04-2 test the potentiometers were knocked off the specimen mid impact, so the data were unavailable. Instead, the adjusted actuator displacement (total actuator displacement minus the distance required to close the cylindrical bumper cavity) was used for this comparison. The loading histories for Frame03 and Frame04-1 were combined in order to represent the full loading stroke on an all-composite, pristine panel.

The initial per-frame stiffness of Frame03 and Frame04-1 (dynamic) was slightly higher than the initial stiffness of Frame01 (quasi-static) due to a smaller frame spacing in the larger Phase II panels (508 mm for Frame01 vs. 457 mm for Phase II panels). Because of the smaller frame spacing, the stiffness contribution of the non-loaded boundary frames was slightly larger due to the frames being closer to the impact area.

In the Frame01 quasi-static test, failure progression occurred within the vicinity of indentation. Also in Frame01, there were competing failure mechanisms between penetration of the C-frame through the stringers and cracking of the C-frames where the stringers contacted the C-frames. Thus, it can be stated that in the quasi-static test, there was local damage and deformation at the stringer-frame contact site and the slow loading speed allowed sufficient time for the load to redistribute through redundant load paths.
When Frame03 was impacted dynamically, the frames experienced a combined torsion+shear+bending failure close to the boundary conditions, and away from the impact area with none of the local penetration failures at the stringer-to-frame contact points which were observed in the quasi-static tests. This suggests the load transfer through the frames into the boundaries, and the resulting response leading to failure of the frame, is loading rate dependent. In both dynamic and quasi-static cases the stringer-frame interaction played a critical role in the damage evolution and both local and non-local (i.e., far away) damage developed. Therefore, for wide area blunt impact events, not only should the impact site be inspected but also the surrounding areas (i.e., where the frames join to other structures, such as the frame connection to the cargo or passenger floors).

Frame04-2 with the stronger aluminum shear ties experienced a higher load
per frame capacity than the panels with the thinner composite shear ties. Unlike the previous specimens with composite shear ties, the stronger aluminum shear ties were not the first component to fail, but instead provided a direct load path to the frames and prevented frame rotation. This forced a more localized failure in close vicinity to the impact region. Therefore, the shear ties played a key role in damage initiation and evolution since they are the structural elements that are directly loaded and transfer load from the impact site to the main load-bearing frames. Whether the shear ties fail (or lose load carrying capability) early or later in the impact event, affects the subsequent failure modes induced in the frames.

3.6.6 Energy Quantification

For the combined load history for the Frame04-1 and Frame03 experiments, the energy required to produce the final state of damage was 1811 J per frame. The energy was determined by calculating the area under the load history curve in Figure 3.6.29. This was the energy required per frame to cause the final damage state (broken shear ties and frames) of a five frame panel with composite shear ties. Two plots are shown in Figure 3.6.29, the combined loading history for Frame03 and Frame04-1 (shown by the solid line), and the impact energy as a function of skin displacement (shown by the dashed line). From the energy plot, the energy to cause intermediate damage (i.e., crushed loaded shear ties) can be determined. For example, it was found that an energy of 865 J per frame was required to completely crush the loaded shear ties. As the energy of the impact event increases, so does the severity of the damage to the impacted aircraft structure.

Recall from Section 3.2, a typical belt loader has a mass of approximately 3030 kg and a cargo loader can exceed 15000 kg. The belt loader traveling at 0.5 m/s would have a kinetic energy of 379 J and the cargo loader traveling at 0.5 m/s would have a kinetic energy of 1875 J. For the cargo loader traveling at 1 m/s the energy would be 7500 J. Therefore, based on energy balance, an impact event between a heavy cargo loader and an aircraft would likely lead to significant internal structural damage, even if the impact event was distributed over several frames.
3.7 Experimental Conclusions

Several large scale impact tests were completed in order to evaluate how damage initiation, damage evolution, and detectability are influenced by several parameters such as impactor material, impact location, impactor velocity and shear tie geometry and material. The conclusions listed below are the key findings of the experimental portion of this research investigation.

- GSE typical velocities were quantified and it was found that a realistic velocity within close vicinity of the aircraft is up to 0.5 m/s (~1.1 mph). If a GSE vehicle (especially a large cargo loader with a mass of 15000 kg or more) contacted an aircraft in the acreage area at this velocity, it would likely lead to significant internal structural damage. The damage could potentially be at locations of increased stiffness away from the impact area (i.e., where the frames connect to the passenger or cargo floors) and it would be important to inspect the surrounding structure as well as the impact area. This type of impact event would be very loud and the entire aircraft would...
move, increasing the probability of the event being reported to the correct authorities.

- Experimental observations showed that the bumper stiffness heavily influences the damage initiation, evolution and detectability. With a soft impactor, the contact pressures and interlaminar stresses in the skin are lower, allowing formation of widespread damage away from the impact site that is not necessarily visually detectable. The rubber material increased the contact area (approximately 155 cm\(^2\) for the rubber D-shaped bumper in comparison to 38 cm\(^2\) for the rigid indentor), and significantly reduced the contact pressures at the impact location. The rubber D-shaped bumper was a similar width (the width and length parameters are labeled in Figure 1.4.3), but the overall length was longer. Because the rubber conformed to the surface of the panel, the entire length essentially made contact with the panel. The rigid indentor did not deform, so only a small portion of the length actually contacted the panel. This allowed much higher contact forces to be applied without local damage, thereby leading to wide spread damage away from the impact location. Also, there was an effect of the edges of the bumper being much softer than the rigid indentor, thereby decreasing interlaminar shear stress at the periphery of the contact boundary.

- Large scale, dynamic impact tests have shown that the damage produced by wide area, blunt impact on composite panels is directly related to the geometry of the internal structural components. Specifically, for the impact location centered between stringers, the shear ties were found to be very influential in the damage initiation, damage evolution, and final damage location observed in the test panels. With a stiffer, more robust shear tie, the overall load carrying capability was found to be higher than the load carrying capability of the structure with the thinner composite shear ties. This was a result of two factors. First, the thicker shear ties did not fail and had a higher load carrying capability than the composite shear ties. Second, the thicker shear ties resisted frame rotation, which prevented loss of global stiffness due to rotation of the open channel of the composite C-frames. Also,
the thin composite shear ties were the weakest component and the first to fail under load, followed by extensive frame rotation and failure outside the impact area. The thicker shear ties forced a shear dominated failure in the frames at the impact area.

- A different loading history and damage state was observed for the Phase I quasi-static tests and Phase II large scale dynamic tests. When indented or impacted between stringers on a panel with composite shear ties, the shear ties were the first components to completely fail, which led to the first major load drop in both cases. However, the stringer-to-frame interaction was different between the dynamic and quasi-static tests. In the quasi-static tests, there were competing failure mechanisms between penetration of the C-frame through the stringer and cracking of the C-frame where the stringer contacted the C-frame. The stringers deformed locally at the contact location and intermediate load drops were observed as the frame incrementally rotated open in the Frame01 test. When Frame03 was impacted dynamically, the frames experienced a combined torsion+shear+bending failure away from the impact area with none of the local penetration failures at the stringer-to-frame contact points which were observed in the quasi-static tests. In the dynamic tests, the C-frames slid along the stringers, never allowing local deformation in the stringer at the contact point. The quasi-static tests experienced a static friction force (as opposed to the sliding friction force in the interaction observed in the dynamic tests) between the stringer and frame, the incremental load drops occurred when the C-frames rotated and overcame the static friction force. In both dynamic and quasi-static cases, the stringer-frame interaction played a critical role in the damage evolution and that both local and non-local (i.e., far away) damage developed.
4 Modeling

The analytical component of this dissertation focused on developing a methodology, using existing commercial FEA code, to predict the damage initiation and evolution for a wide area, blunt impact event. The methodology was then applied to investigate the effects of a wider range of parameters than what was evaluated experimentally. All FEA work in this dissertation used the commercial FEA codes Abaqus Standard and Abaqus Explicit. For damage initiation, quasi-static analysis and stiffness studies, Abaqus Standard was used. All failure predictions and dynamic impact simulations were conducted in Abaqus Explicit.

A FEA model (referred to as the Baseline model) identical to the experimental setup of the Frame03 and Frame04-1 tests was created in order to validate the methodology by comparing key metrics (e.g., force, displacement, skin surface strains) to the experimental results. Two studies were conducted using the developed methodology. Recall, a key component of this research topic is the detectability of a GSE impact event. Therefore, a detailed study that evaluated several hybrid aluminum-composite structures, primarily composite but with one major component of aluminum, was performed with a key metric of interest being the amount of final permanent deformation. A panel with the same geometry as the Baseline model but with all Aluminum 2024 material properties was evaluated, including the unload portion of the loading sequence, to determine the extent of the residual dent after impact.

The second major study that implemented the modeling methodology was a component thickness study. There were two parameters that were evaluated to determine their influence on detectability of the impact event: skin thickness, and stringer thickness. The thicker shear ties were evaluated experimentally, so that
particular study was not repeated analytically.

4.1 Methodology

This section describes the approach and FEA modeling methodology that was developed for this research topic.

4.1.1 Approach

The desired outcome of the methodology was to develop an approach that could accurately predict the failure initiation and evolution for the large dynamic test cases in Abaqus Explicit. Similar to the experimental approach, the modeling methodology was developed in stages, and correlated to experimental results in the process. The initial stiffness models to represent the quasi-static Phase I tests were analyzed in Abaqus Standard. These models had elastic material properties, with no failure parameters. The purpose of the quasi-static models was to give an accurate stiffness prediction when compared to the Phase I experiments.

Significant effort was focused on accurately predicting the initial failure of Frame04-1 and the failure evolution in the Frame03 panel, which is referred to as the “Baseline” FEA model. To achieve this objective, Hashin-Rotem failure initiation and evolution parameters were implemented to predict failure within the plies. The Hashin-Rotem failure criteria allowed the models to capture fiber tension, fiber compression, matrix tension, and matrix compression failure.

In the experiments, frame rotation significantly influenced the global stiffness as well as the stress state and damage evolution of the panel. Therefore, it was necessary to correctly represent the shear ties and their connection to the frames and skin in order to capture the frame rotation. The initial FEA models were defined as one single part and did not account for the fastener line that connected the shear ties to frames, instead the entire surface that made contact between the frame and shear tie was merged together. In Figure 4.1.1, the cross section of the frame to shear tie connection is shown and the dimension “D” defines the contact surfaces between the frame and shear tie. In the “Merged Part,” it can be seen that
the frame and shear tie behave as one continuous part under load. This approach created an overly stiff representation of the shear tie to frame connection and it did not allow for proper frame rotation. The merged part approach was acceptable for the initial stiffness prediction in the initial elastic range prior to frame rotation in the quasi-static models. However, this approach was not valid when predicting failure evolution because without the correct frame rotation, the panel stress state under significant impactor displacement was not correct.

The modeling approach was then adjusted to reduce the area that connected the shear ties to the frames, which is described by the “Separate Parts with Tie Constraints” cross section view in the right side of Figure 4.1.1. This modeling iteration was composed of one merged part, excluding the shear ties within close vicinity of the impact zone. The shear ties were defined by tie constraints between the shear tie and frame above the fastener line, as defined by the dimension “D” in Figure 4.1.1. This provided a smaller contact area between the shear ties and frames, which was more realistic of the experimental setup and allowed more frame rotation.

Even with a more compliant shear tie representation, the model still over predicted the initial failure load when compared to what was observed in the experiments. To address the overly stiff connection region, an effective fastener modeling technique was developed to accurately simulate the fastener line without modeling each individual fastener on the panel. The effective fastener modeling technique was then implemented in the larger dynamic models. The Baseline model correlated well with the experimental results and validated the modeling methodology and the effective fastener technique.
4.1.2 Script Based Model Definition

To build each FEA model individually in Abaqus CAE would be extremely time consuming due to the complex structural configuration of the panel. Each FEA panel model requires extensive partitioning and it would be easy to accidentally skip a partition, or accidentally create component geometry that did not properly match the connecting components. Both user errors would lead to analysis error, both in the form of contact errors and warped elements. To address this problem, and create many different models efficiently and consistently, a Python script used within Abaqus CAE was developed. The script-based model definition allows the study of a variety of parameters by alleviating the time (an potential errors) involved with manual model creation for each different geometry. A flow chart for the structure and implementation of the script is shown in Figure 4.1.2. The parameters were defined in the script, which was run within Abaqus CAE. Then Abaqus CAE was used to create an input file, that was run locally.
The results were then documented for each FEA model. The final output was a database of several impact cases that evaluated different parameters to determine their sensitivity to creating damage that not visually detectable.

**Figure 4.1.2: Python Script Flowchart**

Key parameters describing the geometry and layup of the test panel, as well as the impact event, were defined within the script. The script had the option of composite C-frames or aluminum flat frames to match the experimental setup. In Figure 4.1.3, the Baseline FEA model created with the Python script is shown. Frame #1 and Frame #5 were modeled as flat aluminum frames. The script allowed for static indentation or dynamic impact (with an option to create models with a constant velocity that matched the test protocol). For dynamic simulations, failure within the plies was modeled using Hashin-Rotem failure criteria (fiber tension, matrix tension, fiber compression or matrix compression). The Hashin-Rotem damage criteria was optional in the script and could easily be excluded. To reduce the computational cost and allow for a more refined global mesh, a half symmetric model was defined.
The mesh generation was also automatically defined in the script. Modeling composites in Abaqus is complicated, and requires correct element assignment, material orientation, and correct mesh stack direction. Due to the complex geometry of the structure, this is an extremely time consuming task to perform by hand because each individual cell in the model requires a stack direction assignment. In the five frame panel model, there were several hundred cells. Appendix A provides a detailed tutorial to properly define models of composite structures within Abaqus, including directions to properly assign composite laminate material orientations, shell element stack directions, and Hashin-Rotem failure parameters.

4.2 Model Description and Definitions

This section gives a description of the bumper and panel FEA model details, including the failure parameters for the larger scale FEA models.
4.2.1 Rubber Bumper Modeling

Modeling the full bumper geometry, including the process of the hollow cavity of the bumper collapsing on itself, presents some severe numerical stability challenges due to the self contact issues and the high amount of deformation the bumper experiences. In the experiments, no significant load was transferred to the panel (i.e., no damage was observed) until after the hollow cavity of the bumper was fully collapsed. Also, the bumper fully collapsed and conformed to the steel box beam that was supporting the bumper, which made this support box beam more influential on the impactor geometry than the compliant rubber bumper itself. Therefore, it was deemed unnecessary to model the bumper collapsing in order to capture the correct deformation and stress state of the panel under load. For all FEA studies in this dissertation the bumper was modeled as a solid rubber pad. The dimensions of the rubber pad used to model the bumper are shown in Figure 4.2.1. In the dynamic FEA models, a constant velocity was applied to the backside of the bumper over the same area as the box beam in the experiments, which is depicted in Figure 4.2.1. This allowed the rubber pad to deform around the loaded surface, in the same manner that the rubber bumper conformed to the box beam in the experiments. The modeled rubber pad is the same thickness as the collapsed rubber bumper observed in the experiments (i.e., twice as thick as the cylindrical bumper’s wall). There was good initial stiffness correlation between the experiments and the FEA models when the bumper was modeled as a flat pad, which verified that modeling the rubber bumper as a solid rubber pad is an acceptable approach for this impact problem.

The bumper material definition was defined as elastic isotropic with a Young’s modulus of 10.34 MPa, Poisson’s ratio of 0.49, and a density of 1384 kg/m$^3$. The elements used to model the bumper were elastic C3D8I elements with no failure parameters. The “I” in C3D8I refers to “incompatible modes,” which means that incompatible deformation modes are added internally to the elements so that the element does not exhibit an overly stiff bending behavior [43]. C3D8I is a full integration, linear (or first order) solid brick element with 8 nodes, one at each corner of the element. C3D8I elements were used in the bumper model to
reduce hour glassing. With a reduced element (e.g., C3D8R element in Abaqus), the stress is calculated at one integration point. This can lead to hour glassing, which occurs when the element distorts in such a way that the stress at the integration point is zero, which then causes uncontrollable distortion of the mesh. This mesh distortion was observed in some earlier trial dynamic models that used C3D8R elements (brick with reduced integration), so all models discussed in this dissertation use C3D8I elements in the bumper definition.

![Figure 4.2.1: Solid Rubber Pad Simulating Bumper, Including Dimensions](image)

**Figure 4.2.1**: Solid Rubber Pad Simulating Bumper, Including Dimensions

### 4.2.2 Panel Description

This section describes the material properties that were used in the FEA studies, as well as a description of the failure parameters for the panel.
Material Properties as Defined in Abaqus

Each ply was defined in the laminate editor of Abaqus/CAE. Tables 4.2.2 and 4.2.1 provide the values that were used to define the lamina elastic properties for the tape and fabric materials in Abaqus. To define lamina material properties in Abaqus/CAE, $E_3$, $\nu_{13}$, and $\nu_{23}$ are not required. Table 4.2.3 describes the Aluminum 2024 elastic-plastic properties that were used for the aluminum boundary condition frames and the aluminum component studies. The Aluminum 2024 material definition included plastic yielding parameters, but no failure was defined.

**Table 4.2.1**: X840/Z60 6K Plain Woven Fabric Lamina Elastic Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s Modulus:</td>
<td></td>
</tr>
<tr>
<td>$(E_{11})$ GPa</td>
<td>80.0</td>
</tr>
<tr>
<td>$(E_{22})$ GPa</td>
<td>80.0</td>
</tr>
<tr>
<td>Poisson’s Ratio:</td>
<td></td>
</tr>
<tr>
<td>$\nu_{12}$</td>
<td>0.06</td>
</tr>
<tr>
<td>Shear Modulus:</td>
<td></td>
</tr>
<tr>
<td>$(G_{12})$ GPa</td>
<td>6.5</td>
</tr>
<tr>
<td>$(G_{23})$ GPa</td>
<td>4.1</td>
</tr>
<tr>
<td>$(G_{13})$ GPa</td>
<td>5.1</td>
</tr>
<tr>
<td>Lamina Thickness:</td>
<td></td>
</tr>
<tr>
<td>Ply Thickness (mm)</td>
<td>0.208</td>
</tr>
<tr>
<td>Density:</td>
<td></td>
</tr>
<tr>
<td>Density (g/cm$^3$)</td>
<td>1.6</td>
</tr>
</tbody>
</table>

**Table 4.2.2**: X840/Z60 12k Unidirectional Tape Lamina Elastic Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s Modulus:</td>
<td></td>
</tr>
<tr>
<td>$(E_{11})$ GPa</td>
<td>168.2</td>
</tr>
<tr>
<td>$(E_{22})$ GPa</td>
<td>10.3</td>
</tr>
<tr>
<td>Poisson’s Ratio:</td>
<td></td>
</tr>
<tr>
<td>$\nu_{12}$</td>
<td>0.27</td>
</tr>
<tr>
<td>Shear Modulus:</td>
<td></td>
</tr>
<tr>
<td>$(G_{12})$ GPa</td>
<td>7.0</td>
</tr>
<tr>
<td>$(G_{23})$ GPa</td>
<td>3.7</td>
</tr>
<tr>
<td>$(G_{13})$ GPa</td>
<td>7.0</td>
</tr>
<tr>
<td>Lamina Thickness:</td>
<td></td>
</tr>
<tr>
<td>Ply Thickness (mm)</td>
<td>0.142</td>
</tr>
<tr>
<td>Density:</td>
<td></td>
</tr>
<tr>
<td>Density (g/cm$^3$)</td>
<td>1.6</td>
</tr>
</tbody>
</table>
Table 4.2.3: Aluminum 2024 Material Properties

<table>
<thead>
<tr>
<th>Material Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young’s Modulus: ( E ) (GPa)</td>
<td>73.1</td>
</tr>
<tr>
<td>Poisson’s Ratio: ( \nu_{12} )</td>
<td>0.33</td>
</tr>
<tr>
<td>Strength: Tensile Yield Strength (MPa)</td>
<td>324</td>
</tr>
<tr>
<td>Ultimate Tensile Strength (MPa)</td>
<td>469</td>
</tr>
<tr>
<td>Elongation at break</td>
<td>20%</td>
</tr>
</tbody>
</table>

Element Definition

There are two types of shell elements that can be used to define a structure in Abaqus. Conventional shell elements are defined on one single plane (i.e., no thickness is modeled in the geometry) and the thickness is defined by the section properties. The conventional shell elements have displacement and rotational degrees of freedom at the nodes.

The second type of shell elements are continuum shells. For continuum shell elements, the thickness is defined by the actual geometry and there are only displacement degrees of freedom. Figure 4.2.2 shows how a structure would be modeled with each type of shell element.

For this research, continuum shell elements were used to model the panel structure. Each FEA panel model had a very complex geometry, and the geometry of each component was directly dependent on the geometry of the other components (e.g., the outer radius of the shear tie flange is dependent on the skin radius and skin thickness). Each component was defined individually, then merged together to form one continuous, more complex part. For this research, quadrilateral continuum shell elements (SC8R, reduced integration with hourglass control) were used to define the panel.
Hashin-Rotem Failure Parameters

To define failure within the ply, Hashin-Rotem failure parameters were implemented. Hashin-Rotem failure is the standard approach used for modeling material failure in composites within Abaqus. The four modes of failure are: fiber compression, fiber tension, matrix compression, and matrix tension. Shear failure contributes to matrix tension or compression failure, but is not defined as an individual failure mode within Abaqus. The material failure initiation based on an effective stress tensor $\hat{\sigma}$, shown in Equation 4.2.1, which accounts for damage to the material. $M$ is the damage operator and $\sigma$ is the true stress (i.e., the stress the actual structure would experience). In the description of the damage operator in Equation 4.2.2, $d_f$, $d_m$, and $d_s$ are the damage variables for fiber, matrix and shear. If all of the damage variables have a value of 0, which corresponds to no damage initiation, the effective stress would equal the true stress. As a composite laminate undergoes damage, softening would occur and there would be less material to resist the internal forces. Failure in one mode would significantly influence the other failure modes, which is accounted for in the damage operator. Abaqus uses the damage operator to artificially increases the stress used to determine the
failure initiation for each mode based on whether failure initiation has been met 
in other modes. More details about how the damage variables are calculated are 
described in detail in the Abaqus Manual Section 23.3.2 “Damage initiation for 
fiber-reinforced composites” [43].

\[
\hat{\sigma} = M\sigma
\]  

(4.2.1)

\[
M = \begin{bmatrix}
\frac{1}{(1-d_f)} & 0 & 0 \\
0 & \frac{1}{(1-d_m)} & 0 \\
0 & 0 & \frac{1}{(1-d_s)} \\
\end{bmatrix}
\]  

(4.2.2)

The stress \(\hat{\sigma}_{11}\) indicates the effective stress in the longitudinal direction of 
the ply (i.e., fiber direction in the tape material), \(\hat{\sigma}_{22}\) is the effective stress component 
in the transverse direction of the ply, and \(\hat{\tau}_{12}\) is the in-plane effective shear 
stress within the ply. The failure initiation is defined by the following equations:

- Fiber tension (\(\hat{\sigma}_{11} \geq 0\)): 
  \[
  F^f_t = (\frac{\hat{\sigma}_{11}}{X_T})^2 + \alpha \left(\frac{\hat{\tau}_{12}}{S_L}\right)^2
  \]

- Fiber compression (\(\hat{\sigma}_{11} < 0\)): 
  \[
  F^c_f = (\frac{\hat{\sigma}_{11}}{X_C})^2
  \]

- Matrix tension (\(\hat{\sigma}_{22} \geq 0\)): 
  \[
  F^t_m = (\frac{\hat{\sigma}_{22}}{Y_T})^2 + \left(\frac{\hat{\tau}_{12}}{S_T}\right)^2
  \]

- Matrix compression (\(\hat{\sigma}_{22} < 0\)): 
  \[
  F^c_m = (\frac{\hat{\sigma}_{22}}{Y_C})^2 + \left(\frac{\hat{\tau}_{12}}{S_T}\right)^2
  \]

- Hashin - Rotem (1973): 
  \(\alpha = 0, S^T = \frac{Y_C}{2}\)

- Hashin (1980): \(\alpha = 1\)

\(X_T\) longitudinal tensile strength  
\(X_C\) longitudinal compressive strength  
\(Y_T\) transverse tensile strength  
\(Y_C\) transverse compressive strength  
\(S^L\) longitudinal shear strength (1-2 direction)  
\(S^T\) transverse shear strength (2-3 direction)  
\(\alpha\) coefficient that determines the contribution of the shear stress 
to the fiber tensile initiation criterion
For this research, Hashin-Rotem failure criteria was used ($\alpha = 0$). By using Hashin-Rotem failure parameters, the fiber failure initiation equation is simplified and the transverse shear strength is calculated based on the transverse compressive strength and does not need to be defined. Figure 4.2.3 shows a plot of the material behavior for each of the failure modes, including failure initiation and failure evolution. In Abaqus, the constitutive law is defined as a stress-displacement relationship for each of the four failure modes. The material stiffness and strength properties are defined, and from that the failure initiation ($\sigma_{\text{fail}}$ term in Figure 4.2.3) for each mode and corresponding displacement ($\delta_{\text{fail}}$ term in Figure 4.2.3). The failure initiation occurs when the effective stress equals the failure criteria for one of the four possible modes (fiber tension, fiber compression, matrix tension or matrix compression), and is depicted by point “A” in the stress displacement history.

Failure evolution was included in the dynamic FEA models described in this dissertation. In order to account for failure evolution and material degradation, the longitudinal tensile fracture energy, longitudinal compressive fracture energy, transverse tensile fracture energy, and transverse compressive fracture energy were defined. The fracture energy for each failure mode corresponds to the area under the loading curve labeled “Damage evolution energy,” shown in Figure 4.2.3. Once the failure initiation value is exceeded, the material stiffness is linearly degraded. The final equivalent displacement and slope of the material degradation is dependent on the defined fracture energy. If the partially damaged laminate is unloaded (point “B” in Figure 4.2.3), it follows a linear path back to the origin, which is the degraded material stiffness. Section 23.3.3 “Damage evolution and element removal for fiber-reinforced composites” of the Abaqus Manual describes how Abaqus implements damage evolution in detail [43].

This approach to modeling material damage initiation and failure evolution is based on the work by Hashin and Rotem [44, 45]. Tables 4.2.4 and 4.2.5 contain the Hashin-Rotem failure details used in the models described in this dissertation.
Figure 4.2.3: Hashin Failure Degradation [43]

Table 4.2.4: X840/Z60 12k Unidirectional Tape Lamina Hashin-Rotem Failure Properties

<table>
<thead>
<tr>
<th>Failure Initiation:</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal Tensile Strength, $X^T$ (MPa)</td>
<td>2799.3</td>
</tr>
<tr>
<td>Longitudinal Compressive Strength, $X^C$ (MPa)</td>
<td>1620.3</td>
</tr>
<tr>
<td>Transverse Tensile Strength, $Y^T$ (MPa)</td>
<td>57.2</td>
</tr>
<tr>
<td>Transverse Compressive Strength, $Y^C$ (MPa)</td>
<td>227.5</td>
</tr>
<tr>
<td>Longitudinal Shear Strength, $S^L$ (MPa)</td>
<td>75.8</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Failure Evolution Energy:</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal Tensile Fracture Energy (m-kN/m^2)</td>
<td>100.0</td>
</tr>
<tr>
<td>Longitudinal Compressive Fracture Energy (m-kN/m^2)</td>
<td>100.0</td>
</tr>
<tr>
<td>Transverse Tensile Fracture Energy (m-kN/m^2)</td>
<td>0.2</td>
</tr>
<tr>
<td>Transverse Compressive Fracture Energy (m-kN/m^2)</td>
<td>0.2</td>
</tr>
</tbody>
</table>
Table 4.2.5: X840/Z60 6K Plain Woven Fabric Hashin-Rotem Failure Properties

<table>
<thead>
<tr>
<th>Failure Initiation:</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal Tensile Strength, $X^T$ (MPa)</td>
<td>992.8</td>
</tr>
<tr>
<td>Longitudinal Compressive Strength, $X^C$ (MPa)</td>
<td>772.2</td>
</tr>
<tr>
<td>Transverse Tensile Strength, $Y^T$ (MPa)</td>
<td>992.8</td>
</tr>
<tr>
<td>Transverse Compressive Strength, $Y^C$ (MPa)</td>
<td>772.2</td>
</tr>
<tr>
<td>Longitudinal Shear Strength, $S^L$ (MPa)</td>
<td>77.9</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Failure Evolution:</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal Tensile Fracture Energy (m-kN/m^2)</td>
<td>100.0</td>
</tr>
<tr>
<td>Longitudinal Compressive Fracture Energy (m-kN/m^2)</td>
<td>100.0</td>
</tr>
<tr>
<td>Transverse Tensile Fracture Energy (m-kN/m^2)</td>
<td>100.0</td>
</tr>
<tr>
<td>Transverse Compressive Fracture Energy (m-kN/m^2)</td>
<td>100.0</td>
</tr>
</tbody>
</table>

4.2.3 Effective Fastener Modeling

The initial modeling approach used tie constraints between the frame and shear ties, neglecting any type of fastener modeling, as mentioned in Section 4.1.1. This approach over-predicted the initial failure load carrying capability and did not allow the frames to properly rotate after initial failure. It was found from the experiments accurately capturing the frame rotation is essential to correctly predicting the failure history of a monolithic composite panel under loading. Therefore, it was necessary to account for the fasteners.

There are two approaches to modeling fasteners in Abaqus, either by modeling each fastener as a connector element or rigid beam multi-point constraint. Connector elements can be rigid or have elastic and failure properties defined, and the behavior of the fastener can be evaluated. The second approach uses a rigid beam multi-point tie constraint between two surfaces, which essentially forces the rotation and displacement at the fastener location to be equal between the two tied surfaces. For both methods, the fastener location can be chosen independent of the mesh, and the connection is defined based on the average displacement and rotation of the nodes surrounding the fastener location. However, if the point-based fastener is modeled as a rigid connection and two adjacent fasteners share one or more nodes, the model will be over constrained, leading to singularities. Modeling the fasteners also significantly increases the computational cost[43, 46].

For this research topic since bending failure in the composite always oc-
curred through the fastener line, the desired outcome was to accurately predict the global response and failure history of the panel, rather than the behavior of each individual fastener. In order to do so, an effective fastener modeling technique was developed and is described in this section. The objective was to create a coupon FEA model with a strip of reduced strength along the fastener line that would produce a similar failure behavior as the notched FEA coupon model. This concept is described by the sketch in Figure 4.2.4.

![Figure 4.2.4: Reduced Strength Along Fastener Line in Effective Fastener Modeling Technique](image)

The loaded shear ties on the Frame03 test panel experienced a combination of compression and bending due to the rotation of the open C section of the frames. The adjacent shear ties were primarily loaded in a pull off manner due to the rotation of the frames. Each shear tie had six fasteners, so a FEA model of a laminate (with the same layup and fastener diameter and spacing) with six open holes under pure bending (by means of a four point bend model setup) was evaluated in Abaqus Standard. The stress distribution was the same for each open hole, so a four point bend coupon model of a single hole, with symmetry boundary conditions on the non loaded sides, was used to develop the effective fastener modeling technique. The four point bend setup is shown in Figure 4.2.5. A load ("F" in Figure 4.2.5) was applied at the end of the model, with a roller type boundary condition applied a distance "a" from the loaded end. A half symmetric constraint was applied the boundary with the open hole. The fastener diameter
was 6.35 mm, the width of the coupon was 31.75 mm, and the thickness was 2.5 mm (same thickness as the composite shear ties).

![Four Point Bend Coupon Setup](image)

**Figure 4.2.5:** Four Point Bend Coupon Setup

The first step was to determine a stress concentration factor (SCF) for the fastener hole under bending, which was then factor was then used to reduce the strength properties. To determine the stress concentration factor, a notched and an unnotched four point bend coupon FEA model with elastic isotropic properties, were evaluated. A very small load (F in Figure 4.2.5) of 2.22 N was used in order to prevent any nonlinear geometry effects when determining the stress concentration factor. The stress concentration factor was the ratio of the stress in the principle direction (x direction in Figure 4.2.5) at the bolt hole to the peak stress in the principle direction in the unnotched isotropic specimen, both stress values were taken from the top surface (i.e., the surface under tension). The stress concentration factor was found to be 1.87 for this particular fastener hole diameter and spacing.

Once the stress concentration factor was determined, a coupon FEA model with the same dimensions as the notched coupon (excluding the fastener hole) with a strip of reduced strength, as shown in Figure 4.2.6. The width of the strip was determined based on the geometry of components in the global panel. The
shear ties on the global panel model were already partitioned 8.89 mm below the fastener line (in order to accommodate the geometry defined in the C-frames), so the width of the strip with reduced strength properties was chosen to be the same. The goal was to predict a similar load displacement history in a reduced strength coupon model when compared to the notched FEA model. This was an iterative approach and the load displacement histories for several cases are compared in Figure 4.2.8. A load was applied to the loading end indicated in Figure 4.2.5, which was compared to the displacement of the loaded end.

**Figure 4.2.6**: Four Point Bend Coupon Setup

The FEA models had a layup that was identical to the shear ties $[45/0]_{3S}$ of all fabric material. The laminate with labeled individual plies is shown in Figure 4.2.7. Figure 4.2.8 shows the load displacement history for the notched FEA model, labeled “Notched Model,” as well as several iterations of the reduced strength equivalent coupon model. The notched model failed along the fastener line at a load of 0.911 kN and a displacement of 68.5 mm, which can be seen by the sudden increase in displacement in Figure 4.2.8. The notched model served as a baseline for the rest of the studies.
An unnotched coupon model with no reduced strength properties was also evaluated. The model with no reduced strength properties is shown by the dashed line in Figure 4.2.8. Without reduced strength properties, no failure initiated at the effective fastener line. In the panel model, without failure along the fastener line, the frame rotation was restricted.

Next, the strength was reduced for all plies in the laminate. The first iteration with reduced strength properties was a laminate with reduced strength through the thickness, in essence, the material strength properties of each ply shown in Figure 4.2.7 was reduced by a SCF of 1.87. The laminate failed at the effective fastener strip, but at a lower load and displacement than the “Notched Model”.

Additional cases were evaluated in which the strength properties were only reduced in the outer plies while leaving the center plies at full strength. Finally, the desired failure was somewhere between the model with reduced strength on only one outer ply on either side (plies 1 and 12 in Figure 4.2.7) and the outer
two plies on either side (plies 1, 2, 11 and 12 in Figure 4.2.7). Therefore, the next step was to reduce the strength by a lower SCF of 1.435 on plies 2 and 11, with the strength properties reduced by 1.87 for the outer plies 1 and 12 (as labeled in Figure 4.2.7). The SCF of 1.435 was chosen iteratively via parametric variation until match-up was determined. The coupon model with plies 1 and 12 reduced by a SCF of 1.87 and plies 2 and 11 reduced by a SCF of 1.435 failed at a load of 0.79 kN and displacement of 70.42 mm.

![Graph showing failure history for equivalent fastener models](image)

**Figure 4.2.8: Failure History for Equivalent Fastener Models**

The last iteration of the reduced strength model provided a strong correlation to the notched coupon failure prediction, within 3% of the displacement and 13.4% of the failure load. Because the goal was to accurately allow the frames to rotate, matching the deformation was a higher priority than the load correlation. This modeling technique was then implemented at the fastener lines where the shear ties connect to the skin in the full panel models, as shown by the view of the shear tie to frame connection in Figure 4.2.9.
In the panel model, no additional geometry refinements were required in order to implement the effective fastener modeling technique because the geometry of each component was defined individually, then the components were assembled and merged or tied together. The material strip with reduced strength properties was defined in the shear tie in the Part Module of Abaqus/CAE once, then automatically implemented in the shear ties within close vicinity of the impact area in the FEA model.

![Tie Constraints and Reduced Material Strength in Shear Ties](image)

**Figure 4.2.9**: Tie Constraint and Reduced Material Strength in Shear Ties, Implemented in Baseline FEA Model

### 4.3 Mesh Sensitivity Study

A mesh sensitivity study was performed on a three frame, four stringer panel to determine an acceptable global mesh size and refined mesh size in the critical shear ties. It was important to use a global mesh size that was small enough so it would not influence the structural response, i.e., create a response that was overly stiff. The shear ties played a critical role in the damage initiation and evolution, both by influencing the failure within the shear tie and the frame rotation. For this reason, the shear ties in critical locations had a refined mesh size.
The mesh sensitivity study simulated a three frame, four stringer panel impacted between stringers and across one frame with a rigid impactor. For these models, Abaqus Standard was used and no failure parameters were defined. This was essentially a stiffness study, so elastic models were sufficient. For the global stiffness study, three global mesh sizes were used: 38, 25, and 19 mm, as shown in Figure 4.3.1. No shear tie refinement was implemented in the global stiffness models. Figure 4.3.2 shows the force vs. displacement history for the global mesh sensitivity study. There was good convergence of global stiffness for the 25 mm and 19 mm models in the initial elastic range. The complex geometry of the components created additional mesh size limitations. A global mesh size of 25 mm was acceptable because convergence of the load and displacement relationship was achieved between the 19 and 25 mm model, however it was decided to use a global mesh size of 19 mm in order to prevent excessively warped elements in tightly curved regions (e.g., stringer hat radius). A warped element would have a poor aspect ratio (significantly different length and width dimensions) which can reduce the element’s accuracy.

Figure 4.3.1: Global Mesh Sensitivity Study, Panels with Various Global Mesh Size During Indentation
Figure 4.3.2: Force Displacement History for the Global Mesh Sensitivity Study

The refined mesh size of the shear ties was critical to accurately predict the initial load drop and correct frame rotations. Three models with refined shear tie mesh sizes were evaluated: 19, 13, and 6 mm (with a global mesh size of 19 mm for all models). The model used for the refined mesh size study had one shear tie (with a refined mesh size) directly under the indentor that was connected to the skin and frame via tie constraints, which is shown in Figure 4.3.3. The refined mesh size did not significantly influence the global stiffness, as shown by the global force vs. displacement history for the three models in Figure 4.3.4. The stress (S22) contour plot of the shear tie for the three models is shown in Figure 4.3.5. The S22 stress was the principle material direction of the bottom ply (45 degree material orientation) in the shear tie laminate (the bottom ply was the surface that contacted the frame), as depicted in 4.3.6. The peak stresses (S22) in the shear ties with a refined mesh size of 13 and 6 mm are equivalent in the elastic
range up to an indentor displacement of approximately 15 mm. The shear ties are a key element to the failure process, therefore, a refined mesh size of 6 mm was chosen in order to have elements with similar length and width dimensions in the radius region.

*Figure 4.3.3: Panel Used for Shear Tie Mesh Sensitivity Study*
Figure 4.3.4: Force Displacement History for the Shear Tie Refined Mesh Sensitivity Study

Figure 4.3.5: Stress (S22) Contour Plot of Refined Shear Tie at 10 mm Bumper Displacement
Figure 4.3.6: Peak Stress (S22) in Shear Tie for Refined Mesh Size Study

4.4 Finite Element Analysis Validation

Validating the FEA modeling methodology was a major component of this research project and is a critical first step before conducting the later studies. This section gives a detailed explanation of the FEA results in comparison to the experimental results detailed in Chapter 3. The modeling methodology described in the previous pages was fully implemented in the Baseline FEA model, described in Section 4.4. The initial failure load predicted by the Baseline FEA model was within 3% of the first major load drop in the Frame04-1 test and the order that the components were predicted to fail in the Baseline FEA model matched well with the failure evolution observed in the Frame03 experiment.
Large Panel Baseline FEA Model

The Baseline FEA model for the following studies was a five-frame panel FEA model, impacted by a rubber bumper over three frames. The panel definition was identical to Frame03 and Frame04-1 test panels. In order to incorporate failure initiation and evolution within plies, Abaqus Explicit was used. The geometry of the individual components in the Baseline FEA model was identical to the experimental setup. The Baseline FEA model was a half symmetric model with rotational stiffness boundary conditions on the frame ends which were free to translate at the frame boundary supports. Figure 4.4.1 shows the panel and impactor in the FEA Baseline model. The impactor material was a solid rubber pad, 1016 mm in length, with a Young’s Modulus of 10.34 MPa, as described in Section 4.2.1. The cross section dimensions are shown in Figure 4.2.1 of Section 4.2.1. The baseline applied displacement of the back of the bumper was 76.2 mm, with a constant velocity of 0.5 m/s, identical to the dynamic tests. The effective fastener modeling technique (described in Section 4.2.3) was included in the shear ties of the Baseline FEA model, as indicated in the Figure 4.4.1. This section describes the correlation of the Baseline FEA model to the Phase II test data (Frame03 and Frame04-1), which is summarized in Table 4.4.1.

Figure 4.4.2 shows the total load history of the combined Frame03 and Frame04-1 loading sequences in comparison to the Baseline FEA model. The initial stiffness of the experiments, up to 7.7 mm skin displacement and load of 40 kN, was correctly predicted by the Baseline FEA model. In the experiments there was an intermediate failure, indicated by the small load drop at a skin displacement of 16.0 mm and 64.1 kN of the Frame03 and Frame04-1 combined load data, that was not predicted by the FEA Baseline model. This small failure was likely caused by interlaminar shear and crushing in the radius region of the loaded shear ties. This intermediate load drop was not captured by the shell elements in the Baseline FEA model because the shell elements with Hashin-Rotem failure parameters can only predict failure within the ply, not failure between plies (i.e., delamination). However, the initial peak load of the Baseline FEA model was within 3% of the experimental results of Frame04-1. This large initial load drop observed in both
the Baseline FEA model and the experimental data was caused by complete failure of the shear ties directly under the impactor.

**Table 4.4.1: Baseline FEA Model vs. Experimental Results**

<table>
<thead>
<tr>
<th>Event</th>
<th>Experiment</th>
<th>Baseline FEA</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Disp. (mm)</td>
<td>Load (kN)</td>
</tr>
<tr>
<td>Initial radius damage in shear ties</td>
<td>16.0</td>
<td>64.1</td>
</tr>
<tr>
<td>Peak load prior to shear tie failure</td>
<td>33.7</td>
<td>91.6</td>
</tr>
<tr>
<td>First major initial load drop (loaded shear tie failure)</td>
<td>33.7</td>
<td>91.6</td>
</tr>
<tr>
<td>Second peak load</td>
<td>64.7</td>
<td>73.5</td>
</tr>
<tr>
<td>Frames failure near boundaries</td>
<td>85.2 (severed)</td>
<td>60.9</td>
</tr>
</tbody>
</table>

**Figure 4.4.1: Baseline Test Panel**
The initial failure that occurred in the FEA model was crushing of the shear ties directly under the impactor, leading to the initial load drop depicted by the gray line in Figure 4.4.2. The panel carried a total load of 94.5 kN before the loaded shear ties completely severed, followed by a load drop of 54.5 kN. The loaded shear ties on Frame #2 and Frame #3 experienced a combined bending and compression loading prior to failure, and buckled in two locations in the web region at 94.5 kN. Because the bumper terminated at the edge of the shear tie flange on Frame #4, the skin deformed around the edge of the soft bumper, and provided less resistance to frame rotation. Therefore, due to this rotational compliance, the loaded shear tie on Frame #4 experienced less bending stress than the loaded shear ties on Frame #2 and Frame #3 and failed primarily in compression near the radius region. The shear tie loading prior to shear tie failure is depicted in Figure 4.4.3. Figure 4.4.4 shows the Hashin-Rotem fiber compression failure initiation criteria for the three loaded shear ties in the Baseline FEA model after the shear...
ties failed at a load of 44.5 kN and skin displacement of 23.9 mm. Without the primary load path of the shear ties, the load was transmitted through the stringers to the frames by direct contact between the stringers and frames.

Once the center shear ties failed in the FEA model, there was a jagged increase in load as the displacement of the bumper increases. The small, intermediate load drops in the load history are due to additional, abrupt rotation of the C-frames and dynamic effects in the simulation (noise caused by the energy release as the material failed due to Hashin-Rotem damage evolution). The peak load reached in the FEA model was 106.9 kN, at 61.5 mm of skin displacement in Figure 4.4.2, which was higher than the experimental results from Frame03, Loading L2. Because the shear ties in Frame03 were damaged in the initial loading, it could have resulted in a lower overall load carrying capability for the remainder of the loading cycle in the second loading sequence of Frame03. Frame04-1 was only loaded to a displacement intended to fail the center three shear ties but not damage the surrounding structure. This was necessary to allow Frame04 to be retested with the thicker aluminum shear ties, so it did not include the full loading.

**Figure 4.4.3: Bending in Loaded Shear Tie**
After the loaded shear ties failed, the adjacent shear ties in the center three frames formed hinges and failed along the equivalent fastener line, which allowed additional C-frame rotation and reduced the load carrying capability of the panel. Figure 4.4.5 shows Hashin-Rotem fiber compressive failure initiation criterion for the panel at a skin displacement of 37.1 mm and load of 64.7 kN. It can be seen from the figure that the loaded shear ties have failed and a hinge has formed in the adjacent shear ties along the effective fastener line. The shear ties formed hinges in a specific order for both the Baseline FEA model and the Frame03 experiment, which is summarized in Table 4.4.2. The first shear tie to fail along the effective fastener line (i.e., form a hinge) was the adjacent shear tie on Frame #3, followed by the adjacent shear tie on Frame #2, and finally the adjacent shear tie on Frame #4.

The hinge in the adjacent shear ties initiated from the side of the shear tie...
closer to the impact area, which then propagated through the shear tie along the effective fastener line. There was also crushing in the radius region of the shear tie on the side of the shear tie near the impact area. As an example of the failure progression of the adjacent shear ties, Figure 4.4.6 shows the failure progression of the adjacent shear tie on Frame #3 for both the FEA model and the Frame03 test. The labels FEA1 and FEA2 correspond to the Baseline FEA model, which show the Hashin-Rotem fiber compression failure initiation criteria. The labels Exp1 and Exp2 correspond to still images of the adjacent shear tie on Frame #3 extracted from the Frame03 (second loading) high speed video. The exact load and displacement values for the shear tie images in Figure 4.4.6 are summarized Table 4.4.3.

**Figure 4.4.5:** Impact After Center Shear Ties Fail, Skin Displacement to ~20 - 60 mm
Table 4.4.2: Adjacent Shear Tie Failure History, Baseline FEA Model vs. Experimental Results

<table>
<thead>
<tr>
<th>Event</th>
<th>Frame03 Experiment</th>
<th>Baseline FEA Model</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Disp. (mm)</td>
<td>Load (kN)</td>
</tr>
<tr>
<td>Failure along effective fastener line in adjacent shear tie on Frame #3</td>
<td>64.4</td>
<td>63.5</td>
</tr>
<tr>
<td>Failure along effective fastener line in adjacent shear tie on Frame #2</td>
<td>65.6</td>
<td>64.7</td>
</tr>
<tr>
<td>Failure along effective fastener line in adjacent shear tie on Frame #4</td>
<td>67.5</td>
<td>53.0</td>
</tr>
</tbody>
</table>

Once the adjacent shear ties failed along the effective fastener line in the Baseline FEA model, the frames underwent significant frame rotation. The FEA model was defined to stop at a bumper displacement of 76.2 mm. At this displacement, the C-frames experienced failure initiation and element degradation in the same location as the frames in the Frame03 test, away from the impact site near the reaction points. If further displacement had been defined, it is expected complete frame failure would have been predicted through the entire cross section of the frame in the FEA model, similar to what was observed in the experiments. Figure 4.4.7 shows an image of the failed frames near the boundary after the Frame03 test compared to the predicted compressive failure initiation (Hashin-Rotem failure initiation criteria) in the Baseline FEA model (at 76.2 mm of applied bumper displacement), which indicates failure in the frames near the boundary was predicted by the Baseline FEA model.

Table 4.4.3: Load and Displacement Values Corresponding to Figure 4.4.6 Labels

<table>
<thead>
<tr>
<th>Figure ID</th>
<th>Disp. (mm)</th>
<th>Load (kN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>FEA1</td>
<td>28.9</td>
<td>61.1</td>
</tr>
<tr>
<td>FEA2</td>
<td>46.4</td>
<td>75.8</td>
</tr>
<tr>
<td>Exp1</td>
<td>62.7</td>
<td>73.3</td>
</tr>
<tr>
<td>Exp2</td>
<td>64.4</td>
<td>63.5</td>
</tr>
</tbody>
</table>
Figure 4.4.6: Center Frame, Adjacent Shear Tie Load Description History
This section details the skin strain comparison from the Baseline FEA model and the Frame03 experimental results. The strain predicted in the skin between Frame #2 and Frame #3 of the Baseline FEA model correlated well to the Frame03 experimental results (Loading L2). Figure 4.4.8 shows the element in which the strain data of the Baseline FEA model was extracted. The strain gauge was oriented perpendicular to the stringers on the top surface of the skin (i.e., not the surface impacted by the bumper). Figure 4.4.9 shows the skin strain for the Baseline FEA model and skin strain of Frame03 Loading L2 as a function of time. In the Frame03 test, after the bumper contacted the panel, the skin experienced a strain up to approximately 750 \( \mu \varepsilon \) until the bumper fully collapsed. Once the bumper collapsed, the strain in the skin began to dramatically increase, as seen by the change in slope of the gray line in Figure 4.4.9. The Baseline FEA model used a solid rubber pad, so there was not a change in the slope of the strain due to the collapsing of the bumper. Recall, Frame03 had some pre-existing damage caused by Loading L1, so the peak load prior to failure of the loaded shear ties (and corresponding strain in the skin) was lower than the pristine Frame04-1 test. The Frame03 results were used instead of the Frame04-1 results to compare the strain values in the skin because it provided insight for the entire load history, not just the initial failure of the shear ties.
Figure 4.4.8: Element for Strain Correlation in Skin

Figure 4.4.9: Strain in Skin Correlation between Frame03 and Baseline FEA
Peak Skin Contact Pressures

Contact pressure on the impacted surface of the skin was considered, as this could be related to visible surface damage formation. The rubber material of the bumper significantly reduced the contact stresses between the impactor and the skin when compared to what would be expected by contact with a rigid impactor. Figure 4.4.10 shows where in the load history the peak contact pressures occurred. In Figure 4.4.11, the contact pressure before damage in the shear tie occurred is depicted. The peak pressure of 2.27 MPa was at a load of 94.5 kN and skin displacement of 21.6 mm under the shear ties. The peak pressure was directly under the shear ties, indicating the undamaged direct load path through the shear ties. The stiff shear ties led to the peak skin stresses at these locations, with a fairly even distribution between each frame.

Figure 4.4.12 shows the pressure distribution after the shear ties are completely severed, at a load of 96.5 kN and 62.6 mm of skin displacement. The peak contact pressure reached was 2.31 MPa. This contact pressure distribution indicates a different type of load transfer. At this point, the load was primarily transferred to the frames through the stringer to frame contact points for each loaded frame. There were no pressure concentrations where the shear ties used to be, indicating this was no longer a load path after the shear ties were damaged. The peak contact stresses in the skin were then caused by the biaxial bending in the skin where the bumper terminated, rather than any high stiffness locations of the impacted structure. For both stress cases, the peak pressures were far lower than the matrix material strength, so no cracking in the skin was expected to occur due to the contact between the bumper and the panel.
Figure 4.4.10: Peak Skin Pressures in Load History

Figure 4.4.11: Contact Pressure Before Shear Tie Damage
Skin Failure Prediction

Figure 4.4.13 shows the Hashin-Rotem failure initiation criteria at the peak bumper displacement (76.2 mm), in the direction indicated by the red arrow, for the bottom surface of the skin. No failure in the impact zone (as indicated by the white dashed line) was predicted, but some failure initiation near the edges of the panel in the skin was predicted, although none occurred during the tests.
4.5 Panel Geometry and Material Parameter Effects Studies

This section discusses studies that apply the FEA modeling methodology to explore additional material and geometry parameters that were not evaluated experimentally. There are several parameters that influence the detectability of a high energy, wide area, blunt impact event. These include but are not limited to impactor material stiffness, component geometry (e.g., shear tie thickness), global panel geometry (e.g., frame arc length between supports) and impact location (on the skin between stringers vs. centered on a stringer). This section of the dissertation addresses the bumper material stiffness effects, a frame by frame stiffness comparison, an aluminum component material study and a component thickness study. The FEA models evaluated are described in Table 4.5.1.

The GSE threat was broadly varying and the potential contact area produced by GSE ranges from a discrete D-bumper (approximately 155 cm\(^2\) when flattened) to a long cylindrical bumper of varying sizes ( \(> 2000\) cm\(^2\) for a 1.2 m long bumper). Therefore, it was possible to contact across several frames, just one or two frames, or not any frames at all (i.e., discrete impactor between frames). It was useful to understand the relationship between contacting only a few or several frames in order to evaluate the results on a frame by frame basis.

The material stiffness of the impactor influenced the contact pressures, bending strain in the skin, and overall structural response. A soft compliant bumper produced more skin deformation, which led to higher bending stresses in the skin. However, a rigid impactor caused higher contact stresses at the impactor periphery as well as locations of higher stiffness (i.e., under the shear ties). The impactor material stiffness was very influential to the detectability of the impact event, and was explored in Section 4.5.2.
Table 4.5.1: Panel Geometry and Material Effects Model Summary

<table>
<thead>
<tr>
<th>Study</th>
<th>FEA Models</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Contacted Frames</td>
<td>1 frame, between stringers</td>
<td>Evaluated global stiffness of panel based on number of frames contacted. Each panel had an unloaded frame on either side of the indentation area.</td>
</tr>
<tr>
<td></td>
<td>1 frame, centered on stringer</td>
<td></td>
</tr>
<tr>
<td></td>
<td>2 frames, between stringers</td>
<td></td>
</tr>
<tr>
<td></td>
<td>2 frames, centered on stringer</td>
<td></td>
</tr>
<tr>
<td></td>
<td>3 frames, between stringers</td>
<td></td>
</tr>
<tr>
<td></td>
<td>3 frames, centered on stringer</td>
<td></td>
</tr>
<tr>
<td></td>
<td>4 frames, between stringers</td>
<td></td>
</tr>
<tr>
<td></td>
<td>4 frames, centered on stringer</td>
<td></td>
</tr>
<tr>
<td>Bumper Material Stiffness</td>
<td>$E_{\text{bumper}} = 10.34 \text{ MPa}$</td>
<td>Four frame panel indented across two frames, evaluated influence of bumper material stiffness on detectability.</td>
</tr>
<tr>
<td></td>
<td>$E_{\text{bumper}} = 103.4 \text{ MPa}$</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$E_{\text{bumper}} = 1.034 \text{ GPa}$</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$E_{\text{bumper}} = 10.34 \text{ GPa}$</td>
<td></td>
</tr>
<tr>
<td>Aluminum Component Study</td>
<td>All-Aluminum 2024</td>
<td>Understand the permanent deformation for hybrid composite-aluminum panels and how it relates to detectability.</td>
</tr>
<tr>
<td></td>
<td>Al 2024 Stringers</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Al 2024 Skin</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Al 2024 Shear Ties/Frames</td>
<td></td>
</tr>
<tr>
<td>Component Stiffness Study</td>
<td>Thick Stringers (29 plies vs. 15)</td>
<td>Evaluate the effects of the geometry of the stringers and skin.</td>
</tr>
<tr>
<td></td>
<td>Thin Skin (10 plies vs. 18)</td>
<td></td>
</tr>
</tbody>
</table>

In order to understand how the stiffness of each component influenced the damage initiation and evolution, a component thickness study was performed. Three cases were evaluated in Section 4.5.3: composite with a thin skin, thick stringers, and thick aluminum shear ties. The thin skin FEA model had 10 plies instead of the baseline 18 plies. The thick stringer was 29 plies instead of 15. The interest was to determine how the failure initiation and evolution would change for different configurations.

An important aspect of this research was the detectability of this type of impact event, specifically because composites exhibit largely elastic response up until failure, and they tend to relax to the original shape after impact, thereby leaving no permanent dent. Aluminum structures, however, typically undergo plastic deformation, which would lead to a permanent visible dent. Section 4.5.4 provides a detailed study of how the material (composite vs. aluminum) of each component
influenced the detectability of the impact event. The FEA models were primarily composed of composite material except for the component of interest, which was defined to be Aluminum 2024. Three hybrid composite-aluminum panels were evaluated: composite with aluminum skin, composite with aluminum stringers, and composite with aluminum frames and shear ties. An all-aluminum panel was also evaluated to provide a baseline for what type of dent would be expected.

4.5.1 Contribution of Non-Loaded Frames to Overall Stiffness

The number of frames contacted influenced the global stiffness of the panel. Several cases were evaluated to determine the contribution of each frame to the overall global stiffness. The models ranged from a three frame panel contacted over one frame, to a six frame panel contacted over the center four frames. In Figure 4.5.1, displacement contour plots of the of the three frame and six frame panels, at 10 mm of indentation, are shown. Two impact locations were evaluated, centered on a stringer and between stringers, as shown in Figure 4.5.2. All models described in this section used the FEA code Abaqus Standard. For each FEA model, the bumper terminated in the center of the frame bay with an non-loaded frame on either side of the loaded zone, at the edges of the panel.

The first question was to verify that it was acceptable to model each panel with only one non-loaded frame on either end. Two models were evaluated, a panel FEA model with one non-loaded and a panel model with two non-loaded frames on either side (see Figure 4.5.3 for an image of the panels). At 10 mm of bumper displacement the difference in load between a panel with one non-loaded and two non-loaded frames on either side was within 10%, as shown by the load history described in Figure 4.5.4. In order to verify that the deformation of the loaded frame was equivalent for the two cases, the bending strain on the top flange of the loaded frame was compared and is shown in Figure 4.5.5. The load and frame flange strains for both cases are summarized in Table 4.5.2.

Because the loaded frames behaved similarly (less than 12% difference for the strain comparison at 25 mm applied bumper displacement) between one and
two non-loaded boundary frames, all models were evaluated with only one non-loaded frame on either side of the impact zone in order to reduce computational costs.

**Figure 4.5.1:** Three Frame and Six Frame FEA Panels, Displacement Plot at 10mm Bumper Displacement

**Figure 4.5.2:** Indentation Locations for FrameXX Panels
Figure 4.5.3: Boundary Frame Check, Five Frame Panel Impacted Over One Frame

Table 4.5.2: Three vs. Five Frame Panel Impacted Over One Frame Summary

<table>
<thead>
<tr>
<th>Comparison Metric</th>
<th>3 Frame Panel</th>
<th>5 Frame Panel</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load at 10 mm disp.</td>
<td>37.9</td>
<td>41.4</td>
<td>8.5 %</td>
</tr>
<tr>
<td>Frame flange strain at 10 mm disp. (με)</td>
<td>563</td>
<td>591</td>
<td>11.5%</td>
</tr>
<tr>
<td>Load at 25 mm disp.</td>
<td>85.2</td>
<td>94.7</td>
<td>10.0%</td>
</tr>
<tr>
<td>Frame flange strain at 25 mm disp. (με)</td>
<td>1624</td>
<td>1831</td>
<td>11.3%</td>
</tr>
</tbody>
</table>

Figures 4.5.6 and 4.5.7 show the force per frame vs. bumper displacement history when indented at Location 1 and Location 2. As the number of frames contacted increased, the load per frame decreased and the stiffness contribution from the unloaded adjacent frames became less significant in the overall structural response. A similar response was observed for both impact locations. A summary of the effective frame stiffness (global stiffness divided by number of contacted frames) for each case is described in Table 4.5.3.
Figure 4.5.4: Boundary Frame Check - Three vs. Five Frame Panel Impacted Over One Frame Load Histories

Figure 4.5.5: Boundary Frame Check - Three vs. Five Frame Specimen Impacted Over One Frame
Figure 4.5.6: Frame Stiffness For Multiple Frames Contacted - Location 1

Figure 4.5.7: Frame Stiffness for Multiple Frames Contacted - Location 2
### Table 4.5.3: Number of Contacted Frames Summary

<table>
<thead>
<tr>
<th># of Frames Contacted</th>
<th>Impact Location</th>
<th>Frame Load at 10 mm Bump. Disp. (kN)</th>
<th>Frame Load at 25 mm Bump. Disp. (kN)</th>
<th>Effective Frame Stiffness, Bump. Disp. Range 5-10 mm (kN/mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (3 frm panel)</td>
<td>1</td>
<td>36.6</td>
<td>85.1</td>
<td>3.822</td>
</tr>
<tr>
<td>1 (3 frm panel)</td>
<td>2</td>
<td>37.9</td>
<td>85.2</td>
<td>3.897</td>
</tr>
<tr>
<td>2 (4 frm panel)</td>
<td>1</td>
<td>29.6</td>
<td>66.4</td>
<td>3.080</td>
</tr>
<tr>
<td>2 (4 frm panel)</td>
<td>2</td>
<td>31.2</td>
<td>65.4</td>
<td>3.178</td>
</tr>
<tr>
<td>3 (5 frm panel)</td>
<td>1</td>
<td>27.5</td>
<td>60.7</td>
<td>2.856</td>
</tr>
<tr>
<td>3 (5 frm panel)</td>
<td>2</td>
<td>29.0</td>
<td>59.4</td>
<td>2.956</td>
</tr>
<tr>
<td>4 (6 frm panel)</td>
<td>1</td>
<td>26.3</td>
<td>57.9</td>
<td>2.741</td>
</tr>
<tr>
<td>4 (6 frm panel)</td>
<td>2</td>
<td>27.7</td>
<td>56.3</td>
<td>2.837</td>
</tr>
</tbody>
</table>

While the global stiffness of the panel is dependent on the number of frames indented, the deformation that the each frame experiences during loading is fairly uniform. Figure 4.5.8 shows the six frame panel, indented across four frames. The strain at the top surface of each flange was extracted from the FEA model and is shown as a function of bumper displacement in Figure 4.5.9. It can be seen from the plot the Frames #3 and #4 experienced the same strain level in the flange, and those frames experience the same deformation state. Similarly, Frame #2 experienced the same strain levels as Frame #5. The strain in Frame #2 was 5% lower than the strain in Frame #3 at 10 mm of bumper displacement. If a larger panel was indented across more frames (e.g., an eight frame panel indented across six frames), no new insight would be achieved because the additional frames would behave the same as what was observed in the six frame panel. Therefore, this study shows that a five frame panel is sufficient to capture the behavior of a large panel under indentation (or impact) caused by wide area blunt impact sources.
Figure 4.5.8: Strain in Frames for A Six Frame Panel Indented Over Four Frames

Figure 4.5.9: Strain in Frames for A Six Frame Panel Indented Over Four Frames
4.5.2 Bumper Material Stiffness Study

For this study on the effects of bumper material stiffness, a four frame panel with elastic properties was evaluated in Abaqus Standard. The study consisted of four models, all identical except for the Young’s Modulus (E) value of the rubber indentor. The bumper material stiffness values ranged from Young’s Modulus of 10.34 MPa (actual rubber value) to 10.34 GPa. The load, contact area, peak tensile stress, peak compressive stress, and peak contact pressure for 38.1 mm of back-side indentor displacement for each FEA model is summarized in Table 4.5.4.

Table 4.5.4: Contact Pressure and Peak Bending Stresses in the Bottom of Ply 2 at a Back-Side Bumper Displacement of 31.8 mm

<table>
<thead>
<tr>
<th>Model $E_{bump}$</th>
<th>Load (kN)</th>
<th>Contact Area (cm$^2$)</th>
<th>Peak Contact Pressure (MPa)</th>
<th>Peak Tensile Stress (MPa)</th>
<th>Peak Comp. Stress (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10.34 MPa</td>
<td>156.2</td>
<td>1391.4</td>
<td>3.70</td>
<td>2018.8</td>
<td>310.8</td>
</tr>
<tr>
<td>103.4 MPa</td>
<td>166.4</td>
<td>392.6</td>
<td>17.8</td>
<td>1487.8</td>
<td>473.1</td>
</tr>
<tr>
<td>1.034 GPa</td>
<td>172.9</td>
<td>105.9</td>
<td>78.5</td>
<td>1347.9</td>
<td>792.9</td>
</tr>
<tr>
<td>10.34 GPa</td>
<td>176.6</td>
<td>45.4</td>
<td>202.9</td>
<td>1305.9</td>
<td>1050.8</td>
</tr>
</tbody>
</table>

The rigid indentor transferred a higher load to the panel than the compliant indentor, which is shown by the load-displacement plots in Figure 4.5.10. The non-linearity (loss of stiffness) observed in the load history plots was primarily caused by rotation of the frames. More load was transferred to the frames by the rigid indentor, which led to more frame rotation and a more non-linear response.

With a rigid indentor, the indentor could not conform to the surface of the panel, which resulted in a smaller contact area than what was predicted for the panel indented with a compliant indentor. The rigid indentor caused significantly higher peak contact pressures, approximately 55x higher (202.9 MPa for the rigid indentor in comparison to 3.7 MPa for the compliant indentor). With a further refined mesh, the high contact pressures caused by the rigid indentor would possibly be even higher. Also, the indentor was modeled as a flat pad, with the cross section described in Figure 4.2.1 of Section 4.2.1. If a rigid indentor was modeled with the geometry of the cylindrical bumper, it would provide a sharper geometry of the indentor and further reduce the contact area, which would significantly increase
the contact pressures between the skin and the indentor.

With a very compliant indentor (low E), the indentor deformed to the surface of the skin, as shown in the cross section view (A) on the left side of Figure 4.5.11. Figure 4.5.11 shows cross section views of the indentor and the contact pressure plot at a back-side indentor displacement of 38.1 mm for the most compliant indentor \( (E_{\text{bumper}} = 10.34 \text{ MPa}) \) and the most rigid indentor \( (E_{\text{bumper}} = 10.34 \text{ GPa}) \). The large amount of skin deformation observed in the panel indented with the compliant indentor caused higher tensile bending stresses in the skin at the shear ties when compared to the panel indented with a more rigid indentor. However, with a more rigid indentor, the peak compressive stress due to bending in the bottom of Ply 2 at the edges of the indentor was over 4x higher than when indented with a compliant bumper. Figure 4.5.12 shows the locations of peak tensile and compressive stresses in the bottom of ply 2 of the skin in the principle direction, labeled S11. A ply stack plot for the skin is shown in Figure 4.5.13, where Ply 1 is the indentation surface.

The impact event would likely be more visible with a rigid indentor due to the higher contact pressures and the sharp geometry transition at the edge of the indentor. The visibility would probably occur in the form of matrix cracking at the surface caused by the very high contact pressures. With a further back-side indentor displacement, the bending in the skin at the end of the indentor would be more significant than the tensile stress due to bending at the shear ties. The tensile stress at the shear tie is less significant because the loading is applied equally on either side of shear tie, and will limit the amount of local deformation. This is not the case with the skin at the edge of the indentor, the deformation will increase with further indentor displacement.
**Figure 4.5.10**: Load Transfer to Panel for Various Bumper Material Stiffness

**Figure 4.5.11**: Contact Pressures for Compliant (left) and Rigid (right) Indentors
Figure 4.5.12: Peak Tensile and Compressive Bending Stress Locations in Ply 2 of Skin

Figure 4.5.13: Skin Stack Plot
4.5.3 Relative Component Stiffness Study

This section discusses how the damage initiation and evolution changed for different structural configurations. Two cases are discussed in this section: a thin skin FEA model, and a thick stringer FEA model.

**Thick Stringer FEA Model**

This model was identical to the Baseline, except the layup of the stringers was $[0/45/-45/90/45/-45/0]_{2S}$ with a fabric 0/90 ply over the top surface, resulting in a total thickness of 4.11 mm (90% thicker than stringers in the Baseline FEA model). Figure 4.5.14 shows the load history for the Thick Stringer FEA model, impacted between stringers. In comparison to the FEA Baseline model, the initial failure was very similar. The key values of the load history for the Thick Stringer FEA model and the Baseline FEA model are compared in Table 4.5.5. At a load of 90.7 kN and a skin displacement of 24.2 mm, the loaded shear ties buckled in two places, which caused loss of load carrying capability in the shear ties and a significant load drop of 39.8 kN. At a load of 48.0 kN and skin displacement of 44.4 mm, the primary load path to the internal structure was through the stringer to frame interaction, which can be seen by the increased load after 44 mm of skin displacement.

The thicker stringers experienced less local deformation at the stringer-to-frame contact location than what was predicted by the Baseline FEA model. The thicker (i.e., stiffer) stringer transferred more load directly into the frames than in the Baseline FEA model, which caused more frame rotation. The additional frame rotation led to a lower global stiffness due to the rotated cross section of the C-frames. This eventually led to a failure initiation predicted near the frame boundaries.
Figure 4.5.14: Load History for Thick Stringers FEA Model, FEA Baseline Model and Experiments

<table>
<thead>
<tr>
<th>Event</th>
<th>Baseline FEA Model</th>
<th>Thick String, FEA Model</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Disp. (mm)</td>
<td>Load (kN)</td>
</tr>
<tr>
<td>First major initial load drop (loaded shear tie failure)</td>
<td>22.4</td>
<td>93.3</td>
</tr>
<tr>
<td>Frames contact stringer</td>
<td>32.6</td>
<td>66.9</td>
</tr>
<tr>
<td>Second peak load</td>
<td>61.5</td>
<td>106.9</td>
</tr>
<tr>
<td>Frames failure near boundaries (failure initiation)</td>
<td>80.1</td>
<td>70.8</td>
</tr>
</tbody>
</table>

This impact event would likely lead to visible damage at the skin to stringer location. There was a significant stiffness transition between the skin (2.69 mm) and the skin plus stringer (6.88 mm). Some tension failure initiation (in the direction perpendicular to the stringers) in the bottom most ply of the skin was
predicted by FEA directly under the stringer along the length of the stringer. This was not observed in the Baseline FEA model.

By increasing the thickness (i.e., stiffness) of the stringers, the damage evolution would be similar to the Baseline FEA model. The loaded shear ties would crush and eventually the frames would fail near the boundaries. However, the event would likely be more visually detectable than the Baseline FEA model due to cracks in the skin along the stringer to skin transition location.

**Thin Skin FEA Model**

It was of interest to determine if a thinner skin, with the same geometry and layup for all other components, would lead to visually detectable damage. A model with a thin skin, impacted between stringers was evaluated. The new layup of the skin was $[0/45/90/-45]_S$ with a $0/90$ fabric layer on both sides (10 plies total as compared to 18 plies in the Baseline FEA model) resulting in a skin thickness of 1.53 mm (42% less than Baseline). In Table 4.5.6, the key load and displacement values of the Thin Skin FEA model are compared to the Baseline FEA model. The force histories for the Thin Skin FEA model, Baseline FEA model, and the combined loading profile for the experiments are shown in Figure 4.5.15. The global stiffness of the FEA model with a thin skin was lower than than what was predicted by the Baseline FEA model when comparing skin displacement to total load, as shown in Figure 4.5.15. There are two factors that contributed to a lower global stiffness for the Thin Skin FEA model when compared to the Baseline FEA model. First, the thinner skin provided less resistance to the shear tie and frame rotation, resulting in a lower global stiffness, as shown by the load vs. time plot in Figure 4.5.16. The second factor was the compliance of the thin skin. Prior to the shear ties crushing, the skin lifted off the bumper in the center frame bays more than what was observed in the Baseline FEA model.

The skin in the center of each frame bay lifted off the bumper surface more than what was predicted by the Baseline FEA model. This phenomenon is described in more detail in Figure 4.5.17. The combination of bending and compression in the shear ties led to the deformation state of the skin shown, such
that the gap between the bumper and skin formed. Figure 4.5.17 shows the side view of the specimen deformation and the skin deformation just prior to the shear ties crushing.

**Table 4.5.6: Thin Skin FEA Model vs. Baseline FEA Model**

<table>
<thead>
<tr>
<th>Event</th>
<th>Baseline FEA Model</th>
<th>Thin Skin FEA Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Event</td>
<td>Disp. (mm)</td>
<td>Load (kN)</td>
</tr>
<tr>
<td>First major initial load drop (loaded shear tie failure)</td>
<td>22.4</td>
<td>93.3</td>
</tr>
<tr>
<td>Frames contact stringer</td>
<td>32.6</td>
<td>66.9</td>
</tr>
<tr>
<td>Second peak load</td>
<td>61.5</td>
<td>106.9</td>
</tr>
<tr>
<td>Frames failure near boundaries (failure initiation)</td>
<td>80.1</td>
<td>70.8</td>
</tr>
</tbody>
</table>

**Figure 4.5.15:** Load History for Thin Skin FEA Model, FEA Baseline Model and Experiments
There are a few differences between the Thin Skin FEA model and the Baseline FEA model. A thinner skin led to a lower global stiffness and the adjacent (i.e., not directly loaded) shear ties failed along the fastener line earlier than the Baseline FEA model. The frames underwent more rotation in the Thin Skin FEA model than the Baseline FEA model, as shown by the frame rotation vs. time plot in Figure 4.5.18. The final failure prediction of the Thin Skin FEA model is shown by two images in Figure 4.5.19 at a load of 61.9 kN and skin displacement of 78.1 mm. On the right side is an image of the frame rotation and global deformation of the panel. The left side of figure shows the failure initiation criteria predicted at the frame boundaries.

The biggest difference between was the visual detectability. The thinner skin deformed more than the skin in the Baseline FEA model, and failure initiation was predicted at the skin-to-stringer transition. Figure 4.5.20 shows the Hashin-Roten fiber compressive failure initiation criteria for the Baseline FEA model and the Thin Skin FEA model at 78.1 mm of applied backside bumper displacement.
Figure 4.5.17: Deformation for Thin Skin FEA Model Before Shear Ties Fail

Figure 4.5.18: Frame Rotation vs. Time for Thin Skin FEA Model and FEA Baseline Model
Figure 4.5.19: Failure in Frames at Boundaries for Thin Skin FEA Model

Figure 4.5.20: Failure Initiation in Bottom Surface of Skin for Thin Skin FEA Model and Baseline FEA Model
By decreasing the thickness (i.e., stiffness) of the skin, the global stiffness would be reduced. The frames would experience more rotation prior to shear tie failure because the more compliant skin would deform more locally than the Baseline FEA model. The loaded shear ties would crush and eventually the frames would fail near the boundaries. However, the sharp stiffness transition between the skin and stringer (i.e., abrupt thickness change) would likely lead to visible skin damage in the form of cracking along this stiffness transition.

4.5.4 Component Material Study

This section discusses how aluminum components (or metals exhibiting plastic deformation) in a composite structure influence the detectability of an impact event. Several cases were evaluated, with specific interest in the final deformation of the panel, residual skin dent after impact, and the damage to the internal structure and how this relates to detectability.

All Aluminum 2024 Components Impacted Between Stringers

In order to evaluate the detectability of a GSE impact event on a comparable aluminum structure, a model with identical geometry and configuration to the Baseline FEA model but with Aluminum 2024 material properties was evaluated. The unloading portion of the loading cycle of this model was included to determine the final deformation state after impact. It was found that the aluminum structure experienced a high amount of plastic deformation, specifically in the shear ties and frames, resulting in a visible dent. The black solid plot in Figure 4.5.21 shows the bumper displacement profile, including the unloading portion of the loading sequence. The skin displacement between loaded frames, shown in the plot with markers of Figure 4.5.21, during the uploading and most of the unloading, until a residual permanent displacement level of 31.7 mm. This skin displacement is equivalent to the permanent deformation of the skin after impact (i.e., dent depth).
Figure 4.5.21: Displacement Profile Including Unload

Figure 4.5.22 compares the combined load history for Frame03 and Frame04-1, the FEA Baseline model, and the all Aluminum 2024 FEA model. The Aluminum 2024 FEA model is depicted by the plot with the markers. At a skin displacement of 16.4 mm (89.9 kN), the shear ties began to yield, which caused the load to remain at approximately 90 kN for an additional 20 mm of skin displacement as the shear ties deformed plastically. At a skin displacement of 33.8 mm (85.6 kN), the frame-to-stringer contact became the primary load path. It can be seen that the load of the panel continued to increase until a peak load of 117.2 kN (69 mm skin displacement) where it plateaued until the bumper reached a displacement of 76.2 mm, then the panel was unloaded. A summary table comparing the load and displacement values between the Baseline FEA model and the all Aluminum 2024 FEA model is shown in Table 4.5.7.
Figure 4.5.22: Load History for FEA Baseline, All Aluminum Model and Experiments

Table 4.5.7: All Aluminum 2024 Compared to the Baseline FEA Model

<table>
<thead>
<tr>
<th>Event</th>
<th>Baseline FEA Model</th>
<th>All Aluminum 2024</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Disp. (mm)</td>
<td>Load (kN)</td>
</tr>
<tr>
<td>Loaded shear ties yield</td>
<td>N/A</td>
<td>N/A</td>
</tr>
<tr>
<td>First major initial load drop (loaded shear tie failure)</td>
<td>22.4</td>
<td>93.3</td>
</tr>
<tr>
<td>Second major peak load</td>
<td>61.5</td>
<td>106.9</td>
</tr>
<tr>
<td>Frames failure initiation near boundaries</td>
<td>80.1</td>
<td>70.8</td>
</tr>
</tbody>
</table>

The aluminum shear ties yielded and buckled simultaneously at a load of
89.9 kN and skin displacement of 16.4 mm, shown by the small load drop in Figure 4.5.22. The initial load that the aluminum shear ties began to yield was close to the peak load predicted by the Baseline FEA model prior to the crushing of the loaded composite shear ties. The aluminum shear ties buckled and underwent plastic deformation instead of crushing, so the shear ties provided more resistance to frame rotation, which led to a higher load at the peak bumper displacement. For 76.2 mm applied bumper displacement, the composite Baseline FEA model experienced severe internal fractures, while the aluminum structure deformed plastically, resulting in a visible dent. After unloading, the skin displacement remained at 31.7 mm of indentation, which is indicated in Figure 4.5.22.

Figure 4.5.23 shows the maximum in-plane principle plastic strain after impact of the aluminum panel. The shear ties and frames deformed plastically, but there was actually no plastic strain developed in the skin or the stringers, as indicated by the green color in Figure 4.5.23. The shear ties experienced plastic deformation in two locations, which essentially formed hinges in the shear ties at the radius region and at the bottom of the frame.

Figure 4.5.23: Plastic Strain of Aluminum Panel After Impact
Figure 4.5.24 shows the peak deformation of the frame and shear tie at a bumper displacement of 76.2 mm (left image in the figure). The residual strain and hinges in the shear tie directly in the loaded region are shown in the close up view of the shear tie in Figure 4.5.24. The plastic deformation in the shear ties was primarily responsible for the visible dent shown in the skin displacement plot in Figure 4.5.25.

![Figure 4.5.24: Plastic Deformation of Aluminum Panel After Impact](image)

Figure 4.5.25: Plastic Deformation of Aluminum Panel After Impact

This study of blunt impact on an all aluminum panel concludes that the panel would experience a large amount of plastic deformation in the internal com-
ponents, specifically the frames and shear ties, that would result in a large, visible dent. This would be apparent to ground service personnel, and the pilot during the walk around inspection between flights.

**Composite Components with Aluminum 2024 Skin**

This section describes a FEA model of a composite panel identical to the Baseline FEA model, with the exception of an Aluminum 2024 skin instead of a composite skin. As seen in the load history plot shown in Figure 4.5.26, changing the skin material to aluminum did not significantly alter the damage initiation and evolution. The initial failure occurred in the shear ties directly under the impactor at a very similar load (97.5 kN, 21.3 mm skin displacement) to the FEA Baseline model. Once the shear ties directly under the impactor failed, the load dropped by approximately 51 kN. At this point, the primary load path was through the stringer-frame contact interaction. The load increased as the impactor displacement increased up to a skin displacement of 53.1 mm and developed a peak load of 111.7 kN. At this point each of the adjacent shear ties began to form a hinge along the shear tie-frame fastener line. This led to additional frame rotation and global stiffness loss. Once the adjacent shear ties failed along the fastener line, the C-frame begin to fail near the boundary due to torsion, bending and shear in the C-frames. The load progression was almost identical to the composite Baseline FEA model, as shown in Figure 4.5.26.

There was some local plastic strain in the skin directly where the shear ties connect to the skin, as shown in Figure 4.5.27. The highest tensile in-plane plastic strain in the skin under the shear ties was 7066 με and the highest compressive in-plane plastic strain predicted was 6994 με.
There was some plastic deformation in the skin at the boundaries directly under the stringer, which reached plastic strain levels of 2.2%. Figure 4.5.28 depicts the plastic strain in the skin, viewed from the impact side of the panel. However, this was away from the impact area and most likely a function of boundary conditions and panel size. The edges of the skin and stringers in the FEA model did not have any boundary conditions applied. Therefore, as the impactor displaced into the panel, the stringer and skin at the boundary were not constrained by additional frame bays away from the impact area. This led to more deformation than what would be observed in a larger structure that does have boundary conditions at those locations (i.e., continuous structure).
An aluminum skin would likely not contribute to the detectability of the impact event and the damage evolution is not significantly influenced by the skin material. After an impact event at this location, there might be some localized plastic deformation at locations of increased stiffness (i.e., at the shear ties), but there would not be a highly visible dent from the deformation of the aluminum skin alone. The internal components play a larger role in the visual detectability of the impact event than the material of the skin for this particular impact scenario involving wide area and soft contact.
Composite Components with Aluminum 2024 Stringers

This section describes a FEA model of a composite panel identical to the Baseline FEA model, except the stringers had Aluminum 2024 material properties instead of composite. The failure history of a composite structure with aluminum stringers was almost identical to the baseline FEA model, as shown by the load history for the FEA Baseline model and Aluminum Stringer model in Figure 4.5.29. The initial peak load occurs at 95.5 kN and a skin displacement of 21.5 mm. The first major load drop was caused by the failure of the loaded shear ties in buckling and compression. Once the loaded shear ties failed, the primary load path was through the stringer to frame contact interaction. Hinges formed in the adjacent shear ties, in the same order as the Baseline FEA model, which allowed additional frame rotation and global stiffness loss. At the final displacement, the failure initiation was predicted in the C-frames near the boundaries similar to the Baseline FEA model and the experiments.

The FEA model with the aluminum stringers would likely lead to no permanent deformation. Even with the low yield strength of the Aluminum 2024, the only plastic strain in the stringers occurred where the stringers contacted the C-frames after the failure of the shear ties, which would not contribute to global residual deformation of the skin. Therefore, the impact event would probably not be visible from the impact side of the panel. Figure 4.5.30 shows the plastic strain in the stringers at the final impactor displacement. The locations where the stringers contacted Frames #2 and #4 is circled in red. It was deduced that the material of the stringers does not contribute to the detectability of a GSE impact event for this specific panel configuration and impact location.
A composite panel with aluminum stringers, impacted between stringers, would not likely lead to visible damage. The stringers yielded slightly at the stringer to frame contact points, but this would not cause permanent deformation of the skin. The load history was almost identical between the composite and
aluminum stringers. The material of the stringers, when loaded between stringers, is not influential to the visible detectability of the impact event.

**Composite Components with Aluminum 2024 Frames and Shear Ties**

It was determined from the all aluminum model that the shear ties and frames play a critical role in permanent deformation. Therefore, the FEA model with aluminum frames and shear ties included the unloading profile of the impactor, as shown in Figure 4.5.31. The skin displacement is shown by the dashed plot in Figure 4.5.31, which did not return to zero at the end of the loading cycle. The failure evolution of the FEA model with Aluminum 2024 shear ties and frames was very similar to the all Aluminum 2024 FEA model. The shear ties that were directly impacted yielded, which caused a minor load drop. At 34.7 mm of skin displacement the stringer to frame contact interaction became the primary load path. Table 4.5.8 summarizes the damage evolution for the FEA model with aluminum shear ties and frames in comparison to the all Aluminum 2024 FEA model and the Baseline FEA model.

**Table 4.5.8:** FEA Model With Aluminum Shear Ties and Frames Compared to the Baseline FEA Model and All Aluminum 2024 FEA Model

<table>
<thead>
<tr>
<th>Event</th>
<th>Baseline FEA</th>
<th>All Al. 2024</th>
<th>Al. Frames/Sh. Ties</th>
</tr>
</thead>
<tbody>
<tr>
<td>Loaded shear ties yield</td>
<td>N/A</td>
<td>16.4</td>
<td>89.9</td>
</tr>
<tr>
<td>First major load drop</td>
<td>22.4</td>
<td>93.3</td>
<td>N/A</td>
</tr>
<tr>
<td>(loaded shear tie failure)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Second major peak load</td>
<td>61.5</td>
<td>106.9</td>
<td>69.0</td>
</tr>
<tr>
<td>Frames failure initiation near boundaries</td>
<td>80.1</td>
<td>70.8</td>
<td>N/A</td>
</tr>
</tbody>
</table>
Figure 4.5.31: Load History for FEA Baseline Model, All Aluminum FEA Model, Aluminum Shear Ties and Frames FEA Model, and Experiments

Figure 4.5.32 shows a contour plot of the deformation magnitude in the panel after unloading. There was permanent deformation in the shear ties, which prevented the skin from rebounding to the original shape after impact. It can be seen by the large gap between the impactor and the panel in Figure 4.5.32, that the skin experienced a permanent dent with a depth of 28.2 mm. The frames also remained slightly rotated, in the close view of Figure 4.5.32 the frame flange remained slightly buckled after impact.
Figure 4.5.32: Residual Deformation of Panel with Aluminum Shear Ties and Frames After Impact

Figure 4.5.33 shows the plastic strain in the center frame and shear ties after unloading. The loaded shear ties remained flattened with the hinge directly under the frame. This is shown by the red on the “Directly Loaded Shear Tie” in Figure 4.5.33.

Figure 4.5.34 shows the deformation of the skin after unloading. The red indicates a residual dent of 28.2 mm. This would be a visible impact event, even with composite skin and stringers. For this particular impact location and configuration, the Aluminum 2024 shear ties and frames underwent extensive plastic deformation, in excess of 15% plastic strain.

The frames and the shear ties are very influential on the residual deformation of the panel, and therefore the visual detectability. The shear ties and frames yielded when loaded, which prevented the skin from rebounding to the original position. This led to a permanent deformation with a dent depth of 28.2 mm.
4.5.5 Panel Geometry and Material Parameter Study Discussion

From the series of FEA models used to expand on Baseline FEA model, several parameters were evaluated to determine how they influence the damage
evolution and visual detectability of the impact event. The findings are summarized in Table 4.5.9.

Table 4.5.9: FEA Model Results Summary

<table>
<thead>
<tr>
<th>Model</th>
<th>Initial Failure</th>
<th>Final Damage</th>
<th>Visually Detectable?</th>
</tr>
</thead>
<tbody>
<tr>
<td>Baseline</td>
<td>Loaded shear ties completely fail</td>
<td>Failure in frames near boundaries</td>
<td>No</td>
</tr>
<tr>
<td>All Alum. 2024</td>
<td>Loaded shear ties yield</td>
<td>Plastic deformation in shear ties + frames, large dent</td>
<td>Yes</td>
</tr>
<tr>
<td>Al 2024 Skin</td>
<td>Loaded shear ties completely fail</td>
<td>Failure in frames near boundaries</td>
<td>Possibly</td>
</tr>
<tr>
<td>Al 2024 Stringers</td>
<td>Loaded shear ties completely fail</td>
<td>Failure in frames near boundaries</td>
<td>No</td>
</tr>
<tr>
<td>Al Frames + Shear Ties</td>
<td>Loaded shear ties yield</td>
<td>Plastic deformation in shear ties + frames, large dent</td>
<td>Yes</td>
</tr>
<tr>
<td>Thick Stringers</td>
<td>Loaded shear ties completely fail</td>
<td>Failure in frames near boundaries, possible skin cracks</td>
<td>Yes</td>
</tr>
<tr>
<td>Thin Skin</td>
<td>Loaded shear ties completely fail</td>
<td>Failure in frames near boundaries, possible skin cracks</td>
<td>Yes</td>
</tr>
</tbody>
</table>

4.6 Modeling Methodology Summary

The methodology described in this dissertation followed specific steps in order to model this type of impact. The key aspects are listed below.

1. **Boundary Conditions.** The boundary conditions were important to produce a deformation state in the panel model that was similar to a larger structure model. The appropriate stiffness values at the boundaries were determined by matching the rotations and displacements at the panel boundaries to rotation and displacement values in the global structure.

2. **Panel Size.** It was determined (in Section 4.5.1) that a panel with three loaded frames and one non-loaded frame on each side of the impact zone is
of sufficient size to evaluate this type of impact event.

3. **Symmetry.** In order to reduce computational costs, a half symmetric model was used in the models discussed in this research. This is acceptable as long as there is no coupling (e.g., extension-twisting coupling) in the skin, frame, shear tie or stringer laminates.

4. **Merged Geometry.** The individual components were defined, assembled, then merged together to form one part (excluding the shear ties near the impact location). This reduced computational costs by eliminating several tie constraints.

5. **Effective Fastener Modeling Technique.** The fasteners were modeled as a strip of material with reduced strength instead of modeling each individual fastener. This approach was implemented in the shear ties within close vicinity of the impact zone. These shear ties were connected to the surrounding structure via tie constraints.

6. **Panel Elements.** Continuum shell elements are preferred over conventional shell elements when modeling complex geometries.

7. **Impactor Geometry Representation.** It was found that modeling the impactor as a solid rubber pad was acceptable for this type of impact event due to the compliance of the rubber material. With a stiffer impactor material, the exact impactor geometry would need to be defined.

8. **Scripting Approach.** When performing any type of parametric study, it is preferable to write a program or script that will automatically generate the models. This type of approach allows several cases to be evaluated more efficiently and reduces the probability of inducing user error (e.g., geometry mismatch or incorrect material orientation).
4.7 FEA Conclusions

Finite element analysis was used in combination with the large scale experiments to better understand the parameters that lead to widespread damage that is difficult to detect visually. The modeling methodology described in this chapter correlated well with the experimental results. The effective fastener modeling technique was implemented in the FEA models in order to better predict the frame rotation, which was essential to predict the correct failure evolution. The Baseline FEA model predicted the initial peak load (prior to failure of the loaded shear ties) within 3% of the Frame04-1 test data. The order in which the adjacent shear ties failed in the FEA Baseline model and the Frame03 experiment results was also consistent. However, the adjacent shear ties in the Baseline FEA model failed at different load and displacement levels than the Frame03 experiment.

The impactor material was found to be directly related to the detectability of the event. A soft (i.e., low modulus) indentor caused very low contact pressures because the skin deformed around the stiffer locations. However, this local deformation led to higher bending stresses in the skin at the shear ties. The stiff bumper caused very high contact pressures directly under the shear ties and high bending stresses where the impactor terminated (i.e., around the edges of the impactor). As the stiffness of the indentor increases, so does the likelihood of the event being visually detectable.

The thickness of each individual component was influential on the visual detectability of the impact event. The thicker stringers did not significantly influence the damage evolution, but the event would likely be visually detectable via cracks in the skin along the stringer to skin transition location. When the skin thickness was reduced, the global stiffness was also reduced. More frame rotation was predicted with a thin skin prior to shear tie failure because the more compliant skin deformed more locally than the Baseline FEA model. Similar to the FEA model with the thick stringers, a panel with a thin skin would likely lead to damage in the form of cracking in the skin along the skin to stringer stiffness transition.

The modeling methodology was applied to various parameter case studies
in order to understand the difference between the behavior of an all composite panel in comparison to one containing some or all aluminum components. An all aluminum structure would undergo significant plastic deformation, resulting in a very visible dent that would be detectable from the exterior. The material of the skin and stringers was found to not be influential in the detectability of the impact event, even if the skin is made of ductile metal (e.g., aluminum alloy). The skin did not significantly deform plastically, therefore after impact would return to the original state without a deformed aluminum internal structure to prevent doing so. The aluminum shear ties and frames experienced plastic deformation, which are the main factors in producing a visible dent.
5 Conclusions

5.1 Discussion

This dissertation includes both experimental investigation of, and the definition of a FEA modeling methodology to predict damage caused by wide area, blunt impact to an aircraft structure. The methodology to predict damage and the approach of determining the boundary conditions can be applied to larger, more complex structures. While the experiments and FEA studies described in this dissertation provided deep insight into the damage caused by this type of novel (in sense that it was not previously studied) impact event, it should be noted that the results of the panel tests are not directly representative of what would occur if a GSE vehicle accidentally contacted an actual composite aircraft. This is because the geometry and configuration is similar, but not identical between a full aircraft and the test panels. The actual details of the component structural configuration, the boundary conditions and joints connecting load bearing frame members to other portions of the aircraft, and the impact location significantly influence the initiation and progression of damage. However, despite this departure from "reality," this body of work provides a critical awareness and understanding of the blunt impact event, of what levels of damage are possible without externally visible signs that damage exists, and in this way can aid in more focused detailed studies of actual aircraft configurations.

Finally, the FEA-based modeling methodology did not incorporate the capability to predict delamination between the skin and stringers since it was determined from the Phase II dynamic experiments that stringer-to-skin delamination was not an expected failure mode for this particular impact location and velocity.
Potential future work could include implementing cohesive surfaces in the models to include the capability of predicting stringer-to-skin delamination.

5.2 Conclusions

High energy, wide area, blunt impact (HEWABI) on composite structures is a complex event in which many variables contribute to the visual detectability of any damage incurred. Impact location, bumper size and material stiffness, and structural configuration of the aircraft all influence the failure initiation and evolution, and ultimately whether the damage can be visually detectable. This dissertation examined the structural response of a composite monolithic panel impacted by a compliant rubber bumper. The conclusions listed below are the key findings of the investigation.

1. A methodology to experimentally evaluate impact events between GSE and an aircraft was developed. The methodology addresses the importance of correctly representing boundary conditions, and demonstrated an approach for how to create a similar deformation and stress state between the test panel (sub-structure) and a larger structure.

2. For wide area blunt impact loading involving soft-material bumpers onto composite structures, it is possible to develop severe levels of internal damage possible without visually detectable signs that such damage has been imparted. This includes fractured internal components such as shear ties, stringers, and even frames.

3. Structural configuration and impact location directly influence the damage location and detectability. Exterior visibility of the damage is influenced by loading location relative to stringers. No visible cracks were observed when the bumper was centered on the skin between stringers, but when the bumper was centered directly on top of the stringer the impact event was found to be visible in the form of cracks on the skin. This was caused by the high local bending stress in the skin developing at the large stiffness transition (i.e.,
abrupt thickness change between the skin and skin co-cured to the stringers) locations, which act as lines of bending stress concentrations.

4. It was found that significant load was not applied to the test panels until the hollow cavity of the rubber bumpers were completely collapsed. Modest contact where the bumper just touches the aircraft will likely cause no significant damage. To create widespread severe damage that includes multiple structural components, a major impact event involving high energy is required. An impact event of this scale would likely be very loud and cause the entire aircraft to move. Once this type of event occurred, damage to the main load-bearing members (namely, the frames) at locations away from the impact site can develop along the load path to surrounding internal structure, particularly at joints and locations of increased stiffness (i.e., where the frames connect to the passenger or cargo floor). Global movement of the aircraft could result in damaging contact and impact with other surrounding GSE. It should be noted, however, that such intense interaction between GSE and the aircraft does not guarantee that the event will be (self) reported, particularly if the impacted site shows no visually-observable evidence of denting or other damage.

5. A FEA based methodology was established, providing the capability to model and predict damage initiation and evolution in HEWABI events. While focused on the large panel specimens investigated as part of this research, the methodology is transferable to the investigation of other larger structural configurations. The FEA models employed the nonlinear explicit dynamic FEA code Abaqus Explicit and was validated by comparing key metrics to the experiments: namely force per frame vs. skin deformation response, including key failure events. Once the Baseline FEA model was verified, the methodology was used to create a series of models to perform sensitivity studies to determine which parameters were more influential in the creation of wide spread damage that is not visually detectable. A FEA model-generating script was created to quickly and consistently create FEA models in Abaqus. This was useful as it allowed different configurations to more readily be ex-
explored without the extensive time expense of building these large and very complex models. The script was used to evaluate several additional cases beyond a Baseline model, e.g., thinner skin or thicker stringers in comparison to the Baseline.

6. Experimental observations and FEA case studies showed that the bumper stiffness strongly influences the damage initiation, evolution and detectability. With a soft impactor, the contact and interlaminar stresses in the skin are lower, allowing the formation of widespread damage away from the impact site without the development of visible local damage (namely surface cracks). FEA showed that a soft rubber impactor allowed the skin to locally deform around areas of increased structural stiffness (e.g., in the skin under the shear tie where the load is transferred to the frame), thereby developing higher bending stresses in the skin than with a rigid impactor. However, as the stiffness of the impactor increases, the contact stresses at bumper termination locations (i.e., at the ends of the bumper) and under locations of increased structural stiffness significantly increase by several orders of magnitude, while the bending stress in the skin only moderately decreased. Contact with a rigid impactor would likely lead to matrix cracking on the surface at those stiff location or cracking at the bumper termination location due to high interlaminar shear and bending stresses.

7. FEA case studies found that damage to composite structures is potentially less detectable than aluminum structures. Aluminum structures experience plastic deformation at lower yield stress levels, whereas the high strength composite material behaves almost elastically until failure and thus relaxes back to the original shape after impact if no cracking has developed. In the absence of skin cracking or plastic deformation (for aluminum), FEA-based case studies have found that the internal structural components are more influential in the development of a permanent dent than the skin material. Depending on the impact location and component geometry, the internal structure within close vicinity of the impact area would likely crush or yield and in doing so, holds the skin in and can prevent the skin from rebounding
to the original position, thereby resulting in a visible dent.

8. FEA was used to show that as the thickness of various sub-components changes, so does the damage initiation and evolution. The shear ties play a critical role in the failure progression. A more compliant shear tie would allow more frame rotation and a progressive failure process, possibly creating damage away from the impact site. In contrast, a more robust shear tie resists frame rotation and forces a completely different failure mechanism (and failure location) which would be more local to the impact site.

### 5.3 Future Research Directions

The FEA modeling methodology described in this dissertation provided a well-defined and systematic approach to analytically predicting failures in composite panels caused by a wide area blunt impact event. The modeling methodology included failure prediction, however, delamination between the stringer and skin was not included as a possible failure mode. Cohesive surfaces in Abaqus Explicit should be implemented in the Python script in order to include this failure prediction capability.

Also, in the experiments there was crushing in the radius region of the shear ties that was not captured by the FEA models. Modeling of the radius region of the shear ties directly under the impactor would include a refined failure capability that could predict delamination between plies. The continuum shell elements, used in their standard way to represent the entire laminate thickness, do not account for delamination between plies within a laminate.

The GSE threat is very broad. There are several factors that contribute to the final damage state, and this dissertation only evaluated a handful of possible impact scenarios. Other topics that could be evaluated include: stringer spacing, frame spacing, glancing impacts, different impactor geometry and materials, varying dimensions of the rigid support that the bumper is mounted to, and impactor velocity. The component geometries could also be evaluated, specifically, the radius of the C-frames or geometry of the stringers, which would influence the
stringer-to-frame contact interactions. The stringer-to-frame contact interactions could also be evaluated in detail, by means of FEA models and lab scale tests, to understand the failure relationship caused by these two components pushing onto each other. Finally, the findings in this dissertation could also be used to establish a relationship between lab tests and ground operations.
References


A Cohesive Surfaces/Composites in Abaqus Tutorial

This tutorial is based on the standard failure models in Abaqus, refer to the Abaqus documentation, specifically the Abaqus Analysis User’s Manual Chapter 23.3 “Damage and failure for fiber-reinforced composites”.

A.1 Failure Parameters

Failure within the ply: Hashin failure: Predicts elastic- brittle failure. Four modes of failure are: fiber compression, fiber tension, matrix compression, matrix tension. For this research, continuum shell elements (SC8R with hourglass control, element deletion) were used. See Abaqus Analysis User Manual 6-11 23.3.3. Damage evolution and element removal for fiber-reinforced composites.

Cohesive surfaces to model delamination between plies: Traction – Separation behavior. Use when modeling bonded interfaces that the adhesive is assumed to be very thin and considered to have zero thickness. Represents normal opening, and shear in 1 and 2 directions.
A.1.1 Cohesive Surfaces

**Table A.1.1: Cohesive Elements vs. Cohesive Surfaces**

<table>
<thead>
<tr>
<th>Cohesive Elements</th>
<th>Cohesive Surfaces</th>
</tr>
</thead>
<tbody>
<tr>
<td>Finite thickness</td>
<td>Zero thickness</td>
</tr>
<tr>
<td>Defined as a section with elements</td>
<td>Defined as contact (“penalty”) surface</td>
</tr>
<tr>
<td>Adds mass to model</td>
<td>Does not add mass to the model</td>
</tr>
</tbody>
</table>
| Parameters: Failure Initiation  
Density  
Stiffness  
Failure Evolution | Parameters: Failure Initiation  
Failure Evolution |
| Traction - separation  
Continuum  
Gasket | Traction - separation behavior only |
|                       | Smaller ODB files, shorter run time |
|                       | New to Abaqus 6-11 |

**Cohesive Interaction Property Definition**

Define behavior:

1. Mechanical -> Tangential Behavior
   
   (a) Define as “frictionless”

2. Mechanical -> Normal Behavior
   
   (a) Accept defaults of hard contact

3. Mechanical -> Cohesive Behavior, see Figure A.1.1

4. Mechanical -> Damage
   
   (a) Define damage evolution and initiation, see Figure A.1.2
   
   (b) Define fracture energies and power law behavior
Stiffness defined as 100x the surrounding material stiffness. The higher the stiffness, the more brittle the behavior.

Figure A.1.1: Cohesive Contact Property Definition - Contact Stiffness
Figure A.1.2: Cohesive Contact Property Definition - Failure Initiation
Figure A.1.3: Cohesive Contact Property Definition - Failure Evolution
A.2 Defining a Composite Laminate Material Orientation

1. Define layup in composite laminate editor, even if only one ply.

2. In “Layup Orientation” use discrete option.

   (a) Define all normal surfaces and edge to be along the principle direction (1 material direction)

      i. NOTE: define edge set to select for principle direction and top surface to define normal

3. Define stack direction as element direction 3, see Figures A.2.1 through A.2.2
Figure A.2.2: Material Orientation Assignment

Figure A.2.3: Material Orientation Assignment
A.3 Mesh Stack Direction and Element Type

A.3.1 Mesh Stack Direction

1. Structural Mesh, assign stack direction, see Figure A.3.1

2. Define stack direction to be consistent across ply, see Figure A.3.1
   
   (a) Previous versions of Abaqus required to use sweep mesh controls for all continuum elements. This required every cell to be individually assigned by picking the sweep direction from an edge (very time consuming).

   (b) Abaqus 6-11.1 can use “Structured” mesh controls. Assign the stack direction based on the top surface. Multiple cells can be assigned a stack direction at the same time.

3. Query “Mesh Stack Orientation” to ensure consistent orientations. Figure A.3.2 shows the default stack direction. This will create a model that does not run. The fixed stack direction is shown in Figure A.3.3.

![Mesh Controls](image.png)

**Figure A.3.1:** Mesh Controls
Figure A.3.2: Mesh Without Stack Direction Defined

Figure A.3.3: Mesh With Correct Stack Direction
A.3.2 Mesh Element Type

Assign continuum shell elements SC8R with element deletion and enhanced hourglass control as shown in Figure A.3.4.
B Test Fixtures and Boundary Conditions

This appendix details the test fixtures and boundary conditions used for the StringerXX and FrameXX test panels.

B.1 StringerXX Test Fixtures

This section shows the drawings of the fixtures used to mount the StringerXX panels to the SATEC machine.
B.2 Phase I FrameXX Boundary Conditions

This section details the boundary conditions used for the Phase I FrameXX panels and the loaded frames on the Phase II FrameXX panels.
REFERENCE DRAWING

Steel Shaft, McMaster Part #5947K68
Mounted Bearing, McMaster Part #6667K16
Moment Connection, see Drawing FSWM-C
Stiffness Plate for Roller Assembly, see Drawing FSWM-H
Track Wheel, McMaster Part #1460T23
Roller Reaction Assembly, see Drawing FSWM-I

Mounted Bearing Support, see Drawing FSWM-B
Roller Assembly Base, see Drawing FSWM-G
Track Wheel, McMaster Part #1460T14

All pieces will be bolted together
2x Ø.531 (17/32) THRU BOTH WALLS

Standard 3x3x.25W Steel Tube

SIDE VIEW

END VIEW

ISOMETRIC

TOP VIEW

QNTY: 12
Frame Specimen
Wall Mount
Mounted Bearing Support

PROPRIETARY AND CONFIDENTIAL
THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF [COMPANY NAME]. ANY REPRODUCTION OR DISCLOSURE WITHOUT THE WRITTEN PERMISSION OF [COMPANY NAME] IS PROHIBITED.

A36 Steel

NEXT ASSEY. USED ON
APPLICATION
DO NOT SCALE DRAWING

3.00

8.00

Ref. Pt.

3.00

Ref. Pt.

5.40

1.50

Ref. Pt.

217
FILLETS AND BEVELS

SIDE VIEW

END VIEW

TOP VIEW

Ref. Pt.

0.25

Ref. Pt.

0.25

Straight Edge Bevels

R0.25 fillet

ISOMETRIC

Ref. Pt.

QNTY: 12
Frame Specimen
Wall Mount

C-Channel Connection

A36 Steel

DO NOT SCALE DRAWING

219
Standard 5 x 2 x .25W Steel Tube

SIDE VIEW

END VIEW

ISOMETRIC

QNTY: 1
Frame Specimen
Wall Mount
Roller Reaction
Cross Member

PROPRIETARY AND CONFIDENTIAL
THE INFORMATION CONTAINED IN THIS
DRAWING IS THE SOLE PROPERTY OF
[REQUIRE COMPANY NAME HERE]. ANY
REPRODUCTION, COPYING, DISTRIBUTION OR USE
OF THIS DRAWING WITHOUT THE WRITTEN PERMISSION OF
[REQUIRE COMPANY NAME HERE] IS PROHIBITED.
B.3 Phase II FrameXX Additional Aluminum Frame Boundary Conditions

This section details the additional boundary conditions used for the aluminum boundary frames on the Phase II FrameXX panels.
Top View
SCALE: 1:4

(4) x 1/2 - 13 UNC THRU

Side View
SCALE: 1:4

Reference Point

Isometric View
SCALE 1:4

SolidWorks Student License
Academic Use Only

TITLE: Outer Frame Fixed Connection Plate

MATERIAL: A36 Steel

Quantity: 2

All Dimensions in inches
SIDE VIEW

END VIEW

ISOMETRIC

TOP VIEW

Standard 3x3x.25W Steel Tube

2x Ø .531 (17/32) THRU BOTH WALLS

Frame Specimen Wall Mount Mounted Bearing Support

QNTY: 8

A36 Steel

DIMENSIONS ARE IN INCHES

1ST PLACE DECIMAL ± .20
2ND PLACE DECIMAL ± .10
3RD PLACE DECIMAL ± .05

DRAWN BY
CHECKED BY
ENGINEER
PRINTED

CONFIDENTIAL

PROPRIETARY AND CONFIDENTIAL

THE INFORMATION CONTAINED IN THIS DRAWING IS THE SOLE PROPERTY OF THE COMPANY NAME HERE. ANY USE, DISTRIBUTION OR REPRODUCTION OF THIS INFORMATION WITHOUT THE WRITTEN PERMISSION OF THE COMPANY NAME HERE IS PROHIBITED.
B.4 C-Frame Outer Mold Line Tool

NOTE:
1. Radius of curvature must be constant throughout part
2. Inner cross section must be consistent throughout part
3. Part may be welded and machined
4. Smooth finish as specified in Frame-B throughout part
5. No gaps after welding and machining between parts
6. Surface A must remain planar (flat) throughout entire assembly
7. Welded frame pieces to backplate

---

SolidWorks Student License
Academic Use Only
NOTE:
1. Radius of curvature must be constant throughout part
2. Inner cross section shown in G-G must be consistent throughout part
3. Part may be welded and machined
4. Smooth finish as specified in Frame-B throughout part
5. No gaps after welding and machining between parts (smooth transition)
6. Entire cross section must be a smooth transition (no steps between welded parts)
B.5 Shear Tie Outer Mold Line Tool
B.6 Stringer Silicone Mold

This mold was used to cast the silicone inserts used in the stringers.
B.7 Phase II Skin Tool
ISOMETRIC SUPPORT STRUCTURE VIEW
SCALE: 1:16

ST - 5

SIDE VIEW

END PROFILE
10 Series- 1020 profile

ST - 5

TITLE: Skin Tool - Precut 8020 Extruded Aluminum

NAME: G DeFrancisco

DATE: 11/22/2010

MATERIAL: 8020 Aluminum 1020

TOLERANCE: .005 in

QUANTITY: 2

SIZE: ST - 5

DWG. NO.: A

REV.

All Dimensions in Inches

SCALE: 1:12
### Total Part List

<table>
<thead>
<tr>
<th>ITEM NO.</th>
<th>PART NAME</th>
<th>SHEET NUMBER</th>
<th>DESCRIPTION</th>
<th>QTY.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Rolled Aluminum Panel</td>
<td>ST - 1</td>
<td></td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>Waterjetted Side Support</td>
<td>ST - 2</td>
<td></td>
<td>2</td>
</tr>
<tr>
<td>3</td>
<td>Waterjetted Curved Supports</td>
<td>ST - 3</td>
<td></td>
<td>10</td>
</tr>
<tr>
<td>4</td>
<td>1020 Bottom Side Supports</td>
<td>ST - 4</td>
<td>8020 Extruded Aluminum - 1020 Profile, Precut to 72 in.</td>
<td>3</td>
</tr>
<tr>
<td>5</td>
<td>1020 Bottom End Pieces</td>
<td>ST - 5</td>
<td>8020 Extruded Aluminum - 1020 Profile, Precut to 54 in.</td>
<td>2</td>
</tr>
<tr>
<td>6</td>
<td>1020 Short Interior Bottom</td>
<td>ST - 6</td>
<td>8020 Extruded Aluminum - 1020 Profile, Precut to 25.50 in.</td>
<td>6</td>
</tr>
<tr>
<td>7</td>
<td>1010 Side Long Top Supports</td>
<td>ST - 7</td>
<td>8020 Extruded Aluminum - 1010 Profile, Precut to 84 in.</td>
<td>2</td>
</tr>
<tr>
<td>8</td>
<td>1010 Vertical Supports</td>
<td>ST - 8</td>
<td>8020 Extruded Aluminum - 1010 Profile, Precut to 2,106 in.</td>
<td>10</td>
</tr>
<tr>
<td>9</td>
<td>Part 4119</td>
<td></td>
<td>8020 Fastener - 2 Hole Corner Bracket</td>
<td>90</td>
</tr>
<tr>
<td>10</td>
<td>Part 4114</td>
<td></td>
<td>8020 Fastener - 8 Hole Corner Bracket</td>
<td>26</td>
</tr>
<tr>
<td>11</td>
<td>Part 4175</td>
<td></td>
<td>8020 Fastener - 6 Inside Hole Corner Bracket</td>
<td>6</td>
</tr>
<tr>
<td>12</td>
<td>Part 4117</td>
<td></td>
<td>8020 Fastener - 4 Hole Joining Strip</td>
<td>4</td>
</tr>
<tr>
<td>13</td>
<td>Part 3386</td>
<td></td>
<td>Fastener Assemblies for Waterjetted Parts</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>Part 3393</td>
<td></td>
<td>Flanged Button Head Socket Cap Screw - 1/4-20 length 3/8</td>
<td>125</td>
</tr>
<tr>
<td>15</td>
<td>Part 3321</td>
<td></td>
<td>Fasteners for Corner Brackets</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1/4-20x1/2&quot; FBHSCS &amp; Econ T-Nut</td>
<td>450</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>1/4-20x1/2&quot; FBHSCS &amp; Econ T-Nut</td>
<td>20</td>
</tr>
</tbody>
</table>

**Diagram:**

[Sketch of the product]