Computational Finite Element Analysis Of Extreme Loading Response Of Structural Insulated Panel (Sip) Building Subsystems

Kimberly Anne Tanner

University of Mississippi

Follow this and additional works at: https://egrove.olemiss.edu/etd

Part of the Engineering Commons

Recommended Citation

This Thesis is brought to you for free and open access by the Graduate School at eGrove. It has been accepted for inclusion in Electronic Theses and Dissertations by an authorized administrator of eGrove. For more information, please contact egrove@olemiss.edu.
ABSTRACT

In the reconstruction after Hurricane Katrina, the Mississippi Emergency Management Agency (MEMA) investigated the possibility of utilizing Structural Insulated Panel (SIP) construction for hurricane “cottages” that would be resistant to a Category 3 hurricane through a joint project with a local architect, Bruce Tolar, and FMS Engineering, LLC, Mobile, Alabama. The connections between the SIP panels, including a cam-locking device common to SIP panel construction, were identified as vulnerable under high velocity wind loads. At that time, there were no published guidelines for performing a computational finite element analysis of the SIP system, and thus it was not possible to obtain the necessary system response data.

By observation of blast experimental tests performed by Aberdeen Proving Grounds in association with the United States Military Academy, West Point, major failure of the system was observed at two critical connections in the SIP wall assembly. Using ABAQUS CAE, a model of a static panel transverse bending test was reproduced to validate the modeling procedure. A finite-element based model of the SIP Hut was created and validated using the displacement and impulse data measured during the blast tests and provided by Aberdeen Proving Grounds, Maryland. The FE model of the SIP Hut was then altered to investigate the structural response of the SIP Hut to a high velocity wind loading on the same wall assembly. The detailed connection response gathered from the ABAQUS model was evaluated and improvements to the currently common connection detailing used in OSB/EPS SIP construction are recommended to improve structural performance under high velocity wind loading.
DEDICATION PAGE

This thesis is dedicated to my loving parents, Raymond and Anne Tanner, and my grandmother, Cora Belle George, for their lifetime of guidance and support.
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>SIP</td>
<td>Structural Insulated Panel</td>
</tr>
<tr>
<td>USMA</td>
<td>United States Military Academy, West Point, Maryland</td>
</tr>
<tr>
<td>FEA</td>
<td>Finite Element Analysis</td>
</tr>
<tr>
<td>FE</td>
<td>Finite Element</td>
</tr>
<tr>
<td>MEMA</td>
<td>Mississippi Emergency Management Agency</td>
</tr>
<tr>
<td>FEMA</td>
<td>Federal Emergency Management Agency</td>
</tr>
<tr>
<td>OSB</td>
<td>Oriented Strand Board</td>
</tr>
<tr>
<td>EPS</td>
<td>Extruded Polystyrene</td>
</tr>
<tr>
<td>ASCE</td>
<td>American Society of Civil Engineers</td>
</tr>
<tr>
<td>ICC</td>
<td>International Code Council</td>
</tr>
<tr>
<td>IRC</td>
<td>International Residential Code</td>
</tr>
<tr>
<td>E</td>
<td>Modulus of Elasticity</td>
</tr>
</tbody>
</table>
ACKNOWLEDGEMENTS

I would like to express my appreciation to all of those who have contributed to my educational endeavors and encouraged me throughout this process. To Dr. Christopher Mullen, for imparting not only invaluable lessons in structural engineering, but also in patience and the value of a cool demeanor. I would never have completed this work without his help, guidance, and example.

I would also like to acknowledge the support and guidance of Lt. Col. Dr. Steven Hart (ret.) formerly of the United States Military Academy at West Point, Maryland. Dr. Hart invited me to collaborate on the testing of the SIP Hut with his cadets from West Point, and I was forever improved by the company and collaboration with not only Dr. Hart, but also his cadets. The guidance from the military specialists at the Aberdeen Proving Grounds in Maryland was invaluable.

Last, but certainly not least, I would like to thank my parents, Raymond and Anne Tanner. I would never have been able to go back to graduate school without their support and encouragement. In my mind, their names are on the title sheet of this thesis right next to mine, because they have given just as much as I have the last two years to make this work possible.

December 4, 2015
# TABLE OF CONTENTS

ABSTRACT ............................................................................................................................. ii  
LIST OF ABBREVIATIONS AND SYMBOLS.......................................................................... iv  
ACKNOWLEDGEMENTS .......................................................................................................... v  
LIST OF TABLES .................................................................................................................... vii  
LIST OF FIGURES .................................................................................................................. viii  
1. CHAPTER 1 ..................................................................................................................... 1  
   1.1 Objectives ...................................................................................................................... 1  
   1.2 Motivations and Background ....................................................................................... 2  
   1.3 Literature Review ........................................................................................................ 10  
   1.4 Scope of Research ...................................................................................................... 13  
2. CHAPTER 2 ................................................................................................................... 17  
   2.1 SIP Single Panel – Static Analysis .............................................................................. 17  
   2.2 SIP Three Panel - Eigenvalue Analysis ...................................................................... 24  
   2.3 SIP Hut Building Subsystem - Dynamic Analysis ..................................................... 29  
3. CHAPTER 3 ................................................................................................................... 40  
   3.1 System Response to High Velocity Wind Load ......................................................... 40  
   3.2 Recommended Design Modifications ..................................................................... 43  
4. CHAPTER 4 ................................................................................................................... 47  
LIST OF REFERENCES ............................................................................................................ 49  
LIST OF APPENDICES ............................................................................................................ 52  
APPENDIX A – BLAST EXPERIMENTAL DETAIL, ABERDEEN PROVING GROUNDS, MD ............. 53  
APPENDIX B – RAW AND TRUNCATED BLAST TIME HISTORY DATA ..................................... 55  
APPENDIX C – BLAST EQUATIONS ........................................................................................ 57
LIST OF TABLES

Table 2.1 Material Properties for EPS/OSB SIP Panel, (Premier SIPS 2015) ................................. 18
Table 2.2 Published Data: Single Panel Transverse Bending (Keith 2006) ................................. 22
Table 2.3 Percent Error: ABAQUS vs Published Experimental Deflection (Keith 2006) .......... 23
Table 2.4 Free Undamped Vibration Results – Plate Theory (Timoshenko 1959)(Jawad 2004) . 26
Table 2.5 Abaqus CAE vs Plate Theory, Natural Frequency Comparison ................................. 27
Table 2.6 Analytical Blast Pressure Calculations vs. Measured Pressure Impulse Data, Shot 2.. 33
Table 3.1 Design wind pressures, qz (ASCE 2010) ................................................................. 41
Table 3.2 Required Load Capacity, Critical Connections ......................................................... 46
LIST OF FIGURES

Figure 1.1 Typical SIP with OSB Sheathing, EPS Core, and Cam Locks (Murus 2015) ....................... 3
Figure 1.2 Typical SIP Panel-to-Panel Assembly (Murus 2015) .......................................................... 4
Figure 1.3 Single Story SIP Construction, Aberdeen Proving Grounds, MD, 2014 ......................... 5
Figure 1.4 Timber (2x4) Nailers, Aberdeen Proving Grounds, MD ..................................................... 7
Figure 1.5 IRC 2012, Typical Corner SIP Framing Details (typ.) ........................................................ 8
Figure 2.1 Mesh Size, 4” x 4” and 2” x 4” @ Load Application Site .................................................... 18
Figure 2.2 Boundary Conditions and Applied Load, Single Panel, ABAQUS CAE ............................... 19
Figure 2.3 Transverse Load Test Assembly, (Keith 2006) ................................................................. 21
Figure 2.4 Load vs. Deflection – ABAQUS Static Results and Published Data (Keith 2006) ............ 22
Figure 2.5 Deflected Shape: Single Panel, Static Load Case, P = 2603 lbs, (inches) ............................ 24
Figure 2.6 Mesh Size – Three SIP Panel Wall Subsystem, Simply Supported ................................. 25
Figure 2.7 Eigenvalue Analysis, Homogeneous Material, Mode 1 ..................................................... 27
Figure 2.8 Eigenvalue Analysis, Homogeneous Material, Mode 2 ..................................................... 28
Figure 2.9 Eigenvalue Analysis, Homogeneous Material, Mode 3 ..................................................... 28
Figure 2.10 Experimental SIP Test Hut, Aberdeen Proving Grounds, MD, 2014 ......................... 29
Figure 2.11 Blast Testing Spatial Relationships, Aberdeen Proving Grounds, Maryland .............. 30
Figure 2.12  Node Labeling for the SIP Hut high-speed 3D deflection measurements, Aberdeen Proving Grounds, MD................................................................................................................................. 32

Figure 2.13  Simplified Triangular Pulse Response, image courtesy of FEMA, 2014.................... 34

Figure 2.14  Blast Overpressure: Analytic versus Experimental Pressure Time History............. 35

Figure 2.15  Rigid Body Motion Detected after Blast Test 2, April 2014, Aberdeen Proving Grounds, Maryland ....................................................................................................................... 36

Figure 2.16  Displacement Time History Comparison, Experimental Results vs ABAQUS CAE Output with Rayleigh Damping............................................................................................................ 38

Figure 2.17  Max Displacement @ Node 3, ABAQUS CAE ............................................................ 39

Figure 3.1  160 mph Wind Load Event.......................................................................................... 41

Figure 3.2  Maximum Displacement under 200 mph static wind pressure, Hut Subsystem ....... 42

Figure 3.3  Recommended Design Modifications - Straps......................................................... 42

Figure 3.4  Elements vulnerable to failure under an extreme loading........................................ 43

Figure 3.5  Stress Contours, Wall Cam-Lock Representative Rod, @200mph Wind Load ........... 44

Figure 3.6  Stresses (S11), Steel Rod Representing Roof Cam-Lock, ABAQUS CAE ................. 45
1. CHAPTER 1

INTRODUCTION

1.1 Objectives

The objective of this paper is to develop a computational finite element modeling procedure for ABAQUS CAE to capture the structural response of a Structural Insulated Panel (SIP) subsystem. The modeling procedures are verified by plate theory, beam theory, and modal analysis. The complete subsystem model is verified by comparing experimental blast data to the response output from the ABAQUS CAE (United States Military Academy, Aberdeen Proving Grounds). The subsystem is then evaluated under the extreme load of a high velocity wind event and recommendations are made to improve structural response of the system under a high velocity wind load event.
1.2 Motivations and Background

Structural Insulated Panel (SIP) construction is widely used in both residential and commercial applications. After Hurricane Katrina, the Federal Emergency Management Agency (FEMA) and the Mississippi Emergency Management Agency (MEMA) collaborated with engineers and architects to evaluate the feasibility of constructing “hurricane cottages” from SIP for families that lost their homes due to hurricanes. The resulting design, termed the “Katrina Cottage,” was constructed and utilized on a limited scale along the coastal areas of Mississippi, Alabama, and Louisiana impacted by Hurricane Katrina (Alter 2015). Evidence suggests that SIP construction may exhibit a more robust structural response to high velocity wind loading than the currently-common stick-built construction methods (Amerisips 2015). The International Building Code, among others, contains requirements for SIP construction in high velocity wind design areas (IRC 2012).

A Structural Insulated Panel (SIP) is a composite construction material produced in factories across the United States. Many varieties of SIP are available in the United States which differ depending upon the types of materials used to construct the panels. Regardless of material, the manufacturing process binds the outer sheathing to the EPS core with an adhesive unique to each manufacturer (IRC 2012). The rigid polyurethane core is intended to act as a highly efficient insulator and to provide shear strength to the panel. SIP contributes only negligible structural stiffness to the composite panel. In effect, the extruded polyurethane core (EPS) has been found in this research and by others to have a negligible flexural stiffness (“Murus Structural Insulating Panels Specifications,” n.d.)(Kermani 2006)(Mathieson 2014)(NAHB; Building Works Inc; SIPA 2007)(Talwin 2002)(Vaidya, Uddin, and Vaidya 2010).
The structural response of a SIP panel relies upon the composite action of the outer sheathing layers adhered with a glue to the inner insulating core. For this research, a SIP panel similar to Figure 1.1, consisting of two outer layers of 7/16” thick OSB with a rigid polyurethane (EPS) core is considered.

![Figure 1.1 Typical SIP with OSB Sheathing, EPS Core, and Cam Locks (Murus 2015)](image_url)
In 2006, the APA-The Engineered Wood Association, published a report containing the results of the mechanical properties tests they performed on commonly available SIP panels with a polystyrene core and OSB facing (Keith 2006). Other researchers have published similar experimental results. However, these researchers evaluated the response of single panels under static loading. While this is sufficient to determine mechanical properties such as the modulus of elasticity and the ultimate strength of each panel in racking, shear, and bending; these published results do not address the structural response of a SIP system under the application of an extreme dynamic loading (Keith 2006). To construct a SIP building, the individual panels are typically attached to one another using the cam lock system visible in Figures 1.1 and 1.2.

Figure 1.2 Typical SIP Panel-to-Panel Assembly (Murus 2015)
In the SIP systems considered in this report, the connection details utilized generally conform to the International Residential Code (IRC 2012). The connections between panels at the cam locks are locations of high stress concentrations under an extreme loading event such as a high velocity wind loading. Additionally, the connections between the walls and the roof and between the walls at corners (as shown in Figure 1.3) are vulnerable under a high velocity wind load event due to the stress concentrations at those connection points.

Figure 1.3 Single Story SIP Construction, Aberdeen Proving Grounds, MD, 2014

The connections between the SIP are subjected to high stresses under any extreme loading event, but of particular interest is the response of the system and the subsequent increase in stresses at each connection during an extreme wind load event.
In 2014, this researcher was invited by engineers and cadets with the United States Military Academy, West Point, Maryland (USMA), to collaborate on the preliminary design, testing, and system response evaluation of a single story SIP structure under a prescribed blast loading that was defined by the requirements of the USMA for their research purposes. A series of consecutively more destructive blast tests on the SIP Hut revealed that under a blast event, the critical portion of the SIP Hut system were the connections between panels and the connections at the eaves, much like the expected response from the high velocity wind load case that was previously considered for the MEMA project. The experimental data gathered by the technicians with the United States Army at Aberdeen Proving Grounds in Maryland was provided to Dr. Christopher Mullen, Associate Professor of Civil Engineering, University of Mississippi through Dr. Lt. Col. Steven Hart (ret.), West Point Military Academy. The experimental data was then utilized to verify the FE-based simulation created for this research.

A common practice for stabilizing SIP structures is the use of light gage metal studs around each panel to form a moment-resisting frame that directs the load path downward to the foundation (Kermani 2006). However, the use of metal studs was not feasible in the case of the project for MEMA/FEMA, nor would metal studs be suitable for the USMA’s intended military application. Under circumstances where light gage metal stud framing is not feasible, timber nailers are commonly used to frame each SIP before assembly of the superstructure (IRC 2012; Amerisips 2015; Premier SIPS 2015). For each edge of a SIP that receives a nailer, a
portion of the EPS core is cut out of the panel to create a space for the timber nailer as seen in Figure 1.4.

Figure 1.4  Timber (2x4) Nailers, Aberdeen Proving Grounds, MD

Using the previous experience with SIP panel assemblies under extreme loading garnered from the “Katrina Cottages” project, the University of Mississippi researchers collaborated with the United States Military Academy, West Point to design a full-scale representative SIP Hut for military applications. The SIP Hut test specimen was initially designed to conform to International Residential Building Code, 2012, per the United States Armed Forces (USAF) construction standards. The researchers made changes to the configuration and connection design that diverge from IRC2012 requirements, intended to increase the system's performance under a blast loading. Also, the IRC2012 allows flexibility for
connection design, “SIPs shall be connected at vertical in-plane joints in accordance with Figure R613.8 or by other approved methods” (IRC 2012). Approved methods include the use of a cam-lock system to connect vertical panels, and is commonly available from various SIP manufacturers (Amerisips 2015; Murus 2015). Since this construction is expedient to construct and requires few tools, it was also the more desirable connection detail from the perspective of the USMA. The representative SIP Hut was constructed at full scale using the commonly specified connection for OSB SIP Panels with a cam-lock system between laterally-contacting SIP Panels and the use of 8” long hex screws to connect the SIP Panels at corners, floors, and at the roof as shown in Figure 1.5.

![Figure 1.5 IRC 2012, Typical Corner SIP Framing Details (typ.)](image)

The results from the blast experiments were then used to validate the response of a FE-based computational simulation of the SIP Hut tested by Aberdeen Proving Grounds. The impulse and deflection data were available from the test, so it was used to develop a blast time history to apply to the SIP Hut model. The data provided by Aberdeen Proving Grounds
included over 100,000 data points. However, ABAQUS CAE can only accept a limited number of
data points, so the available experimental data was truncated to capture the significant portion
of the blast impulse, see Appendix. By comparing the load-deflection data provided by ABAQUS
and the load-deflection data provided by Aberdeen from the blast experiment, the model was
validated under an extreme dynamic load. The model of the SIP Hut was then subjected to a
representative high velocity wind load time history expected during a major hurricane event.
The ABAQUS CAE models provide detailed response data on the critical connections of this SIP
assembly for the representative high velocity wind load event. The detailed stress output on
the connections and the overall system response was used to recommend design modifications
to the SIP Hut subsystem to improve structural performance under the high velocity wind load
event.
1.3 Literature Review

Structural Insulated Panels (SIP) are manufactured using various materials for the core and facing to allow for flexibility in application. For instance, researchers have shown that SIP with OSB (oriented strand board) facing materials are prone to disintegration when exposed to floodwaters (Mousa and Uddin 2013). Other materials, such as glass fiber-reinforced polymers and glass fiber magnesium cement are often examined for their potential use in environments detrimental to OSB-facing SIP (Mathieson 2014)(Smakosz 2014).

In 1993, the APA – The Engineered Wood Association (APA), published design specifications for composite sandwich panels with plywood facing materials (APA 1993). These design specifications were based on classic laminated beam theory and did not account for OSB (oriented strand board) facing materials. In the 2007 supplement to the International Residential Building Code, minimum panel properties required for the facers of SIP panels were published (IRC 2007). However, the properties listed in the IRC2007 supplement do not reflect the SIP panels which are composed of OSB facing materials (SBA 2005). Subsequently, the APA released another report in 2009 providing design specifications based on static loading tests for OSB-faced SIP panels (Keith 2009). The APA performed laboratory testing on SIP Panels with OSB facing material and EPS (extruded polystyrene) cores to establish the minimum material properties for these panels. These tests included shear, axial, transverse and lintel testing in accordance with standardized testing methods from ASTM and ICC, among others. The experiments performed by the APA tested only single panels, not a connected assembly of panels, such as a wall system (Keith 2009).
In 2006, Kermani noted that the “method of erection and connection has a large influence on the finished strength of the components” (Kermani 2006). SIP manufacturers commonly recommend the addition of timber nailers seated inside a routed slot of each SIP panel as a standard construction detail (Murus 2015). Kermani’s research is focused on the effects of adding these timber nailers along the edges of the SIP panels. With the addition of the nailers, Kermani subjected the SIP panels to a series of static loading tests including axial compression, and a combined bending and axial compression test. Kermani found that the addition of the nailers (stiffeners) improved the performance of the panel under various loading schemes, as expected. However, no currently published research sources provide results or design recommendations of an OSB SIP system subjected to extreme dynamic loading.

Many researchers including Frostig, et al, and Bozo, have examined the effects of static loading on composite sandwich material (Frostig 1992; Bozo 2002). Smakosz, et al. used ABAQUS CAE to evaluate SIP panels manufactured with cement board facings, but they used ABAQUS to evaluate the acoustic and thermal properties of the panels, and did not attempt a structural response model (Smakosz 2014). Design specifications are readily available for SIP panels, but these design specifications are based on static loading. While the results of static loading tests are valuable, they cannot represent the dynamic response of the system under a dynamic load, especially an extreme load like blast or high velocity wind events (Vaidya, Uddin, and Vaidya 2010).

Additionally, researchers have published results from blast tests carried out on single SIP panels in a steel frame with four edges restrained against rotation or translation. The results of these experiments are significant, but do not provide any information about the system
response for a SIP assembly or structure (Terentiuk 2012). Subsequently, analytical calculations to estimate the stress concentrations at connections between panels is impractical and limited.

In summary, other researchers have performed single-panel static load testing on various types of SIP panels, and their results are readily available. There is no currently published research evaluating the computational modeling procedures necessary to capture accurate SIP system response under the effects of an extreme dynamic load. The effects of wind loading on structural subsystems constructed of materials other than SIP are widely available (Dong 2012; Kermani 2006). The connections between the walls and roof and between the wall panels are areas of high stress concentrations due to extreme loading, so the critical potential failure locations are evident. In 2013, Chowdhury published a paper evaluating the effects of wind loading on roof to wall connections in classic residential timber design (Chowdhury et al. 2012). Utilizing the validated FEA models created during the course of this research, the vulnerabilities of the whole system may be observed and evaluated. More specifically, the stresses accumulating at each of the connections was determined from ABAQUS CAE output and probable modes of failure were identified at the connections under extreme wind loads. By evaluating the dynamic response of the hut subsystem under an extreme dynamic wind loading, the overall structural response of the subsystem may be examined and design modifications recommended.
1.4 Scope of Research

The first chapter of this thesis is the Introduction and covers the Objectives and Motivations of the research. Using the time history of a blast test provided by the USMA from an experiment performed by the USMA and Aberdeen Proving Grounds, computational finite element models were developed in ABAQUS CAE to investigate the points of failure of a representative SIP system (SIP Hut). A detailed examination of the output from the ABAQUS CAE model allows for the detailed response data necessary to identify areas of probably failure under a high velocity wind load. The first chapter also includes a Literature Survey, which investigates some of the publications relating to SIP construction and research, informing the reader on the current state of research in the field.

This research focuses on system response under extreme loading, not on the nonlinear material behavior of the composite SIP. Therefore the composite SIP panel is assembled using three “parts” in ABAQUS CAE. Each of these parts is a 3D deformable solid. The three parts were then connected using a “tie” constraint, essentially “gluing” the layers to one another and preventing delamination. Delamination and other nonlinear material behavior is not within the scope of this research. Additionally, steel bars were “embedded” and utilized in the place of cam locks and the 8” hex screws throughout the FEA model. Detailed modeling of the connections is not within the scope of this research.

Initially, a finite element analysis model of a single SIP panel was created. Using previously published data from static load tests performed by the American Plywood Association, the model of the single panel was validated by recreating the load test performed by the APA (Keith 2006). An ABAQUS model of a single panel was created and compared to
experimental data from the American Plywood Association (APA). The APA performed a three-point transverse static load test on a single SIP panel with OSB facing and EPS core and published those results in 2006 (Keith 2006). ABAQUS provided load-deflection curves from the finite element model and those results were compared to the results from the APA report. This comparison showed a negligible percent error between deflection results, thus validating the finite element modeling procedure for a composite SIP panel.

Using the same modeling procedure utilized for the single panel FEA model, a three-panel wall assembly was created in ABAQUS. By running the frequency analysis in ABAQUS, the eigenvalues and mode shapes were produced. Comparing the natural frequencies obtained from the modal analysis to the natural frequencies given through a simplified analytical calculation of Timoshenko plate theory, the three-panel wall model was validated (Timoshenko 1959).

A finite element analysis model of the test subject SIP Hut used by Aberdeen Proving Grounds for their blast experiment was created using the same modeling techniques that proved successful for the single panel and three-panel cases. The parameters of the blast test performed by the Aberdeen Proving Grounds in association with the USMA is presented in Chapter 2 and the Appendix. The measured pressure time history provided from the blast test was input into the finite element model as an implicit dynamic load case. The ABAQUS results provide the resulting displacement time history. By replicating the blast experiment with the finite element model, the modeling procedures were validated. The technicians at Aberdeen Proving Grounds took measurements with high-speed cameras to measure the deflection of the wall at twelve nodes. The deflection time history results from ABAQUS were then compared to
the deflection results measured by the technicians at Aberdeen. The methodology of creating the finite element models in ABAQUS CAE is presented in Chapter 2.

The final chapter of this thesis presents the results of the ABAQUS SIP Hut model subjected to a high velocity wind event that follows the modeling procedures validated in Chapter 2. The effects of negative pressure inside an enclosed structure were neglected in this instance.

The results from the ABAQUS model under a high velocity wind load provide detailed response information about the necessary capacity of each connection to prevent failure. Previously, the connections were identified as the likely locations of failure in the subsystem, and by viewing the results of the finite element analysis model, this hypothesis is justified. Stress concentrations at the connections are magnitudes higher than at other locations in the subsystem. The response data for the connections was collected and has been presented in Chapter 3. Using these results, design modifications have been recommended and are outlined in Chapter 3 that will improve the performance of this SIP Hut system under a high velocity wind event and provide guidance for other engineers and researchers performing computational FE-based simulations of extreme loading events on SIP systems.

In conclusion, the FE-based simulations created for this report are limited in their applicability. For examination of system response under extreme loading and identification of areas of stress concentrations, these models have proven reliable. However, if failure occurs as a result of nonlinear material behavior like delamination, these models would not represent that failure mode. The finite element models created for this research only consider the linear
elastic material behavior response. The nonlinear material behavior is outside the scope of this paper.
2. CHAPTER 2

FINITE ELEMENT BASED SIMULATION OF SIP

2.1 SIP Single Panel – Static Analysis

A finite element model created was a model of a single SIP panel that was created using ABAQUS CAE and compared to the published experimental results. SIP is a composite material comprising three discrete layers (Tompos 2008). Each of these layers was modeled as a 3D deformable solid in ABAQUS. SIP layers are traditionally attached to one another to form the panel with a type of adhesive that is specific to the manufacturer. Since this research is only interested in the overall linear elastic system response of a SIP structural system, the layers of the SIP were “adhered” together in the FEA model using a “tie” constraint. In ABAQUS CAE, a “tie” constraint joins two nodes together preventing any relative motion between them. Utilizing the “tie” constraint in this way, delamination failure is prohibited by the simulation. The material properties used for the ABAQUS model of the SIP Panel were obtained from published data from a typical OSB-faced EPS SIP Panel manufacturer, Premier SIPS, the largest SIP manufacturer in North America (Premier SIPS 2015). The material properties used in the model are provided in Figure 2.1.
SIP PANEL ASSUMED MATERIAL PROPERTIES:

<table>
<thead>
<tr>
<th></th>
<th>Modulus of Elasticity, E (psi)</th>
<th>Poisson’s Ratio</th>
<th>Weight (lb/ft³)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Facing(OSB)</td>
<td>1350000</td>
<td>0.2</td>
<td>34</td>
</tr>
<tr>
<td>EPS Core</td>
<td>700</td>
<td>0.00001</td>
<td>2.2</td>
</tr>
<tr>
<td>Timber Nailer (SPF #2)</td>
<td>1315000</td>
<td>0.4</td>
<td>27</td>
</tr>
</tbody>
</table>

Table 2.1 Material Properties for EPS/OSB SIP Panel, (Premier SIPS 2015)

Figure 2.1 Mesh Size, 4” x 4” and 2” x 4” @ Load Application Site

The three layers of the SIP panel, two OSB layers and one EPS layer, are modeled as 3D deformable solids in ABAQUS CAE. Representing the layers as 3D deformable solids restrains the rotational degrees of freedom at each node. Other researchers have previously modeled a single SIP panel as a plate element, which frees the rotational degrees of freedom but cannot accurately represent the distribution of stresses through the thickness. By definition, a plate
element is assumed to have negligible stress normal to the plate surface (Timoshenko 1959). These three solid layers must be considered during the meshing process.

The core “Part” was divided into two elements through the thickness and elements at 4” square across the length and width of the panel. The facing “Part” was meshed such that the thickness of the OSB is represented by two elements while the top and bottom meshes were divided into 2” square elements. This relatively fine mesh requires ABAQUS CAE to calculate stresses and strains at each node, which provides output for the stress distribution through the thickness of each solid element. The nailers are standard timber 2x4s. Wood is a naturally anisotropic material (although it can be said to be transversely isotropic), but modeling complex material properties is outside the scope of this research (Gere 2001).

The mesh was refined to ensure capture of the full behavior of the panel with the tie constraint used between the SIP layers. The mesh size is given in Figure 2.2. The boundary conditions were modeled to replicate the physical test which calls for one end of the panel pinned while the other end is on a roller (ASTM 2005), see Figure 2.3.

Figure 2.2  Boundary Conditions and Applied Load, Single Panel, ABAQUS CAE
The timber nailers and the OSB were defined material properties of isotropic, homogeneous material for simplicity. The core is carved out of the SIP panel to allow a space for the 2x4 nailer at the edges, and the nailer is affixed using 8d common nails at 6”-8” o.c. The “tie constraint” option in ABAQUS connects the timber nailer to the SIP panel with a tied connection, thus preventing any separation during loading. The core is assumed to be an extruded polystyrene material with isotropic material properties. In reality, the EPS core material is highly nonlinear. Defining the core material as isotropic considerably simplifies the problem.

This research aims to produce a FE-based model that will represent the linear elastic behavior of the SIP subsystem. Once the materials have been pushed beyond their yield stress limits, the model becomes invalid. Localized material failure such as delamination is disregarded since this is a clear indication that the panel is experiencing stresses beyond the elastic yield stress limit and is therefore outside of the research scope.

The single panel ABAQUS model was then compared to the results of the 2006 report from the American Plywood Association, APA Report T2006P-33 for 8’ x 4 ½” single SIP panel, to validate the modeling procedure for the SIP panels as a composite layered material (Keith 2006). The 2006 report from the APA provides load-deflection data for a three-point transverse static load test performed on a single SIP panel (four test specimens)(Keith 2006). The transverse bending load test performed by Keith, 2006, conforms to ASTM E 72-05 (ASTM 2005). The applied loads shown in Table 2.1 represent a total point load applied to the SIP panel during the experiment. This point load is divided between two load application sites, each of which have a surface area (A) = 192 in².
To apply these loads to the FE-based simulation in ABAQUS CAE, these point loads were halved and divided by this cross-sectional area to give a pressure, p. This pressure was applied to the SIP panel at the location of the applied load, see Figure 2.3.

\[ p = \frac{P}{2A} \]

- **P** = applied load (lbf)
- **p** = pressure (psi) applied at load application
- **A** = area of applied pressure (psi)

*Figure 2.3 Transverse Load Test Assembly, (Keith 2006)*
### Transverse Load Test Results (lbf) for 4-1/2" x 8 ft SIPs

<table>
<thead>
<tr>
<th>Wall</th>
<th>Height (in)</th>
<th>Ultimate Load (lbf)</th>
<th>Slope (lbf/in/4 ft)</th>
<th>Load at Deflection (lbf)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>L/360</td>
</tr>
<tr>
<td>1</td>
<td>96</td>
<td>3691</td>
<td>3473</td>
<td>953</td>
</tr>
<tr>
<td>2</td>
<td>96</td>
<td>3755</td>
<td>3373</td>
<td>972</td>
</tr>
<tr>
<td>3</td>
<td>96</td>
<td>3421</td>
<td>3367</td>
<td>913</td>
</tr>
<tr>
<td>Mean</td>
<td></td>
<td>3622</td>
<td>3404</td>
<td>945</td>
</tr>
<tr>
<td></td>
<td>Calculated allowable Load (psf)</td>
<td>38</td>
<td>n/a</td>
<td>30</td>
</tr>
<tr>
<td></td>
<td>Allowable Load (psf)</td>
<td>n/a</td>
<td>n/a</td>
<td>30</td>
</tr>
</tbody>
</table>

#### Table 2.2 Published Data: Single Panel Transverse Bending (Keith 2006)

Figure 2.4 compares the published results of the transverse bending static load test by Keith, 2006 and the deflection output from the corresponding FE-based model.

---

**Figure 2.4 Load vs. Deflection – ABAQUS Static Results and Published Data (Keith 2006)**
The published transverse bending experimental load-deflection results are the mean values obtained by Keith, 2006 from a series of three experiments on three separate panels. Small differences between the published deflection results and the ABAQUS CAE output results are expected. Three individual SIP panels were tested by Keith, and each of those transverse bending tests yielded slightly different results, as shown in Table 2.1. The measured load values between the three panels was then averaged. This mean load-deflection data was then compared to the ABAQUS CAE load-deflection data, as shown in Figure 2.4. Table 2.2 displays the percent error between the published deflection results from Keith, 2006 and the deflection results given by the FE model. The mean percent error value is 0.08%, verifying the accuracy of the FE-based modeling procedures used for this simulation.

<table>
<thead>
<tr>
<th>Load (lbf)</th>
<th>Deflection, Keith, 2006 (in)</th>
<th>Deflection, ABAQUS output (in)</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>946</td>
<td>-0.27</td>
<td>-0.30</td>
<td>0.11</td>
</tr>
<tr>
<td>1399</td>
<td>-0.40</td>
<td>-0.44</td>
<td>0.09</td>
</tr>
<tr>
<td>1852</td>
<td>-0.53</td>
<td>-0.58</td>
<td>0.08</td>
</tr>
<tr>
<td>2603</td>
<td>-0.80</td>
<td>-0.82</td>
<td>0.02</td>
</tr>
<tr>
<td>Mean</td>
<td></td>
<td></td>
<td>0.08</td>
</tr>
</tbody>
</table>

Table 2.3 Percent Error: ABAQUS vs Published Experimental Deflection (Keith 2006)
2.2 SIP Three Panel - Eigenvalue Analysis

A three panel assembly was then modeled to investigate the necessary modeling procedures to capture the subsystem response with multiple interacting parts. Three SIP panels of the same dimensions as the single panel from Part 2.1 (48 inches x 96 inches) were connected to one another using “tie constraints” in place of the complex cam lock geometry. Since the inelastic behavior of an individual cam lock is not within the scope of this paper, this FE simulation assumes that the individual panels do not separate. Therefore, the three individual panels are constrained to one another such that no relative motion is permitted between the individual 48” wide panels.
The three panels each consist of three 3D deformable solid parts, like the single panel simulation from Part 2.1. The modeling procedure utilizing “tie constraints” between the layers of the SIP panel was repeated for this case. The mesh for the three panel system is shown below in Figure 2.9. Analogous to the single panel case, each of these panels has a 2x4 timber nailer at the top and bottom of the panel.

![Figure 2.6 Mesh Size – Three SIP Panel Wall Subsystem, Simply Supported](image)

In accordance with Timoshenko’s free vibration plate theory, analytical results were calculated for a simply supported plate of consistent modulus of elasticity (Timoshenko 1959). For the FE Based model to satisfy the conditions of plate theory, the material must be a single plate that is homogeneous throughout. Plate theory also requires the FE based model to be a simply supported plate. The equation below is derived from Timoshenko’s plate theory for a plate simply-supported on all four sides and subjected to a uniform load, $q_0$, (Mullen 2014).
\[ w \left( \frac{a}{2}, \frac{b}{2} \right) = \frac{16 q_0 \cdot a^4 \cdot b^4}{\pi^6 D} \left( \frac{1}{\left( \frac{b}{a} \right)^2} + \frac{1}{\left( \frac{a}{b} \right)^2} \right)^2 - \frac{1}{3} \left[ \left( \frac{1}{a} \right)^2 + 9 \left( \frac{1}{b} \right)^2 \right] \left( \frac{1}{\left( \frac{b}{a} \right)^2} + \frac{1}{\left( \frac{a}{b} \right)^2} \right)^2 + \frac{1}{9} \left[ 9 \left( \frac{1}{a} \right)^2 + 9 \left( \frac{1}{b} \right)^2 \right] \left( \frac{1}{\left( \frac{b}{a} \right)^2} + \frac{1}{\left( \frac{a}{b} \right)^2} \right)^2 + \ldots \right) \]

\[ D = \frac{E \cdot t^3}{1 - \nu^2} \]

Table 2.7 contains the percent error between the natural frequency obtained from Timoshenko’s free vibration plate theory and the ABAQUS CAE frequency output. The three panel system is tied together in the ABAQUS model to behave as a single plate (144” x 96”), and thus satisfying the requirements of Timoshenko’s plate theory.

For comparison, the three panel SIP finite element model was modified so that the three layers of the SIP are homogeneous and retain the material properties of OSB. The tie constraints remain between the layers of 3D deformable solid elements, and the frequency analysis results from ABAQUS CAE were compared directly with free undamped vibration plate theory (Jawad 2004)(Timoshenko 1959). The results of these calculations are presented in Table 2.3.

<table>
<thead>
<tr>
<th>Free Undamped Vibration-Plate Theory</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gamma</td>
</tr>
<tr>
<td>Rho</td>
</tr>
<tr>
<td>Rho*(t)</td>
</tr>
<tr>
<td>omega</td>
</tr>
<tr>
<td>(f)</td>
</tr>
</tbody>
</table>

Table 2.4  Free Undamped Vibration Results – Plate Theory (Timoshenko 1959)

The percent error between the natural frequency of mode 1 for the free, undamped plate theory and the natural frequency given by the model is 0.08%, as shown in Table 2.4. The ABAQUS model is simply supported along each of the four edges of the three panel subsystem.
Eigenvalue Analysis Comparison

<table>
<thead>
<tr>
<th>Mode</th>
<th>$\omega_n$</th>
<th>Mode</th>
<th>$\omega_n$</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>70.265</td>
<td>1</td>
<td>64.41</td>
<td>0.08</td>
</tr>
<tr>
<td>2</td>
<td>115.24</td>
<td>2</td>
<td>n/a</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>165.31</td>
<td>3</td>
<td>n/a</td>
<td></td>
</tr>
</tbody>
</table>

Table 2.5  Abaqus CAE vs Plate Theory, Natural Frequency Comparison

Figures 2.7, 2.8 and 2.9 show the first three modes from the homogeneous three panel subsystem simulation described above.

Figure 2.7  Eigenvalue Analysis, Homogeneous Material, Mode 1
Figure 2.8 Eigenvalue Analysis, Homogeneous Material, Mode 2

Figure 2.9 Eigenvalue Analysis, Homogeneous Material, Mode 3
2.3 SIP Hut Building Subsystem - Dynamic Analysis

In April 2014 at the Aberdeen Proving Grounds, Maryland, explosives specialists from the United States Armed Forces and cadets from the USMA constructed the blast test layout based upon the research needs of the USMA. The test configuration is shown below in Figure 2.10 and Figure 2.11. The test included a conventional B-Hut constructed with timber and the proposed SIP Hut, equidistant from the test blast (IRC 2012). The United States Armed Forces currently uses this timber construction with typical stick-built guidelines for their conventional “B-Hut,” per the International Residential Code, 2010 (IRC 2012). The researchers from the USMA hoped to examine the comparative structural performance of the two types of B-Hut constructions during the course of the test.

Figure 2.10  Experimental SIP Test Hut, Aberdeen Proving Grounds, MD, 2014
Three blast tests were detonated using C4 explosives to represent a TNT equivalent charge of 2.25 lbs at three distances: 15ft, 12ft, and 6ft. During the experiment, the SIP structure showed no visible damage during the Z = 15ft blast tests, but showed significant structural damage during the 6ft test. So for the purposes of this paper, the effects of the blast from a distance of 6ft and 12ft (Shot 3 and Shot 2) are evaluated. The results of these blast tests include displacement data measured from high-speed cameras on-site and time-history pressure data measured from a series of pencil gauges. The data acquired by the pencil gauges provides the researchers with real-time pressure-time history data for the proposed blast size. The 120 mm high explosive mortar round was simulated using an equivalent amount of C4. The test involved the construction of both a typical timber construction “B-Hut” and the proposed SIP “B-Hut.” The huts were simultaneously subjected to a series of three successively closer blasts,
each 2.2 lbs of C4 (representative of a 120mm HE mortar round or 107 mm rocket). The first blast was triggered at a distance of 15ft from each hut. The next two blasts were triggered at distances of 12ft and 6ft, respectively.

The experimental SIP hut subsystem is constructed of 6 5/8” thick SIP Panels for the floor and roof, and 4 5/8” thick SIP Panels for the walls. Only the front wall and two supporting side walls, the floor and the roof were constructed. Wherever two SIP panels are connected laterally, there are cam-locks, provided by the manufacturer. The roof is anchored to the walls using 8” long hex screws. Also, the walls are also attached to the floor and at corners using the 8” long hex screws at 8” o.c. Wherever a cam-lock connection exists, a 3D solid cylindrical steel rod was “embedded” to represent the connection. Due to the complex geometry of the cam-locks, and because the scope of this report only addresses linear elastic response, detailed modeling of the connections was unnecessary. Similarly, for each 8” long hex screw, another cylindrical steel solid element was embedded. Section 3 of this report explores how these simplified elements may be utilized. See Figure 2.11 for layout.

The blast at a distance of 15ft from the exterior face of the SIP Hut revealed a failure of the cam-lock connection in the front wall, revealing a potential failure location and mode of the system design. At 6ft the blast resulted in the structural failure of the SIP Hut. Through examination of high speed video, it was apparent that the failure of the SIP Hut was the result of several progressive connection failures. The first failure began with the uplift of the roof from the wall facing the blast which enabled the failure of the 2x6 timber nailer attached along the top of the SIP Hut Walls like a header in traditional stick-built construction. Milliseconds
after this failure, the cam-lock connection between the SIP Panels of the wall failed, further compromising the structural integrity of the Hut.

The explosives specialists with the Aberdeen Proving Grounds provided the nodal displacement of a particular area of the SIP wall during the blast tests. The high speed cameras recorded nodal displacement for a set of 12 nodes that were labeled on the inside of the SIP Hut facing wall as in Figure 2.12.

![Figure 2.12 Node Labeling for the SIP Hut high-speed 3D deflection measurements, Aberdeen Proving Grounds, MD](image)

From Figure 2.14 and after considering the high speed video of the blast tests, it was determined that the nodes of most interest were nodes 3 and 12. Node 3 is located close to the location of the 2x4 timber header observed failing during Shot 3 of the blast testing. This failure was preceded by the uplift of the center roof panel from its attachment near the location of node 2. Nodes 6 and 7 are located on either side of the cam-lock connection that subsequently failed after the 2x4 header at the top of the wall failed.
To verify the validity of the results obtained from processing the pressure time history data and the deflection time history data provided by Aberdeen Proving Grounds, some comparative analytical calculations were made using common blast overpressure and time duration equations by Kinney and Graham and Kingery and Bulmash (Kingery, C.N, and Bulmash 1984; Kinney, G.F. and Graham 1985). Both Kingery-Bulmash and Kinney-Graham used experimental blast data from hemispherical blast tests to develop analytical equations to calculate the blast overpressure and time duration, \( t_d \). The Kingery-Bulmash analytical calculations are taken from the United Nations Kingery-Bulmash Blast Parameter Calculator (United Nations 2015). These equations are available in the Appendix.

<table>
<thead>
<tr>
<th></th>
<th>Kingery-Bulmash Equations (UN)</th>
<th>Kinney and Graham Equations, Kinney 1985</th>
<th>Blast Test Experimental Data, Shot 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>( P_0 )</td>
<td>20.79</td>
<td>14.94</td>
<td>20.60 psi</td>
</tr>
<tr>
<td>( T_d )</td>
<td>4.00</td>
<td>2.00</td>
<td>5.28 msec</td>
</tr>
<tr>
<td>Impulse</td>
<td>19.80</td>
<td>19.79</td>
<td>20.00 psi-msec</td>
</tr>
</tbody>
</table>

**Table 2.6 Analytical Blast Pressure Calculations vs. Measured Pressure Impulse Data, Shot 2**

As given by Chopra and Biggs, a blast loading is a dynamic loading that can generally be simplified into a triangular pulse load with \( P_0 \) (Overpressure) as the peak of the pulse load and \( t_d \) (time duration) representing the length of time that the triangular pulse load acts upon the system (Biggs 1964) (Chopra 2007). The figure below, image courtesy of Biggs 1964, graphically represents this simplified triangular impulse suggested by Biggs for blast loads. The results of the analytical blast equations by Kinney-Graham and Kingery-Bulmash are compared in Figure 2.14 with the experimental blast loading provided by Aberdeen Proving Grounds (Biggs 1964).
1964) (Kinney, G.F. and Graham 1985) (Kingery, C.N, and Bulmash 1984). Figure 2.14 compares the analytical results obtained from the blast equations by Kingery-Bulmash and Kinney-Graham to the truncated experimental pressure time history.

Figure 2.13  Simplified Triangular Pulse Response, image courtesy of FEMA, 2014
Interpretation of the experimental data was necessary during the building of the model in order to represent the nonlinear characteristics of the blast loading data efficiently. The ABAQUS model makes use of the experimental time history developed from the pressure and time duration data provided by Aberdeen Proving Grounds, 2014. The original pressure time history provided by Aberdeen contained over 100,000 data points. ABAQUS CAE cannot accommodate all of the available data points to define the amplitude of the pressure time history loading, so the original pressure time history data was truncated, see Figure 2.11.

The truncated pressure time history extracted from the second blast test at a distance of $Z = 12ft$ from the face of the hut was used to apply the blast loading to the SIP hut subsystem FE model. The third blast test exceeded the elastic yield limits of the subsystem and is
therefore outside the scope of work for this paper. Since no damage occurred as a result of blast 1, blast 2 pressure time history was utilized in the FE simulation.

The roof panels are anchored to the walls by 8” long hex screws, but in this ABAQUS model, these screws have been represented by an embedded rod to simplify the modeling procedure. These FE models only consider the linear elastic case, so if the connections fail, the subsystem becomes nonlinear. Plastic behavior is outside the scope so simplified embedded rods suffice for the purposes of this research. In addition, the technicians at Aberdeen Proving Grounds provided deflection time history data for a section of two of the wall panels. Each of the numbers visible in Figure 2.12 represents a node. For each node, a displacement time history was measured using high-speed cameras.

Figure 2.15  Rigid Body Motion Detected after Blast Test 2, April 2014, Aberdeen Proving Grounds, Maryland
The SIP Hut subsystem was not anchored during the blast experiments. Instead, the hut subsystem rests upon wooden timbers not anchored, either. Rigid body motion was noted after blast test 2, as shown in Figure 2.14. However, the FE simulation created in ABAQUS CAE represents the SIP hut subsystem as a fixed-base model. To account for the differences between the measured deflection time histories and the deflection time history provided by ABAQUS CAE, the rigid body motion was considered. Node 12 is located near the lower right-hand corner of the inside wall of the SIP hut subsystem and therefore the deflection measured at node 12 will more closely reflect the rigid body motion of the hut subsystem than any other node. Node 3 is located nearest to the observed location of failure from the experiments. The difference between the displacement time history of nodes 3 and 12 gives the rigid body motion of the hut subsystem. By subtracting the rigid body displacement at node 12 (represented by $\delta_{12}$) from the measured displacement at node 3 ($\delta_3$), the resulting
displacement time history, $\Delta_3$, closely represents the displacement time history output given by the SIP hut subsystem FE simulation, see Figure 2.15.

![Displacement Time History Comparison, Node 3](image)

**Figure 2.16 Displacement Time History Comparison, Experimental Results vs ABAQUS CAE Output with Rayleigh Damping**

The dynamic response given by the ABAQUS CAE model and shown in Figure 2.15 is the undamped response. Noting that the experimental data will include the effects of damping, the FE simulation was altered, adding Rayleigh damping. For demonstration, estimated values for the Rayleigh damping coefficients ($\alpha$ and $\beta$) were applied to the existing ABAQUS model. Figure 2.16 shows the effect that the application of Rayleigh damping to the undamped case affected the dynamic response. The resulting damped response was compared to the deflection, $\Delta_3$, estimated from the displacement time history experimental results of Node 3 and the estimated rigid body motion, the deflection at Node 12.
Figure 2.16 demonstrates the effects of damping on the displacement time history output from the ABAQUS CAE model. Figure 2.17 compares the adjusted displacement time history, $\Delta_3$, developed from the experimental results to the ABAQUS CAE output for the damped condition.

**Figure 2.17  Max Displacement @ Node 3, ABAQUS CAE**
3. CHAPTER 3

FE BASED RESPONSE EVALUATION FOR EXTREME WIND LOAD EVENT

3.1 System Response to High Velocity Wind Load

The finite element analysis model created in Section 2 of the Hut subsystem was used to evaluate the system response of the hut subsystem to a high velocity wind event. Hurricane and tornadic winds exhibit complex fluid dynamics behavior. For this paper, the wind loadings were simplified to constant static pressures based on ASCE 7-2010 wind design guidelines (ASCE 2010). The minimum design wind loadings specified by ASCE 7-10 provide variations in the pressure depending upon the surface of the structure (ASCE 2010). For instance, during a wind event, internal pressures will accumulate and act from the inside of the building, along with additional wind pressures applied at eaves and along roofs. However, the finite element model under consideration is a representative subsystem and is not enclosed. Therefore, internal pressures were disregarded.

With these considerations in mind, the following static pressures resulting from three hurricane strengths was considered, as shown in Table 3.1. The equation given by ASCE 7-10 was used to calculated the qz, the static windward wall pressure required by ASCE 7-10.
Table 3.1  Design wind pressures, qz (ASCE 2010)

The current hut subsystem design proves to be sufficient for the strongest considered wind load, 200mph.

Table 3.1  Design wind pressures, qz (ASCE 2010)

<table>
<thead>
<tr>
<th>Wind Speed (mph)</th>
<th>qz (psf)</th>
<th>qz (psi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>160</td>
<td>47.3</td>
<td>0.328</td>
</tr>
<tr>
<td>170</td>
<td>53.4</td>
<td>0.371</td>
</tr>
<tr>
<td>180</td>
<td>59.8</td>
<td>0.415</td>
</tr>
<tr>
<td>200</td>
<td>73.9</td>
<td>0.513</td>
</tr>
<tr>
<td>250</td>
<td>115.4</td>
<td>0.801</td>
</tr>
<tr>
<td>300</td>
<td>166.2</td>
<td>1.154</td>
</tr>
<tr>
<td>350</td>
<td>226.3</td>
<td>1.572</td>
</tr>
</tbody>
</table>

Figure 3.1  160 mph Wind Load Event
Figure 3.2 Maximum Displacement under 200 mph static wind pressure, Hut Subsystem

Figure 3.3 Recommended Design Modifications - Straps
3.2 Recommended Design Modifications

Evaluation of the finite element model of the hut subsystem suggested the need for transverse support along the windward face of the hut subsystem. Figure 3.5 shows one possible design modification to improve structural performance under extreme load events. In the blast experiment, the header along the top of the front wall was not continuous. Since this 2x4 timber header failed during Shot 3, the assumption is that this is an area of high stress concentration and therefore vulnerable to failure. Additionally, from the high speed video from the blast experiment, it was noted that the cam locks in the front wall and the 8” long hex screws attaching the roof panels to the front wall were also vulnerable. Adding (4) 24” long x ¼” thick x 2” wide steel straps (like Simpson Strong-Tie) to the model with 8d nails, as shown in Figure 3.3 lessens the Von Mises stress concentrations on the header and cam lock connections in the front wall. In addition, the 2x4 header should be made continuous, although that may not always be practical in construction.

Figure 3.4 Elements vulnerable to failure under an extreme loading
Applying an equivalent static pressure for a 200mph wind event produces an applied pressure of 73.9 psf. From the stress distributions, the vulnerable elements of the subsystem can be identified. The cam-lock connections in the windward wall show high stress concentrations. In this simulation, the complex geometry of the cam-lock is simplified to a steel rod, Figure 3.5. The maximum stresses on the rod can then be converted by the definition for stress to a resulting load on that element. By comparing these loads to the corresponding load capacities of the connectors will reveal if the connection will fail under the given loading. If the stress on the connection is greater than the capacity for that connection, failure will occur.

When the stress reaches the allowable yield strength of the material, plastic flow begins and the material is no longer elastic and the FE-based simulation is no longer valid.
Structural response predictions were made based on the blast experiments from Aberdeen. During the third shot of the blast experiment, progressive collapse occurs and the following failed structural elements are observed. First, the 8” long hex screws attaching the roof panels to the front wall pull out as the roof panels are pushed upward by the blast pressure. Next, the 2x4 timber header across the top of the front wall fails. Finally, the cam locks holding the front wall panels laterally fail. Since these connections failed during the blast experiment, they will also be vulnerable to the applied extreme wind loads.

![Figure 3.6 Stresses (S11), Steel Rod Representing Roof Cam-Lock, ABAQUS CAE](image)

The ABAQUS CAE simulation provides stress output data for each element. Figure 3.6 shows the stress distribution along the length of a representative steel rod used in the model for the roof cam-locks. The maximum stresses (S11) for the highlighted node are shown. Noting the coordinate system, the S11 stress shown in this figure represents the stresses on the rod that could lead to pull-out failure. If the pull-out capacity of the cam-lock is known, these pullout stress values (S11) may be compared to the pullout capacity of the cam-lock. If the
stresses exceed the pullout capacity of the cam-lock, the connection will fail. Table 3.2 summarizes the resulting stresses on the critical connections and presents the load corresponding to that stress value along with the mode of failure. If the pullout and shear capacities of the connections are known, the FE-based simulation can be used to determine if the connections are adequate.

<table>
<thead>
<tr>
<th>Critical Primary Stresses, ABAQUS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pullout Stress</td>
</tr>
<tr>
<td>$S_{11}$ (psi)</td>
</tr>
<tr>
<td>Windward Wall, Cam-Lock</td>
</tr>
<tr>
<td>Roof, Cam-Lock</td>
</tr>
</tbody>
</table>

*Table 3.2 Required Load Capacity, Critical Connections*
Experimental dynamic response data may be used to develop accurate modeling procedures for SIP and other composite material panel subsystems. FE-based response simulation of composite sandwich materials as 3D deformable solids with tie constraints between layers reveals locations of stress concentrations. The primary stresses obtained from the simulation can be converted to a required load capacity and then compared to the yield strength of the material to determine if the connection will fail. A verified model of a subsystem will allow accurate evaluation of both dynamic and static response from extreme loading such as high velocity wind loading. Minimizing the use of both “tie constraints” and “embedded constraints” will simplify the mesh and reduce the number of run-time errors in ABAQUS CAE. “Embedded Constraints” using single representative solid elements may be effectively used to model geometrically complex connections, yet still reveal the stress distribution at those connections.

In future, the tie constraints used between the layers of the composite sandwich material may be changed to allow for delamination and/or sliding effects. Instead of a tie constraint, a penalty friction constraint may reveal more complex system behavior. These modeling techniques may be used to evaluate the effects of changes in connector spacing and placement.
An evaluation of the number and locations of connectors will allow design optimization. The addition of more accurate Rayleigh damping coefficients will also improve the accuracy of the FE-based simulation.
LIST OF REFERENCES


2100eps.
LIST OF APPENDICES
APPENDIX A – BLAST EXPERIMENTAL DETAIL, ABERDEEN PROVING GROUNDS, MD
Courtesy of: Dr. Lt. Col. Steven Hart, (ret.), United States Military Academy, West Point
APPENDIX B – RAW AND TRUNCATED BLAST TIME HISTORY DATA
Truncated Blast Pressure Data: Shot 2, Gauge 3, R=12ft

\[ P_0 = 20.6 \text{ psi} \]
\[ t_d = 5.279 \text{ msec} \]

\[ P_1 = 12.52 \text{ psi} \]
\[ @ t = 0.293 \text{ msec} \]
APPENDIX C – BLAST EQUATIONS
Wave Front Parameters for Hemispherical Blast at Ground Level

References:

Kinney, C.M. and Bulmash, G. (1984), Ambient parameters from TNT: spherical air burst and hemispherical surface burst,  
Tech. Rep. ARBB6-TR-0050, Armament Research and Development Center, Ballistic Research Labs, Aberdeen Proving Grounds, MD

Adapted from Tadeo’s, USM Dissertation 2000

Revised by: Kimberly Tanner  
3/30/2015

Blast Parameters

<table>
<thead>
<tr>
<th>H</th>
<th>8 ft</th>
<th>2.4884 meters</th>
</tr>
</thead>
<tbody>
<tr>
<td>W</td>
<td>3 lbm (IW ca.)</td>
<td>1.360779 kg</td>
</tr>
</tbody>
</table>

\[ Z = \frac{2}{W(W^{20})^{0.5}} \text{ meter/kg}^{2} \]

\[ W = 3.16W \text{ (Baker 1973, Kinney and Bulmash 1984, Smith and Hetherington 1954)} \]

\[ P_{rev} = 103325 \text{ N/m}^2 (Psia) \]

\[ \frac{W}{s} = 5.3986016 \text{ lbm} \]

\[ R = 2.44902 \text{ kg} \]

\[ R = 3.6576 \text{ meters} \]

\[ Z = 8.0024153 \text{ ft} \]

\[ Z = 2.713373 \text{ meters} \]

Pressure: Formula Components:

\[ A = 1.00575027 \]

\[ B = 3.196-0.03038 \]

\[ C = 77.4809315 \]

\[ D = 6.09772525 \]

\[ 1037.36316 \]

\[ \beta \text{ = Overpressure ratio} \]

\[ P_{r} = 100051.994 \text{ N/m}^2 (Psia) \text{ psi} \]

\[ M_r = 1.360779 \text{ Mach number} \]

\[ T = 1.4 \text{ ratio of specific heats of air} \]

\[ \text{Reflected Pressure} \]

\[ P_r = \frac{28329.818 \text{ N/m}^2 (Psia)}{6.999} \]

\[ 41.99 \text{ psi} \]

\[ T \text{ given in seconds} \]

\[ E = 10506046.62 \]

\[ F = 2497156.45 \]

\[ G = 24213.382 \]

\[ H = 1.074562365 \]

\[ 1.54283617 \text{ psi} \]

\[ t_e = 0.0028 \text{ seconds} \]

\[ t_e = 2.077604915 \text{ msec} \]

\[ E = 10506046.62 \text{ psi} \]

\[ F = 2497156.45 \]

\[ G = 24213.382 \]

\[ H = 1.074562365 \]

\[ 1.54283617 \text{ psi} \]

\[ t_e = 0.0028 \text{ seconds} \]

\[ R = 19770.87427 \]

\[ B = 10.2909846 \]

\[ B = 197.298861 \]

\[ B = 2.582155 \]

\[ 7.3629941 \]

\[ 0.50842312 \text{ kg/sec/m}^2 \]

\[ 1979.243 \text{ lb-sec/in}^2 \]

\[ (Kinney and Graham 1980) \]

\[ R = 3.6576 \text{ meters} \]

\[ t_e = 0.0028 \text{ seconds} \]

**Kinney-Bulmash Results**

*Equations are not available (property of U.S. Army). Instead, values are calculated from log-log tables provided by U.S. Army, Kinney-Bulmash and UFC-1-340-62*

**GIVEN:**

\[ R = 12 \text{ ft} \]

\[ W_{net} = 2.45 \text{ kg} \]

\[ t_e (\text{time of arrival}) = 4 \text{ ms} \]

\[ \alpha (\text{shock front velocity}) = 56.62 \text{ m/s} \]

\[ P_r (\text{reflected pressure}) = 143.38 \text{ kPa} \]

\[ P_i (\text{incident pressure}) = 436.71 \text{ kPa} \]

\[ L (\text{incident impulse}) = 136.71 \text{ kPa-m} \]

\[ L (\text{reflected impulse}) = 340.15 \text{ kPa-m} \]

\[ t_{ref} (\text{positive phase duration}) = 0 \text{ ms} \]

\[ t_{ref} (\text{time of duration}) = 4 \text{ ms} \]

**58**
VITA

KIMBERLY TANNER

25700 Hwy 613  ●  Lucedale, MS 39452  ●  228-627-4775  ●  ktanner@go.olemiss.edu

EDUCATION
  M.S., Engineering Science, University of Mississippi, 2015
    Thesis: Computational Finite Element Analysis of Extreme Loading Response of
    Structural Insulated Panel (SIP) Building Subsystems
  B.S., Civil Engineering, University of Mississippi, May 2004

TEACHING EXPERIENCE
  Teaching Assistant, 2013-2014
    University of Mississippi
    Course: Mechanics of Materials, Soil Mechanics

  Research Assistant, 2013-2015
    University of Mississippi
    Dr. Christopher Mullen, Civil Engineering, NCITEC

HONORS AND FELLOWSHIPS
  Alumni Association, University of Mississippi, 2004-Present
  Chi Epsilon, Civil Engineering Honor Society, Secretary, 2003
  American Society of Civil Engineers, 2002-Present
  Society of Women Engineers, 2002-Present
  Gamma Beta Phi Honor Society, 1999
  Tau Beta Pi, Engineering Honor Society, 2003
  Golden Key International Honor Society, 2003-Present