Impact Simulation and Analysis of Sandwich Structures

Daewon Kim
*Embry-Riddle Aeronautical University - Daytona Beach*

Follow this and additional works at: https://commons.erau.edu/db-theses

Part of the Aeronautical Vehicles Commons, and the Structures and Materials Commons

**Scholarly Commons Citation**
https://commons.erau.edu/db-theses/270

This thesis is brought to you for free and open access by Embry-Riddle Aeronautical University – Daytona Beach at ERAU Scholarly Commons. It has been accepted for inclusion in the Theses - Daytona Beach collection by an authorized administrator of ERAU Scholarly Commons. For more information, please contact commons@erau.edu.
IMPACT SIMULATION AND ANALYSIS OF SANDWICH STRUCTURES

by

Daewon Kim

A Thesis Submitted to the Graduate Studies Office in Partial Fulfillment of the Requirements for the Degree of Master of Science in Aerospace Engineering

Embry-Riddle Aeronautical University Daytona Beach, Florida Spring 2006
INFORMATION TO USERS

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleed-through, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.
IMPACT SIMULATION AND ANALYSIS OF SANDWICH STRUCTURES

by

Daewon Kim

This thesis was prepared under the direction of the candidate’s thesis committee chairman, Dr. Yi Zhao, Department of Aerospace Engineering, and has been approved by the members of his thesis committee. It was submitted to the School of Graduate Studies and Research and was accepted in partial fulfillment of the requirements for the degree of Master of Science in Aerospace Engineering.

THESIS COMMITTEE

Dr. Yi Zhao
Chairman

Dr. Habib Eslami
Member

Dr. Frank Radosta
Member

Graduate Program Coordinator, MSAE

Department Chair, Aerospace Engineering

Associate Provost

S/10/06
Date

S/10/06
Date
I would like to thank my thesis committee, Dr. Yi Zhao, Dr. Habib Eslami, and Dr. Frank Radosta, for their advice and constructive criticism throughout my thesis research. A very big thank you goes to my advisor, Dr. Zhao, for not only financially supporting me through undergraduate and graduate school by means of teaching and research assistantships, but also showing me professionalism, integrity and kindness. Warm appreciation must go to Dr. Eslami for his encouragement and advice and Dr. Radosta for his generous support and time. It will long be remembered.

I also would like to thank my wife, Sunyoung, and my family for their love and support. Last but not least, I would like to thank my mother who is with Him. Without your love, I would not have made it here today.
Impact damage on aircraft and spacecraft can be a severe problem that may cause catastrophic results. Several aerospace structures have been used to resist or absorb energy in the occasion of impact. Among these, a sandwich structure has advantages with high stiffness, strength to weight ratio, and its relatively cheap cost. However, finding the impact structural response of a sandwich structure, by performing experiments, has disadvantages. First, impact is a complex problem with a multi-parameter nature and many factors should be considered. Secondly, it is a destructive test and the test subject may be damaged during the test. On the other hand, computer simulation based on a finite element code has advantages considering both the cost factor and technical issues associated with real tests.

The primary emphasis of this research is to establish the correlations between impact parameters and damage modes on sandwich structures using computer simulations. A finite element program, LS-DYNA, is used to perform impact simulations and to analyze results. A number of experiments are conducted to compare practical results with simulations.
# TABLE OF CONTENTS

Signature Page........................................................................................................ ii  
Acknowledgements.................................................................................................. iii  
Abstract.................................................................................................................. iv  
Table of Contents.................................................................................................... v  
List of Tables.......................................................................................................... vii  
List of Figures......................................................................................................... viii  

## CHAPTER 1 INTRODUCTION................................................................................. 1  
1.1 Background........................................................................................................ 1  
1.2 Impact, Crash and Penetration of Aircraft Structures................................. 2  
1.3 Objectives.......................................................................................................... 4  

## CHAPTER 2 METHODOLOGY............................................................................... 5  
2.1 General Description of Sandwich Structures................................................. 5  
2.2 Sandwich Structure Impact Parameters......................................................... 5  
2.2.1 Face Sheet and Core Materials................................................................. 6  
2.2.2 Core Configurations.................................................................................... 7  
2.2.3 Impactor...................................................................................................... 7  
2.3 Finite Element Analysis Software................................................................... 8  

## CHAPTER 3 EXPERIMENTAL STUDIES.............................................................. 10  
3.1 Honeycomb Sandwich Structures................................................................. 10  
3.1.1 Simulation Sandwich Model...................................................................... 11  
3.1.2 Results....................................................................................................... 19  
3.1.3 Discussions................................................................................................ 24  
3.2 Truss Core Sandwich Structure...................................................................... 25  
3.2.1 Plain Square Woven Core........................................................................ 25  
3.2.2 Simulation Model...................................................................................... 26  
3.2.3 Results...................................................................................................... 28  
3.2.4 Discussions................................................................................................ 29  

## CHAPTER 4 FEA SIMULATION OF SANDWICH STRUCTURES.......................... 30
LIST OF TABLES

Table 3.1  Aluminum 2024 properties .......................................................... 13
Table 3.2  Failure strain of 2024 aluminum alloy ........................................ 17
Table 3.3  Maximum deflections for different impact velocities ...................... 24
Table 3.4  Type 304 stainless steel properties ............................................... 27
Table 4.1  Stresses with respect to impact locations ...................................... 37
Table 4.2  Stress and deflection with respect to impact mass ........................... 39
Table 4.3  Core dimensions for different core heights ..................................... 41
Table 4.4  Core dimensions for different cell sizes ......................................... 44
Table 5.1  Bird model properties ................................................................. 61
Table 5.2  Wing properties ........................................................................... 62
LIST OF FIGURES

Figure 2.1 Sandwich structure with hexagonal honeycomb core ........................................ 5
Figure 2.2 Primary impact parameters for sandwich structure ........................................... 6
Figure 2.3 Tetrahedral truss core ....................................................................................... 7
Figure 2.4 Car crash simulation (left) and bullet penetration (right) ............................... 9
Figure 2.5 Flow charts and software for impact simulations ......................................... 9
Figure 3.1 Experimental setup ....................................................................................... 10
Figure 3.2 Honeycomb sandwich terminologies ............................................................ 11
Figure 3.3 Dimensions of honeycomb core .................................................................. 12
Figure 3.4 Impact tup model .......................................................................................... 13
Figure 3.5 Boundary conditions for face sheets; top (a) and bottom (b) ................. 14
Figure 3.6 Aluminum sandwich panel simulation model with impact tup .............. 15
Figure 3.7 Before and after tension test .......................................................................... 17
Figure 3.8 Strain rate effect on yield stress .................................................................... 18
Figure 3.9 Impact damage at velocity 1 m/s (a) FEA simulation (b) experiment .... 20
Figure 3.10 Impact damage at velocity 2 m/s (a) FEA simulation (b) experiment .... 21
Figure 3.11 Impact damage at velocity 3 m/s (a) FEA simulation (b) experiment .... 21
Figure 3.12 Impact energy under impact velocity of 1 m/s ........................................... 22
Figure 3.13 Impact energy under impact velocity of 2 m/s ........................................... 22
Figure 3.14 Impact energy under impact velocity of 3 m/s ........................................... 23
Figure 3.15 Crushing behavior of plain square woven core
   (Percent strains are indicated) ......................................................................................... 26
Figure 3.16 Boundary conditions for plain square woven core sandwich ............... 27
Figure 3.17 Simulation results of plain square woven core
   (Percent strains are indicated) ......................................................................................... 28
Figure 4.1 Impact results for case 1 (a), case 2 (b)
   Bottom view for both cases with penetration (c) ...................................................... 32
Figure 4.2 Penetration impact energy for 8.24 kg impactor mass (case 1) ............... 33
Figure 4.3 Penetration impact energy for 10 m/s impactor velocity (case 2) .......... 33
CHAPTER 1
INTRODUCTION

1.1 Background

Impact, crash, and even penetration on aircraft and spacecraft have recently become important issues due to the increasing safety concerns. For instance, according to International Bird Strikes Committee, over 195 people have been killed from 1988 to April 2005 as a result of bird strikes on aircraft [1-2]. In February 2003, the Space Shuttle Columbia broke apart during reentry resulting in the loss of all crewmembers. The ignition of the tragedy was the breach in the leading edge of left wing, initiated by the impact of thermal insulating foam that had separated from the orbiter's fuel tank. Due to the tremendous consequences of these accidents, impact related problems on aircraft and spacecraft have become important topics that cannot be ignored.

There have been numerous experiments pertaining to impact and crash responses of aircraft and spacecraft structures in order to prevent severe accidents and to develop more reliable structures. As a result of the Columbia tragedy, a structural analysis team conducted impact tests to determine the severity of the foam impact on a space shuttle. To do the test, they shot a 1.67 lb block of foam onto the lower portion of the leading edge [3]. Similarly, three 5 ft diameter composite fuselages were vertically dropped onto various impact media, such as water, soft soil, and hard surfaces, in order to design better energy absorbing fuselage configurations [4]. Furthermore, the Arnold Engineering Development Center of the U.S. Air Force used a chicken gun that could fire 4 lb thawed chicken carcasses at speeds up to 900 mph to simulate direct bird strikes on aircraft [5].
However, experimental investigations have major limitations. First of all, impact or crash problems are complex problems with a multi-parameter nature. Taking bird strikes as an example, locations of the strikes, size of the bird, strike speed, and strike angle must be considered to get accurate results. It is very unlikely that each of these parameters is considered in the experimental tests. In addition, an impact test is a destructive test, meaning that the test subject will be damaged during the test. Since the impacted subjects are aircraft structures, these tests are very costly, time consuming, and many are difficult to perform. On the other hand, computer simulation based on finite element codes has advantages considering both the cost factor and technical issues associated with real tests. Due to these advantages, the automotive industry has been using finite element analysis (FEA) to simulate various impacts, collisions, and crash scenarios. Although the real tests are conducted eventually, computer simulations have become standard procedures in automotive design. Unlike the automotive industry, however, not much research regarding impact has been carried out in the aerospace industry using FEA computer simulations, although there have been many occasions of impact related problems for aircraft and spacecraft operations.

1.2 Impact, Crash, and Penetration of Aerospace Structures

There are several structures that are designed to resist or absorb energy in the event of an impact, crash, and penetration of an aerospace application. Some of them are already being used, and others are under development. McCarthy et al. [6] investigated soft body impact problems on aircraft wings using fiber metal laminates that had high impact strength with lightweights. These structures have been applied to the upper
fuselage of the Airbus 380 and the bulk cargo floor of the Boeing 777. Ubels et al. [7] developed a tensor skin, which replaced brittle composite skin to polyethylene tensor loops, in order to induce skin elongation under impact. For space structures, a gas filled bumper shield concept is applied having pressurized gas to slow down or even melt impact debris and it theoretically redistributes impact energies in all directions [8]. Although these state-of-the-art concepts sound promising for impact problems, manufacturing cost is high, and fabrication processes are more complicated than typical sandwich structures that are commonly used. With the advantages in manufacturing, sandwich structures have high stiffness, strength to weight ratio, and are relatively inexpensive compared to the aforementioned high-tech structural concepts.

Not only are the sandwich structures applied to wing skins and floor panels, but are also used in almost every area of modern aircraft, including ribs, spars, control surfaces and doors. Front-face aircraft components including leading edges of wings, windshields, noses, and engine inlets are also important structural components where sandwich structure should be applied. The reason is that high velocity impact with flying objects including engine debris, hailstones, and even birds can cause fatal damage to aircraft structures. For space operations, orbital debris generated by spacecraft explosions and satellite collisions are another severe threat. The space shuttle orbiter uses aluminum honeycomb structures as skin covers for the intermediate wing section due to various advantages, including energy absorption. The structures of satellites are usually fabricated from aluminum honeycomb sandwich panels with composite face sheets due to their lightweight and superior debris protection [9].
1.3 Objectives

This thesis research is to perform computer simulations on various sandwich structures under different impact conditions. The main aim of the research is to establish the correlations between impact parameters and damage modes, such as penetration or core crushing, on sandwich structures. A finite element program, LS-DYNA, has been used to perform impact simulations and results analyses. A number of experiments are conducted to compare practical results with simulation models. In addition, computer simulation results are compared with other available experimental results. In order to establish the correlations, the research is focused on the structural behavior of the sandwich structure and different core configurations.

This research is distinguished from many other impact researches of sandwich structures. For example, prior research concentrated mainly on impact behaviors of sandwich structures with laminated composite face sheets; however, only a few have investigated the effects of non-composite face sheet sandwich structures. Likewise, impact responses of typical core structures, such as Nomex honeycomb or solid foam, have been studied in many papers, but not much research has been carried out on the response of core structures under a certain impact load when its core parameters vary.
CHAPTER 2
METHODOLOGY

2.1 General Description of Sandwich Structures

A typical sandwich structure consists of thin face sheets separated by a thicker core. Face sheets are made of high strength materials; such as laminated composites, aluminum alloys, or metals. Both top and bottom face sheets are in general identical in material and thickness. Although the core is relatively thick, its weight is fairly light. A core is usually made of honeycomb, lightweight foam, solid foam, corrugated core, or truss core. The most commonly used core in aerospace industry is Nomex honeycomb, which has been used broadly for interior panels. The most widely used adhesives to assemble face sheets and core materials are thermosets due to their high strength and temperature resistance. A typical hexagonal honeycomb sandwich structure is shown in Figure 2.1.

2.2 Sandwich Structure Impact Parameters

The impact damage in sandwich structures depends on various parameters. For instance, the damages developed due to impact are generally correlated to the materials used and geometries of both face sheets and core. Besides, mass and velocity of the
impactor are critical damage factors along with its impact angle. Impact parameters that can be considered as primary factors for sandwich structures are shown in Figure 2.2.

<table>
<thead>
<tr>
<th>Face sheet</th>
<th>Core</th>
<th>Impactor</th>
</tr>
</thead>
<tbody>
<tr>
<td>· Material</td>
<td>· Material</td>
<td>· Velocity</td>
</tr>
<tr>
<td>· Thickness</td>
<td>· Types</td>
<td>· Shape &amp; Size</td>
</tr>
<tr>
<td>· Configurations</td>
<td>· Mass</td>
<td></td>
</tr>
</tbody>
</table>

Figure 2.2 Primary impact parameters for sandwich structure

2.2.1 Face Sheet and Core Materials

For face sheets in aerospace applications, aluminum alloys, including 2024-T3, 7075-T6, or 2014-T6, are extensively used due to their good mechanical properties at reasonable cost. Stainless steel and titanium alloys may also be used for face sheets due to their high strength at elevated temperatures [10]. Materials for the core in sandwich structures should be sufficiently stiff in the direction normal to the plane of the face sheets to keep them apart. They also should have good shear strength to prevent face sheet sliding under certain bending load. Three types of primary cores, such as foam, honeycomb and truss core, are generally used. Foam cores consist of either metallic or polymeric foams, with Polyvinyl chloride (PVC) being one of the commonly used. For honeycomb and truss core with non-composite face sheets, aluminum and steel are normally used due to their simple fabrications.
2.2.2 Core Configurations

While foam cores simply fill the space between face sheets, honeycomb and truss cores have various configurations. The most popular honeycomb structure is the hexagonal honeycomb, and others include triangular and square honeycombs. The parameters that should be considered in honeycomb sandwich design are cell size, cell thickness, cell angle, and core height. Truss core is currently of interest as it has some advantages over the honeycomb core. For anticlastic curvature, such as the leading edge of a wing or engine inlets, the hexagonal honeycomb core is not an adequate choice due to its difficulty of forming into complex shapes. Unlike the honeycomb, the open nature of a truss core can avoid moisture and heat trap that leads to corrosion [11]. There are a variety of truss core topologies, including tetrahedral, pyramidal, and diamond. Tetrahedral cores with triangulated architecture, as shown in Figure 2.3, are among the most structurally efficient structures due to their favorable strength to weight geometry [11]. Similar to honeycomb structure, parameters like tetrahedral unit length, cross-section dimensions, and height should be considered.

2.2.3 Impactor

One of the critical factors for impact response of sandwich structures is an impacting object. Results of the impact depend highly on the parameters; such as

Figure 2.3 Tetrahedral truss core
velocity, mass, size and shape of the impactor. Angle of impact also affects the
deformation process and energy developments during impact.

2.3 Finite Element Analysis Software

In this thesis, special finite element analysis (FEA) software, LS-DYNA, has been used to simulate and analyze impact behaviors on sandwich structures. LS-DYNA was originally developed as DYNA3D in the mid-seventies at Lawrence Livermore National Laboratory. Considerable progress has been made since the Livermore Software Technology Corporation was founded to continue the development of DYNA3D, which was later called LS-DYNA.

Based on theories of elasticity, plasticity, hyperelasticity, viscoelasticity and impact theory etc., LS-DYNA can be utilized to solve difficult dynamic nonlinear problems. This FEA software is particularly effective in dynamic problems associated with short duration simulations; such as impact, crash, penetration, and explosion.

The main applications of LS-DYNA include vehicle design in the automotive industry. The program accurately simulates a vehicle’s behavior in collision and crash scenarios. Automotive companies can test vehicle designs without performing experimental tests on prototypes. Although the real tests should be conducted eventually, a great deal of analysis are done before the new vehicle’s production, saving time and expenses. The program has also been used for military and defense applications, simulating projectile penetrations, blast responses, and explosives.
Furthermore, the program is widely used in the aerospace industry. From a jet engine fan blade containment analysis, to horizontal tail-plane impact analysis, to accident investigation of the space shuttle Columbia tragedy, the LS-DYNA FEA simulation is being increasingly used. Due to its superior ability to conduct computer simulations and analysis associated with impact related problems, applications are not limited in particular types, but are tailored to various fields.

As LS-DYNA is a transient dynamic FEA program, the pre-processing for geometrical modeling and post-processing for analysis are performed using other FEA software. For this thesis project, LS-DYNA is used with FEMB-PC by Engineering Technology Associates, Inc. as a pre-processor and LS-PREPOST as a post-processor. A flow chart for the analysis process is shown in Figure 2.5.
CHAPTER 3
EXPERIMENTAL STUDIES

In order to validate impact results of LS-DYNA simulation models, two experimental results were compared with those of simulation models. First, a number of aluminum honeycomb sandwich structures were hit by a heavy mass impactor with different impact velocities. Secondly, a tested crushing behavior of plane square weave sandwich structure was compared with the simulation result.

3.1 Honeycomb Sandwich Structures

Impact tests are conducted by applying impact loads on the aluminum 2024 honeycomb sandwich structures. For the experiments, nine identical sandwich structures are tested with different impact velocities. The sandwich structures are clamped by the pneumatic clamping fixture of the Instron Dynatup impact machine as shown in Figure 3.1. The sandwich structures used in the experiment consist of two thin aluminum face sheets separated by the same aluminum honeycomb core. The honeycomb core used has the hexagonal shape that is the most popular type for modern aircraft components.

Using the Instron Dynatup impact machine, the sandwich structures are vertically impacted with three different low velocities, by a 16 mm diameter...
spherical tup insert with a drop weight of 8.243 kg. The experimental model is implemented in LS-DYNA to simulate impact tests and to compare the results. The final results show that the simulation models in LS-DYNA experience similar energy developments, as well as total deflections, when compared to the experimental results.

3.1.1 Simulation Sandwich Model

A. Geometry and Property of Sandwich Model

The tested honeycomb sandwich structures were obtained from the Aviation Maintenance Technology Department at Embry-Riddle Aeronautical University. It was known that the sandwich structures were made of aluminum 2024 alloy. Figure 3.2 shows the general terminologies of a honeycomb sandwich structure.

![Figure 3.2 Honeycomb sandwich terminologies](image)

Note that core height in Figure 3.2 is generally called core thickness; however, this term is used to avoid confusion with cell thickness.
Figure 3.3 describes the dimensions of a honeycomb core between face sheets obtained from the department.

All dimensions of core and face sheets are measured with a digital caliper and a three dimensional stereo-zoom microscope. It should be noted that the cell thickness in the ribbon direction is twice that of the transverse direction, due to the honeycomb manufacturing process. After sheets of flat strips of adhesives having a staggered pattern are stacked together and cured, they are expanded to form a honeycomb. Since the adhesives in the ribbon direction bonds two sheets together, its thickness becomes twice that of the transverse direction. Due to the slight inconsistencies of the thicknesses from the manufacturing process, the average thickness values taken from five different locations are used for the simulation models. Physical and material properties of aluminum 2024 honeycomb and face sheets are given in Table 3.1. The properties are obtained from the MIL-HDBK-5.
Table 3.1 Aluminum 2024 properties

<table>
<thead>
<tr>
<th>Properties</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (kg/mm³)</td>
<td>2.77 x 10^-6</td>
</tr>
<tr>
<td>Elastic Modulus (MPa)</td>
<td>73,000</td>
</tr>
<tr>
<td>Yield Strength (MPa)</td>
<td>318</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.33</td>
</tr>
</tbody>
</table>

B. Hemispherical Impact Tup

The impactor in the experiment consists of three components; a frame with drop weights, a reaction plate with tup, and a tup insert. The total weight of the impactor is 8.243 kg. In order to simplify modeling and to save computer processing time, only the bottom portion of the tup insert, in which the real impact occurs, is modeled in the computer simulation, as shown in Figure 3.4. The total weight of the impact assembly is applied into the hemispherical solid tup insert by density manipulation. The density value for this simplified model is $7.94 \times 10^{-3}$ kg/mm³ and the diameter of the tup insert is 16 mm. This hemispherical tup insert travels down and impacts the sandwich panel with three different velocities for each trial; 1 m/s, 2 m/s and 3 m/s, respectively.

Figure 3.4 Impact tup model
C. Boundary Conditions

All tested sandwich panel specimens were clamped by the pneumatic clamping fixture of the Instron Dynatup impact machine as shown in Figure 3.1. A clamping force around 3500 N is applied in order to hold the specimen under the impact loading. There is an unsupported circular region with a 76.2 mm diameter in the middle of the testing area. In order to apply the same boundary conditions to the simulation, the nodes of outer edge lines on the top face sheet are constrained in the x and y directions to prevent the specimen’s movement as shown in Figure 3.5 (a). In addition, from Figure 3.5 (b), the bottom face sheet not in the unsupported circular region is constrained in the z direction, as well as its three axes of rotation, in order to hold the supported bottom face and to prevent the specimen’s rotation under the impact loading, respectively.

Figure 3.5 Boundary conditions for face sheets; top (a) and bottom (b)
The simulation sandwich model shown in Figure 3.6 is constructed by bonding the share nodes of core and face sheets together. The average rectangular element size of the core is 2 by 2 mm and the face sheet is 1.7 by 2 mm. The hemispherical impactor, shown in the center, collides into the sandwich panel’s top face sheet with different impact velocities.

![Figure 3.6 Aluminum sandwich panel simulation model with impact tup](image)

**Figure 3.6 Aluminum sandwich panel simulation model with impact tup**

D. Material Model Definitions

As seen in previous research of Anghileri et al. [9] and Loikkanen et al. [12], a couple of LS-DYNA material models have been used for impact on aluminum alloy structures. The Johnson-Cook (MAT 15) and the piecewise linear plasticity (MAT 24) are the material models used in the research. The Johnson-Cook material model is primarily used for large strains, high strain rates, and high temperatures; such as ballistic penetration or explosive metal forming. This model defines the yield strength in terms of plastic strain, strain rate, and temperature. Although the LS-DYNA keyword user
manual [13] mentions that this material model remains valid down to lower strain rates, it is found that this model is not suitable for this problem after several simulations were performed. Meanwhile, the piecewise linear plasticity is a general material model mainly used for metals, taking into account strain rate effects. For this relatively low velocity impact simulation, the latter model is used to represent thin aluminum 2024 honeycomb sandwich structures.

There are several parameters in the piecewise linear plasticity model that should be appropriately defined in order to achieve more accurate results. With material data previously mentioned, such as density, elastic modulus, yield strength, and Poisson's ratio, two more parameters of failure strain and strain rate effect should be considered. First, the failure strain should be set based on the reduction in cross-sectional area when the test specimen fails due to tensile load. Secondly, strain rate effect should account for the material behavior since mechanical properties, such as yield strength, are changed due to the initial impact conditions.

The failure strain can be obtained from a simple tension test where the strain can be read at failure point. However, Weimar [14] gives a recommended formula to be used to get the failure strain,

\[
\text{Failure strain} = \ln \left( \frac{\text{original area}}{\text{area at failure}} \right) \quad (1)
\]
Using the Tinius Olsen tension and compression testing machine, a quasi-static tensile load was applied to aluminum 2024 bar specimens in order to get the failure strain. Four specimens were used to determine the failure strain of the aluminum alloy. The results are shown in Table 3.2. The average failure strain is found to be 0.315.

Table 3.2 Failure strain of 2024 aluminum alloy

<table>
<thead>
<tr>
<th>Tests</th>
<th>Original area (mm$^2$)</th>
<th>Area at failure (mm$^2$)</th>
<th>Failure strain</th>
</tr>
</thead>
<tbody>
<tr>
<td>Specimen #1</td>
<td>56.7</td>
<td>41.6</td>
<td>0.310</td>
</tr>
<tr>
<td>Specimen #2</td>
<td>56.6</td>
<td>41.1</td>
<td>0.320</td>
</tr>
<tr>
<td>Specimen #3</td>
<td>56.7</td>
<td>41.5</td>
<td>0.312</td>
</tr>
<tr>
<td>Specimen #4</td>
<td>57.6</td>
<td>41.9</td>
<td>0.318</td>
</tr>
<tr>
<td>Average</td>
<td></td>
<td></td>
<td>0.315</td>
</tr>
</tbody>
</table>

Strain rate effect is another parameter that should be considered. Although the elastic modulus, yield strength, and tensile strength of a material are independent properties that do not change upon static loading, they can generally be increased as the strain rate increases. Therefore, it is necessary to represent the material's behavior in accordance with strain rate, in order to achieve correct results. Johns [15] discusses constitutive equations for the strain-rate-sensitive behavior of some materials. Among those, the Cowper and Symonds constitutive equation can be used since the piecewise linear plasticity material model allows inputs of Cowper and Symonds strain effect parameters.
Boner and Symonds [16] presented Cowper and Symonds coefficients for aluminum alloy that were used in the equation,

$$\dot{\varepsilon} = D \left( \frac{\sigma_y}{\sigma_o} - 1 \right)^q, \quad \sigma_y \geq \sigma_o,$$

where $\dot{\varepsilon}$, $\sigma_y$, and $\sigma_o$ are strain rate, dynamic yield stress, and static yield stress, respectively. The parameters $D$ and $q$ are empirical constants for a particular material. The stress-strain rate curves and analytical approximation for mild steel and aluminum alloy are shown in Figure 3.8 [16].

![Figure 3.8 Strain rate effect on yield stress](image)

The behavior of the aluminum alloy is essentially strain-rate-insensitive, as shown in figure 3.8. The yield stress does not change much as strain rate increases for the aluminum alloy. The parameters $D$ and $q$ for aluminum are 6500 and 4, respectively. These constants are the last input parameters of the material model data.
3.1.2 Results

For most impact problems, simulations are started with an initial kinetic energy, no internal energy or so called strain energy, and no external works such as forces or pressures are normally applied. During impact, the kinetic energy will decrease, but the internal energy will increase. Some energy will be transformed to other types; such as sliding interface (friction) or damping energy. Although these transformed energies are part of the total energy, the amount of transformed energies is usually not significant compared to the kinetic and internal energy. The kinetic energy will decrease in most cases due to the reduced velocity of the impactor during collision. The internal energy, however, will increase since the impact load will be absorbed by the target deforming its unstrained state. Therefore, evaluating internal energy, or strain energy, is rational to look at how much damage is done to the object.

In this section, final deformation figures are juxtaposed to compare the simulation and experimental models. Furthermore, the phase development of internal energy for each model is evaluated at each impact velocity. Then, the final deflections due to impact are compared and discussed.

A. Damage Comparison

For the sandwich panel with 1 m/s impact velocity, the tested structure sustained damage that was not readily noticeable. In the experiment, the front face sheet had an obvious small bowl-shaped depression that was similar in shape to the tup's hemispherical head. The simulation model also made a similar spherical depression like
the one shown in the experiment. Both simulation and experimental impact results are shown in Figure 3.9.

Figure 3.9 Impact damage at velocity 1 m/s  
(a) FEA simulation  (b) experiment

For the sandwich panel with 2 m/s impact velocity, the tup’s spherical head split open the contact surface of the top face sheet in both simulation and experiment. It penetrated into the middle point of the core height as shown in Figure 3.10. The back side of the bottom face sheet protruded a little (1 to 2 mm) due to the impact load for both the simulation and the experimental models.
A penetration occurred during the 3 m/s velocity test, as well as in the simulation, as shown in Figure 3.11. Aluminum core cells were crushed and split. Eventually, the impact made a hole at the center of the sandwich panel.
B. Energy Comparison

The internal energy from the computer simulation closely matches the experimental results. The following three figures show the internal energy formation processes as testing time increases. The energy curves in the beginning increase as time passes by, then respond according to the structures’ behavior. It should be noted that there were two energy humps in Figure 3.14 because the impact tup went through the top and bottom face sheets.

![Figure 3.12 Internal energy under impact velocity of 1 m/s](image1)

![Figure 3.13 Internal energy under impact velocity of 2 m/s](image2)
C. Deflections

The maximum deflections from FEA simulations appeared to be lower than the experimental results for 1 m/s and 2 m/s. The deflection results in millimeters with percentage differences are shown in Table 3.3. The third deflection value for the 2 m/s experiments seems larger than the other two in the column, possibly due to the impacting location of the honeycomb cell. It has been observed that when the impact tup hits the wall-edges of the honeycomb core cell, such as the simulation and the first and second experiments for the 2 m/s impact case, the deflections seemed to be smaller than when it hits the empty center of the individual honeycomb cell. In Table 3.3, the maximum deflections were measured from the original state of the top face sheet to the impactor’s traveled distance into the sandwich structure.
Table 3.3 Maximum deflections for different impact velocities

<table>
<thead>
<tr>
<th>Impact velocity (m/s)</th>
<th>Maximum deflection (mm)</th>
<th>% Difference (Average)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Simulation</td>
<td>Experiments</td>
</tr>
<tr>
<td>1</td>
<td>2.74</td>
<td>3.01</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3.24</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3.16</td>
</tr>
<tr>
<td>2</td>
<td>6.84</td>
<td>7.47</td>
</tr>
<tr>
<td></td>
<td></td>
<td>7.29</td>
</tr>
<tr>
<td></td>
<td></td>
<td>8.24</td>
</tr>
<tr>
<td>3</td>
<td>Penetrated</td>
<td>Penetrated</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Penetrated</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Penetrated</td>
</tr>
</tbody>
</table>

3.1.3 Discussions

As shown in the internal energy graphs and the deflection results table, there are slight differences between experiments and simulations. Many factors can be considered to cause the differences. One of them may be the thickness used in the simulation. The thicknesses measured were not consistent at all locations and it was also difficult to measure due to the relatively thick adhesives on the inside surface of the face sheets. For the simulation models, perfect bonding between core and face sheets was assumed so that the exclusion of adhesives made them around 3 grams lighter than the experimental models (37 grams). In addition, the boundary conditions may not be the same. Only the outer edge nodes of the top face sheet were constrained in the simulation models; however, all of the areas except for the unsupported circular region of the top face sheet are clamped in the experiments, as shown in Figure 3.1. The clamped condition may also be a factor. The simulation models were impacted under the perfect clamping condition; however, the experiment models could be clamped loosely during the impact.
In spite of the slight differences, internal energies due to impact are developed in a similar way for both simulations and experiments. Energy stored in sandwich structures is lower for the simulation than the ones for the experiments. This means that simulation models are less deflected when compared to the experimental results. From Table 3.3, it clearly shows that the simulation deflections are smaller than the ones for the experiments, which differs by less than 15 percent between them. Overall, the simulation results can be concluded as reasonable results closely matching to the experiments.

3.2 Truss Core Sandwich Structure

A sandwich structure with tetrahedral truss core is also examined to study impact responses on various sandwich structures. In order to examine the physical behavior of a truss core during impact, the crushing behavior of a plain square woven core sample made in this truss-like fashion was compared with a numerical simulation model.

3.2.1 Plain Square Woven Core

The crushing behavior of a plain square woven core sandwich was investigated by Caulfield et al. [17] using pre-crimped 304 stainless steel wires. This 2-dimensional sandwich structure was quasi-statically compressed to quantify mechanical performance on transverse loading. Snapshots were taken during deformation to compare the undeformed configuration (0%) to the maximum deflection (70%), as shown in Figure 3.15.
3.2.2 Simulation Model

As shown in Figure 3.15, only one layer of the structure is examined. Although the experiment was performed by quasi-statically compressing the top face sheet down slowly, the simulation was done in fairly short period. The main reason is that the crushing behavior of the core, i.e. core overlapping at the center first then corrugating over the entire core, is the primary focus of this study. In addition, computer processing time can be saved due to its shorter time duration.

A. Geometry and Boundary Conditions

The core wire is modeled as a rectangular shape, each side having 1 mm in length. However, the actual specimen had a circular cross-section with a diameter of 1.2 mm. The thickness of the face sheet is 2 mm and the core height is 50 mm. As shown in Figure 3.16, the nodes of the bottom face sheet are fixed for the three axes translation and rotation, but the top face sheet moves down, following the negative z direction.
B. Material Model Definitions

For 304 stainless steel, the piecewise linear plasticity (MAT 24) used in the honeycomb sandwich model is employed. The material data is shown in Table 3.4.

<table>
<thead>
<tr>
<th>Properties</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (kg/mm³)</td>
<td>8.00 x 10⁻⁶</td>
</tr>
<tr>
<td>Elastic Modulus (MPa)</td>
<td>200,000</td>
</tr>
<tr>
<td>Yield Strength (MPa)</td>
<td>215</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.29</td>
</tr>
</tbody>
</table>

The failure strain for 304 stainless steel is reasonably assumed by comparing the elongation at break parameters of two materials, i.e. aluminum 2024 and 304 stainless steel. According to the appendix in Callister [18], the percent elongation of aluminum 2024 is 18% and 304 stainless steel is a minimum 40%, which indicates that 304 stainless steel is more ductile than aluminum 2024. Since the 304 stainless steel stretches more in...
tension, its area at failure should be smaller than the one in aluminum. From Eq. (1), therefore, the failure strain of the 304 stainless steel will be larger due to the reduced area at failure. For this material, the value 0.5 is assumed as the failure strain, considering aforementioned material characteristics.

The strain rate parameters of 304 stainless steel, as in Eq. (2), are obtained from Figure 3.8. Although these parameters are used for the mild steel, they are also used for 304 stainless steel because two steels are similar materials. Therefore, the parameters \( D \) and \( q \) are the same, i.e. 40.4 and 5, respectively.

3.2.3 Results

Four snapshots taken for different percentage strains are shown in Figure 3.17. The core at the center first overlaps, like shown in Figure 3.15, then corrugation extends to the outer edges as the top face sheet is compressed down.

![Figure 3.17 Simulation results of plain square woven core (percentage strains are indicated)](image-url)
3.2.4 Discussion

In general, a fairly good correlation can be established between the test results and the simulations, although there is a slight difference in deformation. A possible reason for this is due to the material behavior under compressive loading, in which the practical test was done quasi-statically, but the simulation was performed in a relatively short time period. In addition, the wire configuration used in the experiment is circular with a diameter of 1.2 mm; however, the simulation model uses a rectangular shape with each side being 1 mm in length. Lastly, the strain rate parameters used for 304 stainless steel may not be appropriate. Overall, the crushing behavior of the core is basically the same between the test and simulation models, showing overlapping behavior in the center and then spreading to the outer directions.
CHAPTER 4

FEA SIMULATION OF SANDWICH STRUCTURES

This chapter consists of three sections that investigate structural impact responses of sandwich structures in accordance with changing their parameters. In the first section, two different studies are investigated. First, impact energies of two identical sandwich structures are compared, changing only the impactor’s parameters. The kinetic energies of the impactor are unchanged, manipulating mass and velocity inversely. Second, the elastic behavior of the structure is studied by observing stresses and deflections when the initial impact force is applied. In the second section, impact resistance of the structure is investigated by changing core heights and cell sizes at which other parameters remain unchanged. Finally, other types of core configurations other than honeycomb cores are also studied. A tetrahedral truss core and foam core are adopted as core configurations, without changing other parameters.

4.1 Structural Response Study

4.1.1 Impact Energy

Impact energy is described as the amount of energy required to fracture a material subject to a shock loading. Charpy, Izod, and drop weight impact tests are generally used to get impact energy of a material. For a free-falling impactor case, the basic principle is that initial potential energy will be converted to maximum kinetic energy when the impactor collides into a target. This kinetic energy is converted to impact energy when a
material is fractured. If there is an initial kinetic energy, impact energy is defined as potential energy and kinetic energy, ignoring friction and sound energies produced by the impact. In the following impact simulations to compare impact energies, the potential energy of the impactor is ignored due to the very short distance between the impactor and the target structure. Therefore, the impact energy is assumed to be calculated only from the kinetic energy.

In the kinetic energy, two terms are considerable elements, i.e. the mass and velocity of the impactor. It should be noted that the same kinetic energy can be calculated from manipulating these terms inversely. For instance, the same kinetic energy can be achieved using parameters from a heavy-mass and low-velocity (case 1), to a light-mass and high-velocity (case 2). Therefore, it is interesting to know which case would cause more damage on the structure under the same impact energy.

A. Simulation Models

The honeycomb sandwich structure used in the previous study is implemented in this comparison. All dimensions of the sandwich structure are the same as the one used in the experiment. Only the impactor's parameters are changed for both impact cases. The first case fixes the mass (8.243 kg) of the impactor and varies the impact velocity. On the other hand, the second case fixes the velocity (10 m/s) of the impactor and varies its mass.
B. Results

In order to compare the two cases, the bottom face sheets of both honeycomb sandwich structures are examined. The impact energy is calculated when the initial penetration is observed on the bottom face sheet. The range of impact energy from 20 J to 40 J for both cases is investigated. The initial bottom face sheet penetrations are juxtaposed as shown in Figure 4.1 for both cases.

![Figure 4.1 Impact results for case 1 (a), case 2 (b), Bottom view for both cases with penetration (c)](image)

Both sandwich structures penetrated on the top face sheets and perforated slightly on the bottom face sheet, after elongating about 5 mm down from the original flat bottom. These two simulation results are identical so that the impact energy at this state can be compared. The impact energy for case 1 is shown in Figure 4.2, and the case 2 impact energy is shown in Figure 4.3.
Figure 4.2 Penetration impact energy for 8.24 kg impactor mass (Case 1)

Figure 4.3 Penetration impact energy for 10 m/s impactor velocity (Case 2)
In the simulations, the penetration on bottom face sheet was first observed with the heavy-mass and low-velocity case at 32.89 J. On the other hand, the penetration was observed at 36.47 J for the other case. From these results, assuming all parameters are fixed except the impactor, it can be said that more damage may occur when the impactor is heavy and its impact velocity is low if applied impact energy is the same.

4.1.2 Elastic Behavior

For an acute impactor, such as a broken part of an aircraft having a sharp edge or tools dropped from a certain height, the damage would be seen first on the surface where it impacts. In other words, the face sheet would fail first if the sharp edge of the impactor hits somewhere between each core cell wall. On the other hand, the core would fail first if the impacted point in the face sheet is located on the edges of core walls. However, the initial damage caused by a blunt-body impactor, such as a bird, would be different. The damage modes would no longer depend on the shape of the impactor, rather it depends on the force produced by it. In this study, therefore, an obtuse impactor having a smooth-continuous surface impacts on the sandwich structure with a relatively high speed (10 m/s), but with light mass, in order to examine initial damage modes. Two types of cell size are investigated, i.e. 3 mm cell size and 7.5 mm cell size sandwich structures.

A. Geometry and Property of Sandwich Structures

The core dimensions for both sandwich structures are shown in Figure 4.4. The average element size of the core is 1.5 by 2.4 mm and the face sheet is 1.3 by 1.5 mm. The masses of both sandwich structures are 147 grams and the top and bottom face sheet
thicknesses are the same (0.56 mm). The FT/CT ratio, where FT is face sheet thickness and CT is cell thickness (ribbon direction), is 7.4 for the 3 mm cell sandwich structure and 2.9 for the 7.5 mm sandwich structure. Aluminum 2024 is used and the material properties are the same shown in Table 3.1.

![Figure 4.4 Dimensions of honeycomb core](image)

**Figure 4.4 Dimensions of honeycomb core**

B. Boundary Conditions

All edges on both face sheets are constrained in a way to prevent translational and rotational movements of the sandwich structure. The nodes around the edges on the top face sheet are constrained in the x and y directions. In addition, the outside nodes on the bottom face sheet are also constrained in the z direction and on the three axes of rotation.
C. Impactor

To avoid the stress concentration due to a pointed edge of an impactor, it is assumed that the impactor does not have a sharp point. Rather, the contact surface is continuous so that it could represent a blunt body. In this study, the portion of a sphere with a 500 mm radius is assumed to represent such an impactor. The impact velocity is 10 m/s, with varying masses from 1 gram to 10 grams. The side view of the impactor with a sandwich structure is shown in Figure 4.5.

D. Simulations

In order to compare the effect of impact location on damage modes, two impact points were chosen. The first impact point was located in the center of the core cell (red circle) while the second point was on the edge of the cell walls (blue circle), as shown in Figure 4.6. Once it was found that the damage modes are not relevant to the impact points, the rest of the simulations were performed assuming that the impact occurred in the center of the core cell (red circle) for both sandwich configurations. A total of 12 simulations were carried out, 6 for each configuration, varying the impactor’s mass from 1 gram to 10 grams and fixing the velocity (10 m/s). The size of the elements in the sandwich structure is restrained, each having 1.5 mm by 2.4 mm in length.
E. Results

As a result of the simulations that compared the locations of the impact, it is observed that the core always fails first, whether the impact occurred at the center of core cell or on the edges of the core walls. No matter where it hits, the core fails prior to the face sheet, i.e. the core elements surpass the yield strength of the material (318 MPa) first, as shown in Table 4.1.

Table 4.1 Stresses with respect to impact locations

<table>
<thead>
<tr>
<th>Impactor mass (g)</th>
<th>Impact location</th>
<th>Cell-center (red circle)</th>
<th>Cell-wall (blue circle)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Cell size (mm)</td>
<td>3</td>
<td>7.5</td>
</tr>
<tr>
<td></td>
<td>Location</td>
<td>Core</td>
<td>Face</td>
</tr>
<tr>
<td>1</td>
<td>Max. stress (MPa)</td>
<td>343.0</td>
<td>274.0</td>
</tr>
<tr>
<td></td>
<td>Core</td>
<td>233.4</td>
<td>199.7</td>
</tr>
<tr>
<td></td>
<td>Face</td>
<td>407.4</td>
<td>301.8</td>
</tr>
<tr>
<td>2</td>
<td>Max. stress (MPa)</td>
<td>470.2</td>
<td>203.3</td>
</tr>
<tr>
<td></td>
<td>Location</td>
<td>Core</td>
<td>Face</td>
</tr>
<tr>
<td></td>
<td>Core</td>
<td>470.2</td>
<td>203.3</td>
</tr>
<tr>
<td></td>
<td>Face</td>
<td>484.3</td>
<td>237.2</td>
</tr>
</tbody>
</table>

*Bold* marks for the stresses that exceed the yield strength

Table 4.1 clearly shows that the core stresses due to the cell-wall impact are higher than the ones impacted in the cell-center; however, interestingly, the face stresses due to the cell-center impact are not always higher than the ones impacted on the cell-wall, especially for the 7.5 mm cell size sandwich structure. This is probably because the core elements in the cell-wall impact plastically deform, exceeding the yield strength (318 MPa). This deformation allows the face sheet to bend further, resulting in bigger stresses.
In addition, the core in the smaller cell size fails first due to the higher FT/CT ratio, even though there are more cells in the 3 mm-cell sandwich structure. The following figure shows the core elements that exceed the yield strength when the 1 gram mass impactor hits the sandwich structure. Note that the transverse-direction core elements adjacent to the impact point fail, but the ribbon-direction core elements do not fail due to their doubled thickness.

Figure 4.7 Core elements exceeding yield strength for 1 gram impactor

The cell-center impacts are modeled for different impactor masses from 1 gram to 10 grams. Table 4.2 shows the correlation between maximum core stresses and the deflections at contact points. When the maximum core stress exceeds the yield strength, there are permanent deformations, such as all 3 mm cell size cases; however, when the maximum core stress is less than the yield strength or close to it, the material deforms elastically and returns to its original shape, thus there is no final deformation.
Table 4.2 Stress and deflection with respect to impact mass

<table>
<thead>
<tr>
<th>Core Max. stress (MPa)</th>
<th>Impactor mass (g)</th>
<th>Cell size</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>3 mm</td>
</tr>
<tr>
<td>1</td>
<td>343.0</td>
<td>233.4</td>
</tr>
<tr>
<td>2</td>
<td>407.4</td>
<td>336.1</td>
</tr>
<tr>
<td>5</td>
<td>478.1</td>
<td>442.0</td>
</tr>
<tr>
<td>10</td>
<td>498.6</td>
<td>476.7</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3 mm</td>
</tr>
<tr>
<td>1</td>
<td>0.003</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0.012</td>
<td>=0</td>
</tr>
<tr>
<td>5</td>
<td>0.030</td>
<td>0.003</td>
</tr>
<tr>
<td>10</td>
<td>0.050</td>
<td>0.020</td>
</tr>
</tbody>
</table>

From the above table, the smaller cell (3 mm) sandwich structures experience bigger stresses and have more deflections than the bigger cell (7.5 mm) sandwich structures under the same impact condition. In other words, if a blunt impactor, such as a bird, hit an airplane sandwich structure, more damage would be seen on the smaller cell sandwich structures than the larger cell ones.

4.2 Core Parameter Study

This investigation is to study how different core configurations effect the results of the impact. The impact circumstance is considered as the case of a hailstone hitting an airplane wing or a rock on runway hitting an airplane due to engine jet flow on the ground. Two categories of core configurations are examined. First, assuming the weight (36.7 g) and the face sheet thickness (0.56 mm) of the sandwich structure are fixed, the core height is varied. Secondly, with the same conditions, the cell size is varied.
4.2.1 Core Height Comparisons

Typical core height of aluminum sandwich structures used in the aviation industry varies depending on its application. In the following study, six different core heights are used to investigate the impact behavior of sandwich structure from a core height of 4 mm to 24 mm in length. For all core configurations, the weights of the core remain constant.

A. Geometry

Figure 4.8 shows two simulation models, in which (a) is the shortest and (b) is the tallest. The size of each cell is the same, having 6 mm in length. The average rectangular element size of the core is 2 by 2 mm and the face sheet is 1.7 by 2 mm for all simulation models. The thickness of the face sheet is 0.56 mm and the weight of all models remains unchanged. Most of the sandwich dimensions are basically the same used in the experimental study of chapter 3.

Figure 4.8 Core height: 4mm (a) and 24mm (b)
The core dimensions for all simulated models are shown in Table 4.3. The smallest core height (4 mm) sandwich structure has 1.2 FT/CT ratio and the largest one (24 mm) has 7.4 FT/CT ratio.

Table 4.3 Core dimensions for different core heights

<table>
<thead>
<tr>
<th>Core height (mm)</th>
<th>4</th>
<th>6</th>
<th>10</th>
<th>12</th>
<th>18</th>
<th>24</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ribbon thickness (mm)</td>
<td>0.45</td>
<td>0.3</td>
<td>0.18</td>
<td>0.15</td>
<td>0.1</td>
<td>0.075</td>
</tr>
<tr>
<td>Transverse thickness (mm)</td>
<td>0.225</td>
<td>0.15</td>
<td>0.09</td>
<td>0.075</td>
<td>0.05</td>
<td>0.0375</td>
</tr>
</tbody>
</table>

B. Boundary and Initial Conditions

The constraints in the sandwich structure are the same shown in Figure 3.5. The edge node lines on the top face sheet are fixed in the x and y directions. In addition, the nodes on the unsupported circular region of the bottom face sheet are constrained in the z direction and on its three axes of rotation. As explained earlier, the impactor is regarded as a rock or hailstone, which could cause severe damage when it hits an airplane structure. In the simulations, only the portion of this object is considered to hit the sandwich structure. This portion can be represented as a half sphere (16 mm diameter), like the one used in the experiment study as shown in Figure 4.9. It is assumed that the mass of the impactor is 200 grams and the velocity is 20 m/s. This impactor will hit normal to the surface of the sandwich structure. Only to compare the impact results of this relatively light-mass with high-velocity impactor, the impact conditions used in the experimental study of section 3.1, i.e. heavy mass (8.243 kg) with low velocity (3 m/s), are also studied simultaneously.
C. Results

As shown in Figure 4.10, it seems that the core height does not have an effect on the overall impact response of the structure, whether the impactor is heavy or light. The deflection values are measured by the impactor’s traveled distance from the bottom face sheet to the maximum deflection state. Although the final deflection results are slightly different for each core height configuration, the longest distance between maximum and minimum points is less than 0.5 mm for both simulation cases. The legends (a) and (b) in all of the following figures represent each simulation case. In case (a), the impactor has a light mass (0.2 kg) with high impact velocity (20 m/s). Case (b) corresponds to the impactor having heavy mass (8.243 kg) with low impact velocity (3 m/s). The impactor made a small hole on the bottom face sheets for all case (b) models; however, it did not penetrate the bottom for case (a) models.

![Figure 4.10 Deflection results with different core heights](image)

(a) light mass (0.2 kg) high velocity (20 m/s)
(b) heavy mass (8.243 kg) low velocity (3 m/s)
Inasmuch as cell thicknesses decrease as core height becomes thicker to maintain the mass of the sandwich structure, the dimensionless term is depicted on the following figure, showing decreasing concave up. In fact, it exactly follows cell thickness variation since deflection almost does not change. Here, the cell thickness is the ribbon-direction thickness. If the deflection results are also decreased as core heights become larger, it would be close to a straight line.

![Dimensionless Term with Different Core Heights](image)

**Figure 4.11 Dimensionless term with different core heights**

4.2.2 Core Cell Size Comparisons

Like the core heights, core sizes used in industry also vary depending on where and what they are used for. In the following study, four different cell sizes are used to study the impact behavior of sandwich structure varying from cell size of 3 mm to 7.5 mm, in increments of 1.5 mm. For all core configurations, the weights of the core also remain constant.
A. Geometry

Two core configurations, the smallest (3 mm) and lightest (7.5 mm) cell sizes, are described in Figure 4.12. The core heights of all sandwich structures are maintained with a fixed length of 12 mm. The average rectangular element size of the core is 1.5 by 2.4 mm and the face sheet is 1.3 by 1.5 mm. These element sizes are smaller than the ones used in core height study. The thickness for the face sheet is also 0.56 mm. Sandwich dimensions other than core configurations are basically the same as used in the experimental study.

(a)  (b)

Figure 4.12 Cell size: 3 mm (a) and 7.5 mm (b)

The core dimensions for all simulated models are shown in Table 4.4. The FT/CT ratio for the smallest cell size is 7.4 and the biggest is 2.9.

<table>
<thead>
<tr>
<th>Cell size (mm)</th>
<th>3</th>
<th>4.5</th>
<th>6</th>
<th>7.5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ribbon thickness (mm)</td>
<td>0.076</td>
<td>0.116</td>
<td>0.150</td>
<td>0.190</td>
</tr>
<tr>
<td>Transverse thickness (mm)</td>
<td>0.038</td>
<td>0.058</td>
<td>0.075</td>
<td>0.095</td>
</tr>
</tbody>
</table>
B. Boundary and Initial Conditions

The boundary and initial conditions used in this study are the same as used in the core height study.

C. Results

In the below figure, the deflection is measured by the same way described in the core height study. For both cases, as cell size becomes smaller, it is less deformed with core mass unchanged. The impactor made a hole on the bottom face sheets for all simulation models.

![Graph depicting deflection results with different cell sizes](image)

**Figure 4.13 Deflection results with different cell sizes**

The cell thicknesses almost linearly increase as shown in Table 4.4, and the largest cell thickness is more than twice the smallest one. However, the deflection, from Figure 4.13, is increased about a quarter or less from the lowest deflection. The
dimensionless term, therefore, shows an increasing concave up form, as seen in Figure 4.14.

![Figure 4.14 Dimensionless term with different cell sizes](image)

Unlike the impact results by the blunt impactor discussed in section 4.1.2, this study shows that there is more damage as cell size becomes bigger. Since both simulation results are exactly opposite, there must be a correlation between impactor size and cell size. In other words, this correlation can also be a crucial part of damage modes.

4.3 Other Types of Core Study

Truss and foam cores are also investigated to study core behaviors under an impact load in addition to the honeycomb core. In the previous section 3.2, the plane square woven core made in a truss-like fashion was studied in order to show the structural response of beam-like triad units used in truss core sandwich structures. In
addition, a honeycomb core was studied due to its popularity among many other core configurations.

4.3.1 Truss Core Sandwich Structure

Sypeck [19] investigated a truss (tetrahedral) core that would have advantages over a honeycomb core, such as forming feasibility of anticlastic curvature and improved corrosion resistance. A tetrahedral core among others is especially of interest due to its structural efficiency. A basic tetrahedral truss unit is shown in Figure 4.15. The truss core is made of these triad units with leg members of length $L$ and rectangular cross-section dimensions of $t$ and $w$. The angle between members is $\text{acos} \left( \frac{1}{2} \right)$, and the angle between each member and an extending line from triad base is $\text{acos} \left( \frac{\sqrt{2}}{3} \right)$. The height $H$ is about $L \left( \frac{\sqrt{2}}{3} \right)$.

A. Geometry

In his investigation, Sypeck fabricated the truss core sandwich structure, metallurgically bonding three different types of aluminums. The 1.0 mm thickness of face sheet is made from Al-6951 with a single side Al-4004 braze alloy cladding. This clad side faces the core, which is made from Al-6061. Each core unit has a square cross-section with thickness of 0.81 mm. The truss core height measured in the sandwich structure is about 9.7 mm. The fabricated truss core sandwich structures are shown in Figure 4.16.
Although the computer simulation models are designed in the same way, different dimensions and materials are used to simplify its modeling and reduce necessary input parameters. In the simulation models, only aluminum 2024 is used for both the face sheet and core materials. The face sheet thickness is 0.56 mm and the core has a rectangular cross-section with 0.96 mm thickness. The truss core height is 12 mm and the angles are followed with the descriptions given in section 4.3.1.

B. Boundary and Initial Conditions

Like the boundary conditions used in the core parameters study, both face sheets are constrained to prevent the specimen’s movement during impact. The mass and velocity of the impactor are 8.243 kg and 3 m/s, respectively.
C. Results

The simulation results show the buckling/bending behavior of the truss core due to impact that was rarely observed in a honeycomb core sandwich structure, as shown in Figure 4.17. This figure shows how the truss core would behave under impact loading.

Figure 4.17 Truss sandwich impact results: initial (a), maximum (b), and final (c)
4.3.2 Foam Core Sandwich Structure

Polyvinyl chloride (PVC) foams are one of the most commonly used core materials for sandwich structures. In this study, the Divinycell HP 100 foam from DIAB is chosen as the core material because mechanical properties of this PVC are superior to the other types of foams. The density of this foam is 100 kg/m$^3$ and Young’s modulus is 140 MPa.

A. Geometry and Impact Conditions

Since the foam core simply fills the space between both face sheets, rectangular solid elements having 3 by 3 by 3 mm dimensions are employed to represent the solid foam. The same boundary and initial conditions, as used in the truss core study, are adopted.

B. Results

Due to the highly crushable characteristic of the foam core, most of the kinetic energy of the impactor is absorbed by the foam. In addition, the foam’s resiliency after a compressive load and the elasticity of the face sheet make the sandwich structure go back to its original state, as shown in Figure 4.18.
Seemingly, the foam core sandwich structure is better than other types of sandwich structures under this impact condition; however, it is not exclusively used in commercial or military aviation industry mainly because it is not suitable for high temperature performance.
CHAPTER 5

BIRD STRIKES SIMULATION

From the previous studies, it was found that the core height in honeycomb sandwich structures did not have an effect on the impact results as long as the mass of the sandwich structure is maintained. The parameter that has an effect on the overall impact results was the cell size, fixing the thickness of the face sheets. Two different sandwich configurations varying their cell sizes are therefore adopted as part of an aircraft wing in bird strike simulations. The bird configuration and striking conditions are thoroughly explained and discussed.

5.1 Analysis of Bird Strikes on Sandwich Structure

The Federal Aviation Administration in the United States and Joint Aviation Authorities in Europe have established airworthiness standards regulations, which address the ability of aircraft to safely withstand bird strike. The U.S. Federal Aviation Regulations manual sets out a number of specific requirements in parts 23, 25, 29 and 33, in order to certify new aircraft pertaining to bird strikes [20]. Under transport category aircraft, for instance, the entire airplane must be able to successfully complete a flight after striking a 4 lb bird at cruse velocity. In addition, in the same category, the empennage structure must demonstrate bird strike tolerance against an 8 lb bird impact at the same velocity.
There have been various experimental methods of testing a bird strike on aircraft structures. The Arnold Engineering Development Center of the U.S. Air Force uses a chicken gun, which uses high-pressure air launch equipment with a 60 ft long tube. This testing equipment can fire a 4 lb thawed chicken carcass at speeds up to 900 mph to simulate a direct bird strike [21]. Although a real bird must be used for final aircraft certification tests, it cannot be determined that the real bird test is ideal. Differences in density and size of bird species with the same mass would cause different impact results. Even differences between individuals of the same species may produce no identical results. The American Society for Testing and Materials (ASTM) covers the standard testing method of certifying safe aircraft structures to bird strikes in ASTM F330-89 [22]. ASTM also allows the use of imitation birds in its certification.

Barber [23] investigated the use of substitute materials (gelatin and air) to replace real birds during bird strike testing. From the experimental and analytical investigations conducted in research, he came up with several important conclusions. Most importantly, birds can be explained as a fluid during impact. In addition, the fluid behavior of a bird during impact can be replaced with a circular cylindrical body composed of gelatin with ten percent air porosity. Finally, bird orientation at impact can have important effects on the results. Any angle of attack of a projectile effectively lowers the length-to-diameter ratio of the bird, and it can affect the impact pressure decay process.

5.1.1 Bird Model Design Process

A substitute artificial bird for a real bird is thoroughly discussed and developed by Budgey [24]. The International Birdstrike Research Group (IBRG) collected data of
commonly struck bird species and identified relationships across the species between mass, density and diameter. The relationship between bird mass and density without feathers is shown in Figure 5.1.

![Figure 5.1 Relationship between bird mass and density](image)

From figure 5.1, the density of a bird slowly decreases as the mass of the bird increases. The general equation for density can be obtained from the regression line in Figure 5.1,

\[
\text{Density of bird} = -0.063 \times \log_{10} \text{bird mass} + 1.148
\]

Using Eq. (3), the density of a 4 lb (1.81 kg) projectile would be 0.94 g/cm\(^3\) and an 8 lb (3.63 kg) projectile density becomes 0.92 g/cm\(^3\). Densities of birds do not change significantly, even though their mass can more than double.
The relationship between the mass of a bird and the diameter of its torso with feathers removed is shown in Figure 5.2.

![Figure 5.2 Relationship between bird mass and body diameter](image)

Unlike almost uniform density variation in Figure 5.1, the diameter of a bird’s torso increases more rapidly as the mass of a bird increases. From the regression line in Figure 5.2, the equation of diameter can be obtained as,

\[
\log_{10} \text{diameter of bird} = 0.335 \times \log_{10} \text{bird mass} + 0.90
\]

Using Eq. (4), the diameter of a 4 lb projectile becomes 98.1 mm, and an 8 lb projectile diameter is 123.7 mm.

Various shapes of artificial bird models are suggested by Budgey [24]: straight-ended cylinder, ellipsoid, and hemispherical-ended cylinder. These geometries are
chosen to reflect shapes of real birds. However, it is not hard to recognize that the straight-ended cylinder can rarely represent the shape of a bird. An ellipsoid seems to best represent the torso of a bird; however, there is no standard elliptical shape that can represent all birds tested. Every bird species has a different eccentricity of the ellipse that varies the shape of their torso. Therefore, using a hemispherical-ended cylinder for a bird substitute seems appropriate for computational bird strike simulation. The length of a hemispherical cylinder is assumed to be twice the diameter of the cylinder. Geometry of the bird model is shown in Figure 5.3.

![Figure 5.3 Geometry of bird model](image)

5.1.2 Finite Element Formulations for a Bird

Several finite element formulations have been applied to calculate bird impact on aircraft parts; such as wing, windshield, and engine. Hormann et al. [25] used Lagrangian and Arbitrary Lagrangian Eulerian (ALE) formulations in order to compare bird strike results with those from the national aerospace laboratory in the Netherlands, and optimized the leading edge of the wing. McCallum [26] presented ALE and Smoothed Particle Hydrodynamics (SPH) formulations to compare the strike results with a traditional bird shape model. Anghileri et al. [27] and McCarthy et al. [28] investigated
SPH bird strikes onto the composite intake of a turbofan engine, and the wing leading edge structure with fiber metal laminate skin, respectively. Brief descriptions of each formulation are described in some technical papers. Zukas [29] describes typical examples of Lagrangian and Eulerian finite element methods, as shown in Figure 5.4. Buyuk et al. [30] also explain Lagrangian, Eulerian, ALE and SPH formulations by comparing ballistic impact simulation results.

A. Lagrangian method

Lagrangian formulations follow the motion of fixed elements of mass, i.e. the nodes and elements move with the material. This formulation is used widely because of its ability to track material interfaces efficiently. It is generally used to represent solid materials; however, when elements are distorted significantly due to a high velocity impact, such as bird strikes, numerical inaccuracies grow resulting in negative volume masses. A typical example of the Lagrangian method is shown in Figure 5.4 (a).

B. Eulerian method

In the Eulerian approach, the computational mesh is fixed in space while physical material passes through it. This formulation is generally used to represent fluids and gases. However, since many elements are needed to enclose the whole space in which the material will be located during the simulation, extra computational time is required in order to maintain material interfaces and to limit numerical diffusion. Therefore, this method is not a good choice for bird strike simulations. A description of this method is shown in Figure 5.4 (b).
C. Arbitrary Lagrangian Eulerian (ALE) method

The purpose of ALE formulations is to capture the advantages of both Lagrangian and Eulerian finite elements while minimizing the disadvantages. In the ALE solver, it is well suited for a variety of both fluid and structural interaction problems. However, Audic et al. [31] mentioned that deformations are sometimes so large that ALE mesh became too distorted. Moreover, computational time increases because the formulation calculates diffusion terms, and a large computational domain is required for anticipating fluid dispersion during impact.
D. Smoothed Particle Hydrodynamics (SPH) method

SPH is a meshless Lagrangian formulation developed initially to simulate astrophysical problems [28]. This method replaces solid finite element meshes to a set of discrete interacting particles. Thus, mesh tangling in severe distortions can be handled. Due to its nature as Lagrangian formulation, it allows for tracking material deformation and time history behavior. Moreover, unlike Eulerian formulation, one only needs to model regions of material, rather than modeling all regions in which void material (air) may exist. This SPH method is used for the bird strikes simulations.

5.1.3. Methodology

Two distinct simulations are performed: i. An artificial bird is impacted on an aircraft wing, changing angles and adhering ribs, in order to see a general bird strike response of a wing structure. ii. A leading edge of a wing, assuming its flat surface, is modeled for a bird strike simulation to compare the impact results due to different core cell sizes of honeycomb sandwich structure.

i. The SPH bird is modeled with a total mass of 4 lbs (1.81 kg) and an initial velocity of 100 m/s in the global x direction. The target leading edge is modeled with properties of aluminum 2024 in the shape of a NACA 0012 airfoil. The leading edge structure is fixed in translation and rotation at both cutting edges. For the first simulation, the SPH bird model is impacted on the leading edge directly without an angle of attack. Then, under the assumption of taking-off and landing of aircraft, the leading edge was
struck by the bird model with angle of attack of 20 degrees. In order to observe differences in deformation results, another simulation was carried out with two ribs located symmetrically inside of the leading edge.

ii. The same bird model is used with an initial velocity of 28 m/s, which is the common take off velocity of a Cessna 150 aircraft. The target honeycomb sandwich structures, having 3 mm and 7.5 mm cell sizes, are modeled in the same way discussed in the elastic behavior study of section 4.1.2, but these ones have bigger dimensions. These structures are vertically impacted by the SPH bird and the results are compared.

A. Model Constructions

The bird model was defined as a hemispherical-ended cylinder using 8237 SPH particles. For the material behavior of bird, a null material model (type 9) in combination with a Gruneisen equation of state (EOS) is chosen. Barber [23] investigated that birds behave as a fluid during impact. Since the null material model in LS-DYNA has no yield strength and behaves in a fluid-like manner, it can be used with an EOS [13]. An EOS is required in order to accurately simulate material behavior since it is useful in describing the properties of gases and fluids.
(a) Geometry and material properties of SPH bird model (material type 9)

Table 5.1 Bird model properties

<table>
<thead>
<tr>
<th>Properties</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (kg/mm$^3$)</td>
<td>0.94 x 10^{-6}</td>
</tr>
<tr>
<td>Diameter (mm)</td>
<td>98.1</td>
</tr>
<tr>
<td>Length (mm)</td>
<td>196.2</td>
</tr>
<tr>
<td>Dynamic viscosity (kN·ms/mm$^2$)</td>
<td>0.4 x 10^{-9}</td>
</tr>
</tbody>
</table>

(b) Gruneisen equation of state (EOS)

The primary reason to use the EOS is to predict the state of gases and fluids. There is no general EOS that accurately predicts the properties of all substances under all conditions. For the bird strike simulation, McCallum [26] used the Gruneisen EOS. The Gruneisen EOS determines the hydrostatic behavior of the material by calculating pressure as a function of density. The general Gruneisen equation defines pressure for compressed materials as,

$$p = \frac{\rho_0 C^2 \mu \left[ 1 + \left( 1 - \frac{\gamma_0}{2} \right) \mu - \left( \frac{\gamma_0}{2} \right) \mu^2 \right]}{1 - (S_1 - 1) \mu - S_2 \left( \frac{\mu^2}{\mu + 1} \right) - S_3 \left( \frac{\mu^3}{(\mu + 1)^3} \right)} + (\gamma_0 + a\mu)E$$  \hspace{1cm} (5)

where

- $C$ = the intercept of the $v_e$-$v_p$ curve (wave propagation)
- $S_1, S_2, S_3$ = coefficients of the slope of the $v_s$-$v_p$ curve
- $\gamma_0$ = the Gruneisen gamma
\[ a = \text{the first order volume correction to } \gamma_0 \]
\[ \mu = \rho / \rho_0 - 1 . \]

McCallum [26] suggested some parameters for Gruneisen EOS used in the SPH bird simulations;

Intercept of curve (C) = 1482.9 mm/ms
- Coefficient of the slope (only S1) = 2.0367

The rest of the parameters in Eq. (5) are assumed to be zeros for the simulation.

(c) Geometry and material properties of target structure

i. Leading edge of wing

A NACA 0012 airfoil is chosen as a primary structure. Geometry and material properties are shown in Table 5.2. The size of the elements in the curved leading edge is 5 mm by 20 mm and the rest of the wing is 30 mm by 20 mm.

<table>
<thead>
<tr>
<th>Table 5.2 Wing properties</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td>Geometry</td>
</tr>
<tr>
<td>Span wise length (mm)</td>
</tr>
<tr>
<td>Density (kg/mm^3)</td>
</tr>
<tr>
<td>Chord wise length (mm)</td>
</tr>
<tr>
<td>Elastic modulus (Mpa)</td>
</tr>
<tr>
<td>Thickness (mm)</td>
</tr>
<tr>
<td>Yield strength (Mpa)</td>
</tr>
<tr>
<td>(mm)</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
</tr>
</tbody>
</table>
ii. Honeycomb sandwich structures

The geometry of the simulated models is shown in Figure 4.4. The only differences are the ribbon and transverse lengths due to the size of the bird model. Both dimensions are around 415 mm. The material properties of aluminum 2024 implemented in previous studies was used. The size of elements is approximately 3 mm by 3 mm and the mass of both structural models is 590 grams.

5.1.4 Results

A. Bird Strike on the Leading Edge of Wing

Due to the SPH finite element formulation of a bird model, dispersions of fluid material are observed during and after the impact as shown in Figure 5.5. The bird strikes simulations were performed on a single PC workstation of 2.0 GHz CPU with 512 MB RAM. It took less than 10 minutes for the simulations without ribs, and approximately 2 hours with ribs to complete a 2.5 ms event.

Figure 5.5 Before (a) and after (b) bird strike on leading edge
The leading edge of the wing with two symmetric ribs was less deformed since the ribs also absorbed impact energy as shown in Figure 5.6 (a) and (b).

![Figure 5.6 Deformation of leading edge without (a) and with (b) ribs at 0 degree angle of attack](image)

The same phenomenon occurred at the 20 degrees of angle of attack simulations. There was also less deformation on the bottom skin of the leading edge with ribs, as shown in Figure 5.7 (a) and (b).

![Figure 5.7 Deformation of leading edge without (a) and with (b) ribs at 20 degrees angle of attack](image)
B. Bird Strike on the Honeycomb Structures

After the SPH bird impactor strikes on the sandwich structures, a narrow hole was observed in the smaller (3 mm) cell size structure, but not in the larger (7.5 mm) one, as shown in Figure 5.8. This high-eccentricity-orbit shape hole is formed due to the thickness difference between ribbon and transverse directions of core. As discussed in section 4.1.2, the SPH bird impactor, like a blunt body, produces more damage in the smaller cell sandwich structures assuming the masses of face sheets and core do not change.

![Figure 5.8 Bird strike results on 3 mm (a) and 7.5 mm (b) cell size sandwich structures and their side views](image)

During the impact, the SPH bird particles came out through the hole for the 3 mm cell structure, but some of the particles also came out in the 7.5 mm cell structure,
although there was no penetration. The particles that came out through the bigger cell structure are possibly due to small cracks that developed during impact.
Since the birth of aviation, aerospace engineers have developed more efficient structures that can enhance overall aircraft performance. One of these structures developed was a sandwich structure that has high stiffness to weight and strength to weight ratios. Although much research regarding this structure has been done, nonlinear problems pertaining to impact still have much to explore.

Through this research, the correlation between impact parameters and damage modes are investigated, especially with different core configurations. The following results are achieved based on this research:

- If the applied impact energy is the same, the impactor with heavy mass and low velocity produces more damage than the light mass with high velocity impactor.
- A core always fails first if an impactor is a blunt body, whether it hits on the center of core cell or on the edge of the cell wall of a honeycomb core.

If the overall weight of a honeycomb sandwich structure remains unchanged and the only variable term is the core configuration, such as different core heights or cell sizes,

- The impact results are not affected by different core heights of the sandwich structure. As long as the weight of the core remains the same by changing cell thicknesses, the final impact results are always the same.
• The impact results are correlated with the size of the impactor and cell size of the sandwich structure. For a blunt body impactor, more damage is seen in the smaller cell size sandwich structure; however, more damage occurs in the bigger cell size sandwich structure when impactor size is a little bit bigger than core cell size or in close proximity.

Further research is recommended to investigate the impact on curved sandwich structures and composite sandwich structure models to find damage modes; such as delaminations, matrix cracks, and fiber breaks, under impact loading. The correlation between impactor size and cell size of honeycomb sandwich structures can also be an interesting research topic. In addition, multi-loadings, such as impact on pre-stressed structures or compression after impact, can be investigated.
References


Bibliography


2. Tabiei, A., LS-DYNA Course and Training, Livermore Software Technology Corporation, January 2004