FINITE ELEMENT ANALYSIS OF A GOLF BALL AND DRIVER HEAD
IMPACT: UNDERSTANDING THE FEASIBILITY
OF AN ACOUSTICAL OPTIMIZATION

A Thesis

Presented to

the Faculty of California Polytechnic State University,
San Luis Obispo

In Partial Fulfillment

Of the Requirements for the Degree

Master of Science in Mechanical Engineering

by

Nickolai Volkoff-Shoemaker

January 2010

© 2010

Nickolai Volkoff-Shoemaker

ALL RIGHTS RESERVED

COMMITTEE MEMBERSHIP

TITLE: FINITE ELEMENT ANALYSIS OF A GOLF
BALL AND DRIVER HEAD IMPACT:
UNDERSTANDING THE FEASIBILITY OF
AN ACOUSTICAL OPTIMIZATION

AUTHOR: Nickolai Volkoff-Shoemaker

DATE SUBMITTED: January 2010

COMMITTEE CHAIR: Dr. Tom Mase
Mechanical Engineering Professor

COMMITTEE MEMBER: Dr. Peter Schuster,
Mechanical Engineering Professor

COMMITTEE MEMBER: Dr. Jim Meagher,
Mechanical Engineering Professor

Abstract

FINITE ELEMENT ANALYSIS OF A GOLF BALL AND DRIVER HEAD IMPACT: UNDERSTANDING THE FEASIBILITY OF AN ACOUSTICAL OPTIMIZATION

By

Nickolai Volkoff-Shoemaker

An acoustic profile from the impact between a golf ball and driver head was produced using FEA. Following this, the results were analyzed to determine the feasibility of an acoustical optimization of such an impact. A validation of the FEA program LS-DYNA® was undertaken to ensure a proper solution was attained in the software analysis. An experimental and theoretical validation was performed that involved firing a golf ball at a titanium plate and comparing data from the impact to FEA simulations. Once the FEA program was validated, a golf club driver head model was used to generate an acoustical output using LS-DYNA®. By comparing this acoustic output to real driver head sound data, a feasibility of profiling good and bad sound was established. Using the optimization add-on program LS-OPT® a simple shape optimization was performed to maximize the speed of the ball after the impact. This new geometry of the club head was acoustically analyzed and the results were compared to the non-optimized case and shown to be distinct. The simulation results, however, did not compare well with the real

driver head data and more analysis would be needed to improve upon the results. Overall, the project was able to establish an analytical relationship to the acoustics generated and produce a solid foundation for the possibilities of an acoustical optimization.

Acknowledgements

The process of completing my thesis was a difficult one and would not have been possible without the help and guidance of several individuals.

I am grateful for all of the professors who took the time to help me overcome obstacles and hurdles that were encountered. I would like to thank Dr. Self and Dr. Mello for helping provide some insight into the field of experimental design. I would also like to thank Dr. Mello for giving me access to the Composites lab so that strain gauges could be installed. I want to thank Dr. Birdsong for helping with the LDS-Analyzer and providing the LEMO cables to connect to the strain gauges. Once these cables were obtained, I would not have been able to operate the LDS-Analyzer without the help of Chris Nosti. I would also like to thank Chuck Keezer for his help with making sure the experimentation process went smoothly. I'd also like to thank Dr. Chen for letting us use the high-speed camera of the ME department and showing us how it operated. My thesis committee advisors, Dr. Schuster and Dr. Meagher, also provided very useful guidance throughout the thesis process.

I would like to especially thank my thesis advisor Dr. Tom Mase. Three years ago I sent him an unassuming email about what it is like to be in the golf industry. Little did I know that we would work together on numerous projects and develop a great partnership over those three years. I would like

to thank Dr. Mase for giving me the skills, tools, and life lessons necessary for the real world. The information passed on to me was invaluable and will be used to establish a solid foundation for my engineering career.

I would like to thank my friends and family for always being there for me, no matter what. My family has supported me throughout my educational process and without them none of my successes and accomplishments would have been possible. For this I am truly thankful.

Table of Contents

List of Tables.....	xii
List of Figures.....	xiv
Nomenclature	xviii
1 Introduction.....	1
2 Finite Element Analysis	9
2.1 Background	9
2.2 Validation.....	14
2.3 LS-DYNA®.....	16
2.3.1 Keyword Format.....	16
2.3.2 Explicit/Implicit.....	17
2.3.3 Units.....	19
2.3.4 The Parts.....	19
2.3.5 Section Properties.....	20
2.3.6 Contact settings.....	20
2.3.7 Meshing.....	21
2.3.8 Hourglassing.....	21
3 Experiment.....	23
3.1 Set up.....	23
3.2 Air Canon	24
3.3 Impact Apparatus	31
3.4 Strain gauges	35
3.4.1 Strain Gauge Selection	37
3.4.2 Gauge Length	37
3.4.3 Gauge Pattern/Grid Type.....	38
3.4.4 Options	39

3.4.5	Resistance	39
3.4.6	Temperature Issues.....	40
3.4.7	Test Duration.....	40
3.4.8	Installation Issues.....	40
3.5	Data Acquisition	41
3.6	Strain Gauge Testing	46
3.6.1	Strain Gauges: Simple Beam Test	46
3.6.2	Strain Gauges: Central Force Applied to Plate.....	52
3.6.3	Strain Gauges: Impact Test.....	58
4	FEA with the USGA Plate/Ball Impact.....	64
4.1	The Plate	64
4.2	The Ball	66
4.3	The Ball and Plate	72
4.4	Mesh Convergence.....	74
4.5	The Bracket.....	84
4.6	The Supporting Structure	85
4.6.1	The Two Beam Structure.....	87
4.6.2	Contact Settings – Two Beams.....	88
4.6.3	Loading Conditions – Two Beams	89
4.6.4	Boundary Conditions – Two Beams	89
4.6.5	The Entire Structure.....	90
4.6.6	Loading/Boundary/Contact Conditions – Entire Structure.....	91
4.7	Section Properties	93
4.8	Model Check	94
5	Comparing FEA to Experimental Results.....	98
5.1	Clamped Plate with a Concentrated Force	98
5.2	Impact Tests.....	100

5.2.1	Results.....	100
5.2.2	Impact Hand Calculations	104
5.2.3	Frequency Results.....	110
6	The Driver Head/Ball Impact.....	113
6.1	LS-DYNA® Settings	116
6.1.1	Section Properties.....	116
6.1.2	Contact Settings	116
6.1.3	Rigid to Deformable.....	117
6.2	Acoustics Using Boundary Element Method	117
6.3	Acoustic Settings in LS-DYNA	120
6.4	Loading Conditions.....	121
6.5	Boundary Conditions.....	122
6.6	Meshing/Model Check	123
6.7	Mechanical Results of the Simulation.....	127
7	Golf Industry Driver Head Sound Data.....	129
8	Optimization.....	135
8.1	Defining an Optimization.....	136
8.1.1	Design Variable	136
8.1.2	Objective.....	137
8.1.3	Constraint	138
8.1.4	Problem Solution	138
8.2	LS-OPT®.....	139
8.2.1	Solvers.....	139
8.2.2	Optimization Methods.....	139
8.2.3	Algorithms	142
8.2.4	Strategy.....	143
8.2.5	Optimization Settings in LS-DYNA®	143

8.3	Driver Head COR Optimization	144
8.3.1	Driver Head LS-DYNA settings	145
8.3.2	Objectives/Constraints	145
8.3.3	Metamodel Type	145
8.3.4	Algorithm	146
8.3.5	Histories/Response	146
8.3.6	Optimization Results	146
9	Comparison of Acoustic Simulation Results	153
9.1	Acoustic Results	153
9.2	The Feasibility of an Acoustic Optimization.....	159
10	Discussion and Recommendations	161
10.1	Experiment.....	161
10.2	Computation.....	162
10.3	Acoustics.....	164
10.4	Optimization	165
10.5	Future Plans	168
11	Conclusion.....	169
	References	172
	SolidWorks® Drawings and BOM	A-1
	Hand Calculations	B-1
	LS-DYNA® Keyword Example File	C-1
	MATLAB™ Code Calculations	D-1

List of Tables

Table 1: Ideal Ball Flight Characteristics for the Driver	5
Table 2: English Units in LS-DYNA®	19
Table 3: Air Canon Speed Results	30
Table 4: Strain Gauge Selection	41
Table 5: Gauge Setup Types	42
Table 6: Data Acquisition Settings for Strain Gauges.....	45
Table 7: Data Summary of Central Force Tests.....	54
Table 8: Theoretical Calculations	57
Table 9: Data Acquisition Settings	58
Table 10: Different Test Speeds.....	59
Table 11: Plate's Geometric Properties	65
Table 12: Plate's Mechanical Properties	66
Table 13: Elastic Moduli of the Golf Ball Components.....	68
Table 14: Golf Ball Curve Fitting Properties	69
Table 15: Golf Ball Hyperelastic Properties.....	70
Table 16: Golf Ball Modified Hyperelastic Properties	70
Table 17: Golf Ball Mechanical Properties.....	71
Table 18: Number of Nodes for the Different Mesh Densities	75
Table 19: Mesh Convergence Strain Data for Clamped Plate with Force	79
Table 20: Mesh Convergence Strain Data for Ball/Plate Impact	81
Table 21: Mesh Convergence Displacement Data for Plate with Force.....	82
Table 22: Mesh Convergence Displacement Data for Ball/Plate Impact.....	83
Table 23: Supporting Parts Mechanical Properties.....	92
Table 24: Supporting Parts Geometric Properties.....	93
Table 25: Number of Nodes and Elements for the Supporting Structures.....	93
Table 26: FEA data for Concentrated Force Case.....	99
Table 27: Drop Test Results.....	106
Table 28: Summary of Strain Results	107

Table 29: Force/Strain Ratio for Each Case	108
Table 30: Frequencies of Cases for the 30 psi Chamber Pressure	110
Table 31: Properties of Components of Driver Head FEA Model.....	115
Table 32: Acoustic Settings	121
Table 33: Mesh Density of Components of Driver Head.....	123
Table 34: Types of Elements in Driver Head Mesh	123
Table 35: Shell Element Quality Checks.....	124
Table 36: Peak Frequency Values for the Three Driver Heads.....	134
Table 37: Optimization Results for the Objective and Constraint.....	149
Table 38: Optimization Results for the Design Variables	149

List of Figures

Figure 1: Golf Ball	2
Figure 2: Golf Shafts	3
Figure 3: Driver Head Components.....	4
Figure 4: Evolution of the Driver Head (Golf Digest)	6
Figure 5: Beam with 1-D/1 DOF Elements	10
Figure 6: Beam with 1-D/2 DOF elements	11
Figure 7: Different Types of Finite Elements	11
Figure 8: Plate and Ball Impact	15
Figure 9: Experimental Set Up.....	24
Figure 10: Air Canon.....	25
Figure 11: Air Canon Calibration Setup	29
Figure 12: Air Canon Calibration Data.....	29
Figure 13: Air Canon Fitted Data.....	30
Figure 14: Impact Apparatus (a) without foam (b) with foam	31
Figure 15: Acoustic Foam.....	32
Figure 16: Clamping Bracket.....	33
Figure 17: Brackets and Plate	34
Figure 18: Plate and Bracket Connection to Impact Apparatus	34
Figure 19: (a) Actual Strain Gauge (b) Diagram of a Strain Gauge	35
Figure 20: Wheatstone Bridge	36
Figure 21: Plate with Strain Gauge Setup.....	38
Figure 22: Strain Gauge Input Port (LDS-Dactron)	42
Figure 23: Half Bridge for Strain Gauges	43
Figure 24: (a) Half Bridge Configuration (b) Full Bridge Configuration	44
Figure 25: Installed Strain Gauges on Simple Beam	47
Figure 26: Simple Beam Strain Gauge Setup.....	47
Figure 27: Strain Data of Dynamic Loading	50
Figure 28: Experimental vs. Theoretical Values.....	51

Figure 29: Load Cell	53
Figure 30: Center Force Setup.....	53
Figure 31: Central Force Plate Data	54
Figure 32: Theoretical Plate Calculation Variables.....	56
Figure 33: Radial Strain Profile of Plate.....	57
Figure 34: Strain Data for First Test at 30 psi	61
Figure 35: Raw Strain Gauge Impact Data.....	61
Figure 36: All Strain Gauge Impact Test Data	62
Figure 37: All Strain Gauge Impact Test Data Separated	63
Figure 38: Titanium Plate.....	65
Figure 39: Golf Ball Section View.....	66
Figure 40: The Three Components of the Golf Ball	71
Figure 41: Force Profile at Impact.....	72
Figure 42: Different Ball and Plate Meshes.....	73
Figure 43: Golf Ball Mesh	74
Figure 44: Boundary Conditions for Plate	76
Figure 45: Load Curve for Concentrated Force Applied to Plate	76
Figure 46: Elements Examined for Mesh Convergence.....	77
Figure 47: Different Meshes Strain History for Clamped Plate with Force...	78
Figure 48: Mesh Convergence Strain Plot of Clamped Plate with Force.....	79
Figure 49: Different Meshes Strain History for the Ball/Plate Impact	80
Figure 50: Mesh Convergence Strain Plot of Ball/Plate Impact	81
Figure 51: Mesh Convergence Displacement Plot for Plate with Force.....	82
Figure 52: Mesh Convergence Displacement Plot of Ball/Plate Impact	83
Figure 53: Bracket Mesh.....	85
Figure 54: T-slot Sectional Properties.....	86
Figure 55: Mesh of Two-Beam Supporting Structure.....	88
Figure 56: Boundary Conditions for Two-Beam Support Structure	90
Figure 57: Entire Supporting Structure Mesh.....	91
Figure 58: Boundary Conditions for Entire Supporting Structure	92
Figure 59: Quadratic Elements	94

Figure 60: Aspect Ratio of All Parts for Validation Simulations	95
Figure 61: Maximum Angle of All Parts for Validation Simulations.....	97
Figure 62: Comparison of Concentrated Force Test	99
Figure 63: Comparison of 30 psi Results.....	101
Figure 64: Comparison of 40 psi Results.....	101
Figure 65: Comparison of 50 psi Results.....	102
Figure 66: Interpolated Data from 40 psi run.....	104
Figure 67: Drop Test Picture	105
Figure 68: Force Profile of Impact for 30 psi.....	107
Figure 69: Longer Time Duration Comparison.....	111
Figure 70: Driver Head FEA Model.....	114
Figure 71: Boundary Element Method.....	119
Figure 72: Driver Head Boundary Conditions	122
Figure 73: Aspect Ratio Values for the Driver Head Mesh	125
Figure 74: Warpage Values for the Driver Head Mesh	125
Figure 75: Jacobian Values for the Driver Head Mesh	126
Figure 76: Required Time Step Values for the Driver Head Mesh	127
Figure 77: Driver Head and Ball Impact.....	128
Figure 78: Wave File Data for the Different Driver Heads	130
Figure 79: FFT Data for the Different Driver Heads	131
Figure 80: Specific FFT Regions for the Different Driver Heads.....	132
Figure 81: Pareto Front.....	138
Figure 82: Metamodel Visual Representation	141
Figure 83: Velocity Maximization of Driver Head Optimization	147
Figure 84: Mass Constraint of Driver Head Optimization	148
Figure 85: Metamodel of Driver Head Optimization	150
Figure 86: Feasibility Distribution for the Optimization	151
Figure 87: Metamodel Accuracy for the Velocity Objective.....	152
Figure 88: Amplitude vs. Time for Driver Simulations	153
Figure 89: Short Time Duration Acoustics.....	155
Figure 90: FFT Data from LS-DYNA® for Driver Simulations	156

Figure 91: FFT Data from MATLAB™ for Driver Simulations.....	157
Figure 92: Pareto Fronts Using LS-OPT®	166
Figure 93: Topological Optimization Example.....	167
Figure 94: Actual Club Head Design Dimensions	168

Nomenclature

B	Shape Function Derivative
COR	Coefficient of Restitution
$C01$	Coefficient of Mooney-Rivlin Function
$C10$	Coefficient of Mooney-Rivlin Function
$D1$	Coefficient of Mooney-Rivlin Function
D	Constitutive Material Relationship
E	Elastic Modulus
F	Force
$G(t)$	Relaxation Function
ΔG	Change in Momentum
G_i	LS-DYNA [®] Rate Dependency Shear Modulus Parameter
GF	Gauge Factor
I	Moment of Inertia
I_1	First Invariant of Strain Energy Density Function
I_2	Second Invariant of Strain Energy Density Function
I_3	Third Invariant of Strain Energy Density Function
K	Stiffness
K_H	Hertz Force Equation Constant
L	Length
M	Moment
P	Pressure
R	Electrical Resistance
V	Volume
V_{EX}	Excitation Voltage
V_O	Output Voltage
W_{net}	Net Work

W_{thermo}	Thermodynamic Work Done
W	Strain Energy Density Function
a	Radius of Circular Plate
b	Cross Sectional Width
c	Distance of Stress State Location
c_{sound}	Speed of Sound
f	Objective Function
g	Optimization Inequality Constraint
g_1	Tanaka Rate Dependency Parameter
h	Optimization Equality Constraint
k	Wave Number in Helmholtz Equation
m	Mass
n	Polytropic Index Value
r	Radial Direction for Circular Plate
t	Cross Sectional Thickness
u	Displacement, or DOF
v	Speed
$\mathbf{x}$	Design Variable
z	Normal Direction for Circular Plate
β	LS-DYNA® Rate Dependency Decay Parameter
δ	Displacement
ε	Strain
μ	Shear Modulus
ν	Poisson's ratio
σ	Stress
τ_1	Tanaka Rate Dependency Decay Parameter
ω	Frequency

Chapter 1

Introduction

Golf is a simple game. The object of this game is to get the ball in the hole in as few strokes as possible. It sounds easy enough, but is surprisingly difficult. Golf's origins are somewhat cloudy, but most people trace it back to Scotland in which golf balls were once made of feathers and clubs from wood (Encyclopaedia Britannica Online). Since that time, however, the game of golf has undergone quite a transformation. The game has come from humble beginnings to become embedded in mainstream culture and has utilized the latest technological advancements in every phase of the game. The equipment that is used in the game today can be broken down into three main components: the ball, the shaft, and the club head.

The golf ball began as a pellet of feathers and has evolved to become a very precisely engineered piece of equipment that has enabled players to have more control than ever when playing the game. The dimples surrounding the outer layer of the ball are there for a reason: to give the ball a more aerodynamic and efficient flight. The material the ball is made of is an advanced polybutadiene rubber that allows the ball to perform at its best

whether you are swinging a driver over 100 mph or hitting a simple chip shot a few feet away from the hole. The amount of engineering that goes into the ball is remarkable, and the task of modeling it is very difficult. A golf ball is generally composed of three parts, which include the mantle, the core, and cover (Encyclopaedia Britannica Online). The golf ball core is a non-linear material and has more complicated responses than most materials. The mantle and the cover also have unique properties and play key roles in the performance of the ball. Figure 1 shows the golf ball used for testing and a view of its profile. Each of the components will be more thoroughly explained later.

Figure 1: Golf Ball

Another component of golf equipment is the club's shaft. The shaft has also evolved from its beginnings as a simple piece of wood, to hollow metal shafts in the early 1900's, and in the past decade into "graphite" shafts which are actually composite materials composed of carbon fiber (Encyclopaedia Britannica Online). These composite shafts are lighter and have comparable

mechanical properties to the metal shafts. The ability to swing the club faster with these shafts makes them a favorite choice when coupled with the driver head. Figure 2 below shows a composite shaft and steel shaft.

Figure 2: Golf Shafts

The driver head is a component of the golf club that is used to make the ball travel the farthest. It attaches to the end of the shaft and can have varying geometries and shapes. The overall structure has a few different sections which when combined form the driver head. Some of these components can be seen in Figure 3, while others will be discussed more in later sections.

When designing a driver club head each of these components can have their thicknesses adjusted to alter the ball's flight after an impact.

Figure 3: Driver Head Components

When the ball rebounds off the face there are certain properties that are looked at to ensure an idealized flight. These properties include the launch angle, the spin rates, and the launch speed. The laws of physics dictate that when an object is launched and undergoes projectile motion, the maximum distance it can travel is determined by the launch angle and the speed of the ball. When the ball is treated as a rigid body, the computations become more complicated. Because the ball has dimples and different spin rates, the trajectory equation becomes more complex and depends on not only the launch angle and speed but also the spin rates (Jorgensen). For an idealized flight it is important to have certain launch angles and spin rates, which depend on the specific club impacting the ball. The ideal flight properties for the driver can be seen in Table 1 and will be discussed more later on in the report (Jorgensen). You will notice that there is not an idealized speed

because ideally you want the maximum speed possible in order to maximize the distance the ball travels.

Table 1: Ideal Ball Flight Characteristics for the Driver

Characteristic	Value
Back Spin	2400-2600 rpm
Launch Angle	12-16°
Speed	-

Equipment advancements in recent years have prompted golf's governing bodies to make changes of their own. Two major bodies involved in the game today are the United States Golfers Association (USGA) and the Royal and Ancient Golf Club at St. Andrews (R&A). Both of these bodies dictate the rules of the game as well as the equipment guidelines (R&A and USGA). Technology has played such an important role in golf, that certain equipment restrictions have been placed by the USGA in the past few years. One of these major restrictions relates to the allowable size of the driver head. As the Figure 4 shows, over the past two decades the size of the driver head has continued to increase. The amount of volume the driver head occupies has more than tripled since 1985 and as you will notice in Figure 4, the driver head has undergone some radical transformations in style, material characteristics, and performance. The governing bodies wanted to make sure the integrity of the game was maintained, and as a result the driver club head now has a volume restriction set at 460 cubic centimeters.

Figure 4: Evolution of the Driver Head (Golf Digest)

Another restriction the USGA has implemented deals with the impact between the ball and the club face. The coefficient of restitution (COR) is a term that is defined as the ratio of the relative outgoing velocities to the incoming velocities of objects. In terms of a golf ball, it is the velocity of the ball after the impact divided by the velocity of the ball before impact. This equation can be seen below.

$$COR = \frac{v'_A - v'_B}{v_B - v_A} \quad 1.$$

The USGA has decided that in order to maintain the game's integrity the COR of the golf club face-to-ball collision must not exceed the value of 0.83.

These two rules along with others have forced engineers and club manufacturers to reevaluate their designs and figure out ways to maximize the performance with these constraints. In this case, the goal in the golf industry is to make the ball go farther, faster, and spin less to optimize distance. For these goals to be met, the equipment must be analyzed and continually redesigned to achieve the best results. The clubs need to be lighter, contain the best materials, have a good feel, sound good, and of course get the ball to have the optimal characteristics mentioned in earlier. One of the ways manufactures have achieved some of these results is by changing the geometry of the driver head. However, some companies have only focused on a few of the characteristics mentioned above. They have only looked at an aspect like hitting the ball farther, while disregarding the sound the club makes. Some of the drivers of recent years have been ridiculed, and as a result consumers have shown they aren't willing to spend top dollar for something that does not have an appealing sound. In a twelve month design phase at one golf company, the poor sound the driver was making set the design cycle back three months (NDA).

With all of these conditions in mind, our project involved analyzing the club that gets the ball moving the fastest: the driver. The goal of this project focused around the idea that the physical sound produced by a golf club striking a golf ball can be analyzed analytically. I worked with a colleague of

mine, Roger Sharpe, and we explored how to analytically interpret and understand the acoustic profile a driver head generates. An optimization routine was also performed to try and obtain the optimal golf ball flight characteristics mentioned earlier. With these analyses, an idea of optimizing the sound of the driver head was better understood. To perform these tasks, certain steps had to be taken. First, a simple Finite Element Analysis (FEA) was performed and then validated with experimental data. The models were fine tuned to show that more advanced simulations could be performed by using similar settings. After experimental results validated the FEA code, we used the FEA code to analyze a driver head model and perform an analysis of the acoustical output. This driver head structure was also analyzed mechanically. The structure was altered so that its shape could produce optimal flight characteristics. The overall goal was to try and understand the feasibility involved with the concept of optimizing the acoustics and structure of an object. It is a rarely explored concept which could provide a basis for how products of all types are analyzed and designed in the future.

Chapter 2

Finite Element Analysis

2.1 Background

Similar to the golf industry, the engineering world has undergone some revolutions of its own in the past few decades. With the digital revolution, engineers are able to tackle the most difficult problems in the discipline. With computers readily available, the concept of Finite Element Analysis (FEA) has become a much more practical alternative to solving engineering problems. Finite element analysis is a numerical technique for approximating complex mathematical representations like integral equations or partial differential equations (Hiermaier). These functions are modeled so that they can be solved at certain discrete locations and instants of time. These discrete locations are known as nodes, and when all nodes of a system are combined, they represent the entire structure, or the domain of an analysis. Adjoining nodes are connected to form an element, which is assigned the material properties of the structure being analyzed.

A simple example of FEA can be seen through the analysis of a cantilever beam, which can be seen in Figure 5. In this example, a beam is bound at one end and a force is applied at the free end. There are three elements and four nodes for this structure and the numerical data of interest can only be calculated at the discrete nodal points. By increasing the number of elements you in turn increase the number of nodes. With more nodes, one can get the solution at more points and get a more accurate representation of what is happening to the structure.

Figure 5: Beam with 1-D/1 DOF Elements

For a structural problem like the one in Figure 5, the displacements at each node are the primary variables, and represent the degrees of freedom (DOF) of the system. The beam in Figure 5 only has 1 DOF per node, because only the axial direction is of interest. However, as seen in Figure 6, when the system is stressed differently, more information is needed to understand what is going on with the structure, and as a result nodes with information in both the axial and transverse directions are required.

Figure 6: Beam with 1-D/2 DOF elements

The number of degrees of freedom per node is defined by the user and depends on what is being modeled. One can use a simple 1-D truss element with 1 DOF and 2 nodes per element or use something more advanced like a 3-D solid hexahedral element with 3 DOFs per node and 8 nodes per element. These examples can be seen in Figure 7 below.

Figure 7: Different Types of Finite Elements

Although not technically correct, one can simplify FEA to the idea that every element of the structure represents a tiny spring that shares the same properties as the material being modeled. When the structure is stressed, all of these tiny springs have a change in displacement which is dependent on their stiffness, or material properties. The goal of using FEA is to solve for all of the DOFs of the system and then use that knowledge to identify stresses, strains, or other characteristics of the system. In FEA, a proof using the minimum potential energy principle results in an equation that is used to solve for all the DOF of the system (MacDonald). Versions of the global equation for all the elements can be represented in matrix form in Equations 2 and 3.

$$\{F\} = [K]\{u\} \quad 2.$$

$$\{u\} = [K]^{-1}\{F\} \quad 3.$$

$$K = \int [B]^T [D] [B] dV \quad 4.$$

In these equations, F represents the global force term, K represents the global stiffness term, and u represents the global DOF of system. In calculating the DOFs of the system, many steps must be taken, which is why computers are so useful for FEA. First, stiffness properties and force data are assigned to each element and assembled for all elements in the system.

This is done by using Equation 4, which requires the materials constitutive relationship, D , and the shape functions derivatives, B . Shape functions are an approximation of the DOF distribution across an element (Hiermaier).

The idea of using a shape function is an important assumption in FEA that must be made for a solution to arise. The assumption is made by the user and depends on the problem. For most problems one will assume a linear change of the DOF from one node to the next. This assumption shows why it is important to try and maximize the number of nodes in a problem, so there are more known data points in the system, and less approximated ones.

After the shape functions are assumed and the material constitutive relationship is identified, the stiffness matrix is inverted so Equation 3 can be used to find the DOF matrix.

When you do have a global DOF matrix, you are representing all the elements of the system. It is important to note that if you are running a system with 10000 nodes with 1 DOF per node, a 10000 x 10000 matrix would have to be computed to solve for all the DOF of the system. This is why the use of computers is so critical, because hand calculations for a problem involving so many elements would be extremely inefficient. With the displacements known at all the nodes, the stresses and strains can be found using Equations 5 and 6.

$$\{\varepsilon\} = \frac{d}{dx}\{u\} \quad 5.$$

$$\{\sigma\} = [D]\{\varepsilon\} \quad 6.$$

The advantage of modeling an object and performing FEA is that you can test different designs in a computer environment and see visual computational results. This eliminates having to go through a traditional design cycle of producing a prototype, testing, and rebuilding a design.

2.2 Validation

FEA is a very powerful tool for engineers, but only when it's used correctly. Because there are so many inputs and changes that can be made in a simulation, it is very easy to have incorrect results.

There are two ways to validate the FEA simulation. One way to validate the FEA results is to use hand calculations and analytical models. A golf ball impact problem is a difficult problem to model, and would require advanced engineering techniques to solve properly. The other way to validate FEA is with experimental testing (MacDonald). A simpler FEA simulation would have to be examined and then an experimental test would be performed to see if the experimental data matched values found in the FEA.

To make sure the FEA can be validated, both simplified hand calculations and experimental testing was done. For our purposes an analysis was required that still involved an impact, but one that was easier to analyze both acoustically and structurally. It was decided that a simple normal impact between a plate and golf ball would be the best test to perform for validation purposes. The golf ball was modeled with unique properties while the simple plate represented a geometry that was easier to analyze, compared to an object like a driver head. The simulations were performed using the FEA program LS-DYNA®.

Figure 8: Plate and Ball Impact

2.3 LS-DYNA®

LS-DYNA® is a transient dynamic finite element solver capable of simulating very complex real world engineering problems. LS-DYNA® was developed by John Hallquist in 1976 and has evolved over the years into a robust program with an extensive material database (Livermore Software Technology Corporation). This material database can model just about anything, from lung tissue to composites and everything in-between. This extensive material database is especially useful for modeling nonlinear materials, or materials that have rate dependencies, like a golf ball.

2.3.1 Keyword Format

LS-DYNA® operates by using keywords that activate a function used by the program. This format is quite a leap from the card deck era of finite elements. A Graphical User Interface (GUI) is used to mesh the object from a part file that has already been generated in the format of an IGES file by using a Computer Aided Design (CAD) software program. The powerful options of LS-DYNA® are implemented by editing a text file containing all the geometry and control commands. Once you understand which keywords correspond to certain functions, you can write your own code and specify exactly what you want the program to do. This is an advantage compared to other FEA programs like ABAQUS®, which don't let the user control much of

the inner workings of the program or require the user to sift through multiple layers of the GUI to set the desired options. The LS-DYNA® GUI, which it calls the pre-processor that can be utilized for activating all the keywords that LS-DYNA® has to offer. This pre-processor makes the program more user-friendly and easier to engage. The actual solver that was used was an updated double precision solver, version LS971_D_R4_54444. A double precision solver is better to use, compared to a single precision solver, because it holds more decimal places for all the numbers involved in every calculation. As a result, it reduces the round-off error you would get when performing so many calculations. These calculations include those for the stiffness matrices as well as the cumulative analysis for all the time steps.

2.3.2 Explicit/Implicit

In FEA, there are two ways in which a time-dependent solution can be solved. One way is to use an implicit solver. The implicit approach uses the Newmark Forward Difference method (Livermore Software Technology Corporation). In this method a complicated equation is evaluated wherein the global stiffness matrices are inverted and the displacements are solved at each time step. The advantage of this method is that there are no constraints on the time step. The other way to solve a time-dependent problem is to use the explicit solver. The explicit solver uses the central difference method in which information for the next time step depends on the previous time steps.

The explicit solver does not depend on inverting the stiffness matrix. This is a huge advantage because without having to invert and solve for stiffness matrices, the explicit solver becomes much more advantageous in terms of a computational perspective. The downside with the explicit solver is that a minimum time step must be used or the solution can not be reached. This minimum time step is calculated by LS-DYNA® and is different depending on the type of element being used. The time step is generally dependent on the element geometry and the material properties of that element. Therefore, if you have a complicated mesh which has warped elements, the time step will have to be decreased to make sure the explicit method doesn't become unstable.

For problems in which only a short duration analysis is needed, the explicit solver is a much better choice. The amount of time that takes place in an impact scenario is very short and depends on the speed of the incoming object. When a golf ball is impacted by a driver, the driver head is moving at speeds of around 100 mph. From the change in momentum equation, you can show that the impact time will occur on the order of a millisecond at these high speeds. Therefore, because we are dealing with short time spans, and because of the heavy computational cost of using the implicit solver, the explicit solver will be used in the analyses performed by LS-DYNA®.

2.3.3 Units

All FEA programs are unitless and it is the user's responsibility to keep track of the unit system being used. LS-DYNA® is no different, and it is critical that a consistent set of units be used, or else the results could be completely invalid. For our purposes, the English units system will be used for all simulations. A table of the units LS-DYNA® requires can be seen in Table 2 (Livermore Software Technology Corporation).

Table 2: English Units in LS-DYNA®

Dimension	Unit
Length	inch
Time	second
Mass	lb _f -s ² /in
Density	lb _f -s ² /in ⁴
Pressure	psi
Force	lb _f
Energy	lb _f -in

2.3.4 The Parts

The first step when creating a simulation is to generate a part that represents the structure of the object of interest. This part must then be assigned a material property and a sectional property, or a type of element. The material properties will be discussed more when specific simulations are addressed.

2.3.5 Section Properties

In FEA there are different elements that can be used to represent the simulation. In our case, both shell sections and solid elements were utilized. The advantage of shell elements is that they require less computation time, but don't provide a solution to how the out of plane stresses are affecting the problem. The solid elements show the stresses of cross sections, but add more computation time. To get the same accuracy as shell elements, at least three solid element layers must be used to model the thickness (MacDonald). Both shell elements and solid elements will be used in certain situations depending on the type of problem.

2.3.6 Contact settings

The contact based problems that are solved with FEA are especially challenging, and in order to predict correct results the proper settings must be chosen. In LS-DYNA® there are numerous contact settings to choose from. One of the standard contact interactions that the program uses is the "SURFACE_TO_SURFACE" contact setting, which has the element's normal vectors check for surface penetration. The advantage of this keyword is that both of the parts involved in a contact situation check their respective normal vectors for penetration. The "AUTOMATIC_SURFACE_TO_SURFACE" setting goes a step further by automatically checking both normal directions

for penetration so the user doesn't have to specify the normal vectors themselves (Livermore Software Technology Corporation).

2.3.7 Meshing

When more complicated parts are analyzed with FEA, the structure must be characterized with elements so the geometry of the part is not compromised. This is done by creating a mesh of the part, which organizes all the elements so they are all connected and come together to represent the part. To validate the code a mesh convergence is required to make sure a proper solution has converged. There are a few different ways to perform a convergence of the FE model. These include increasing the number of elements, changing the integration scheme, or altering the element biasing (MacDonald). For these analyses, different meshes were made having varying element densities. In each of these meshes a specific location was analyzed and simulation values at that location were compared amongst the different meshes.

2.3.8 Hourglassing

Hourglassing is an FEA error associated with elements that undergo zero-energy deformations. The element in the simulation will deform in a way so that no change in the element's energy state takes place (Hiermaier). In real life these deformations would not take place, so they represent a crucial error

to the simulation. However, hourglassing only occurs when you use reduced integration. For our simulations we will be using full integration of elements and therefore hourglassing will not be an issue. When FEA is used, integrals are solved by using numerical integration. One of the main numerical integration schemes is Gaussian Quadrature. This technique simplifies the integral into a series of terms multiplied by weighted values (Hiermaier). This technique, however, assumes that a polynomial is being evaluated in the integral. The type of quadrature level used depends on the order of this polynomial. Gaussian Quadrature minimizes the number of function evaluations to achieve the correct integral. By using reduced integration, the quadrature level is one less than what should be used, and as a result a less accurate answer will be obtained.

Chapter 3

Experiment

As mentioned in Section 2.2 of the report, experimental testing is an important step that should be used in order to validate FEA results. The testing we did was done to try and create an environment and scenario that could be replicated in computer simulations. The overall testing involved gathering mechanical data as well as acoustical data. Roger Sharpe confirmed his acoustical model in LS-DYNA® while I verified stress data.

3.1 Set up

To validate our results we fired a Titleist Pro VI Practice Ball at a circular titanium plate which was securely clamped, mounted with strain gauges, and placed in a sound dampened environment to gather the data we needed. The strain gauges were validated by performing some basic tests to make sure they were outputting accurate values. The golf ball was fired at high speeds with an air canon that was custom built for this experiment. A robust data acquisition system with a digital interface was used for the experiment to

gather the data accurately and efficiently. The entire experiment took place in an enclosed area with the protection of a golf net as shown in Figure 9.

Figure 9: Experimental Set Up

3.2 Air Canon

The air canon was composed of three main parts, the air chamber, the solenoid valve, and the release tube. The air chamber was schedule 40 PVC pipe, capable of operating at pressures up to 130 psi, as stated in ASTM D2466-06. The chamber had a pressure gauge and input valve for the compressed air. In order to reduce costs, a smaller solenoid valve was purchased and reducers were used to connect the solenoid value to the chamber and release tube. The release tube was a two feet long piece of pipe with a diameter of 1.70 inches, leaving just enough room for the golf ball,

which has a diameter of 1.68 inches. This tight fit allowed more air to stay behind the ball as it exited the air canon, meaning higher speeds could be attained. The different pieces of polycarbonate were fastened together with PVC cement to ensure a tight seal. The air canon was surrounded by polycarbonate sheets on two sides and wood sheets on the other sides for safety purposes, in case the chamber failed during testing. The air canon was also attached to a wood column and supported with wood beams at the base to make sure it wouldn't tip over when fired. Custom made foam place holders provided stable support for the air canon inside the enclosure. This ensured that the air canon was set securely in the enclosure and maintained repeatability with testing. To fire the air canon, a trigger was connected to the solenoid terminals and a power supply. The electronically controlled solenoid made sure the air was released efficiently for every test. The air canon drawings and parts can be seen in Appendix A.

Figure 10: Air Canon

In order to perform simulations to try and match experimental results, we needed to know what speeds a golf ball could reach after it had been fired out of the air canon. To find out these speeds, a correlation was needed between the pressure of the chamber and the release velocity. When the air canon was first designed, the primary goal was to make sure we could attain speeds that would be comparable to the speeds a golf ball experiences when it is struck by a golf club. A dynamic analysis was used to analyze the forces acting on the ball throughout the release tube to find out how the ball was accelerating. Once the air canon was assembled, we used a golf ball launch monitor to capture images of the ball coming out of the release tube. We observed that the theoretical calculations did not match the actual speeds the air canon was producing. A more thorough theoretical analysis was performed, which took into account the fact that the pressure was not a constant value acting on the ball. An assumption was made that the release of air through the solenoid valve was a polytropic process. This assumption was based on findings from a Cal Poly experiment, in which air was released from a large chamber through a valve into free space (Volkoff-Shoemaker). The main polytropic equation can be seen in Equation 7, in which a pressure, P , multiplied by a volume, V , to the power of an index value, n , yield a constant. For our case we will assume the same index constant used in the Cal Poly experiment, which was a value of 1.4. By using the work done onto the ball by the air from Equation 8, the work done on the ball can be

converted into a kinetic energy seen in Equation 9. These calculations can be found in Appendix B and D.

$$PV^n = \text{const} \quad 7.$$

$$W_{thermo} = \int P dV \quad 8.$$

$$W_{net} = \frac{1}{2} mv^2 \quad 9.$$

This new theoretical value was closer but still not close enough to the actual speeds. One reason for this discrepancy was that the solenoid valve might have been causing some issues. The flow of air through the solenoid valve could have been undergoing choked flow. Choked flow occurs in certain situations when a fluid flows through a restriction from a higher pressure to a lower pressure (Fox, McDonald and Pritchard). A formula for the minimum pressure ratio requirement for air can be seen in Equation 10 and shows that the initial pressure of the chamber, P_0 , relative to the exit pressure, P_e , has to follow this ratio or the flow will be choked. The pressures we operated at yielded a higher ratio than this minimum pressure requirement, so it was very likely that we were experiencing choked flow.

$$\frac{P_e}{P_o} \geq 0.528 \quad 10.$$

Because the air canon did not produce speeds that were similar to the theory, we had to find a way to experimentally determine the correlation. The golf ball launch monitor was difficult to use for this purpose and something else was needed to get experimental data. Luckily, with the help of Dr. Chen, the ME department was able to let us use a high-speed camera which could capture the golf ball's high speed flight frame by frame. By setting up a known datum in the background of the ball's flight, the speed could be found for different chamber pressures. As Figure 11 shows, the datum in the background was created with a half-inch space of black and white lines for 2 feet of length. The software program we used allowed us to set the datum in the video and then select two locations in the video at different points in time. We had the program analyze the tail end of the ball at two different points in time, and from that the speed could be found. The software program was also validated with our own calculations to make sure we correctly obtained the speed values. These calculations can be seen in Appendix D.

Figure 11: Air Canon Calibration Setup

Once the camera was ready and proper lighting was setup we were able to use the camera and record videos at different pressures. The software was used to find the speeds at these different pressures and the overall results can be seen in Figure 12.

Figure 12: Air Canon Calibration Data

Figure 12 appears to have a slightly non-linear trend, but when we examine our region of interest, which only includes the pressures above 30 psi, we see that it has a much more linear trend that matches the data quite well. This linear trend can be seen in Figure 13 while the theoretical comparison calculations can be seen in Table 3.

Table 3: Air Canon Speed Results

	Speed (mph)
Theory - Constant Pressure Assumption	141.56
Theory - Polytropic Assumption	109.5
Actual Speed	61.19

Figure 13: Air Canon Fitted Data

With this correlation between pressure and speed known, we were able to set a pressure for the air canon and have a good approximation of what the speed of the ball was when it hit the plate.

3.3 Impact Apparatus

The impact area was set up to allow for the necessary data collection. We purchased T-slots in order to construct a sturdy, light weight cube structure, with the plate mounted on one face of the cube. The T-slots were positioned in the best way to try and reinforce the impact area without causing any interference with the acoustic data that was collected. Figure 14 shows pictures of this cube structure with foam and without foam along with the clamping system holding the titanium plate.

Figure 14: Impact Apparatus (a) without foam (b) with foam

The rest of the cube aided in making sure the structure provided a solid foundation for all the impacts that occurred. The cube was also clamped to a base structure that had height adjustments to make sure the ball could be targeted to strike the center of the plate. The area inside the cube provided an environment to take acoustic and strain data. Acoustic foam surrounded the structure to try and simulate an anechoic chamber that would eliminate any sound waves that bounced off objects or walls in the room. The goal was to capture just the sound from the impact so that data could be compared to our simulations. The circuit for the strain gauges was also placed in the enclosure and then connected to the data acquisition system outside of the net. Figure 15 shows the acoustic foam we used to line the entire enclosure.

Figure 15: Acoustic Foam

The actual plate was clamped with brackets that we designed and had made in the manufacturing building at Cal Poly. A picture of the bracket can be seen in Figure 16. Four of these brackets were made so the plate could be properly secured along its outer edges.

Figure 16: Clamping Bracket

The sequence of photos in Figure 17 shows how the plate assembled together with the brackets. Notice that the plate did not have a constant thickness profile and that the thicker layer provided an ideal clamping area for the brackets. The geometric and material properties of the plate will be discussed more in the FEA modeling section.

(a)

(b)

(c)

Figure 17: Brackets and Plate

The holes in the brackets lined up so that screws and nuts could be used to clamp the brackets and plate to provide a secure fit. The holes in the brackets also provided spots where custom T-slot plates could be used to provide a connection between the brackets and the T-slots. This setup can be more clearly seen in Figure 18. This overall structure ensured that the plate was well constrained for the impact testing.

Figure 18: Plate and Bracket Connection to Impact Apparatus

3.4 Strain gauges

To verify the FEA, strain gauges were used to find stress data that was directly compared to LS-DYNA® simulations. A strain gauge is a small device that can be attached to objects which undergo strain. The strain gauge is composed of a long, thin metallic wire that runs in parallel lines with itself along the strain gauge (Reese and Kawahara). When installed properly the strain gauge becomes permanently attached to the object of interest with the help of epoxy and other bonding agents. Figure 19 (a) shows an installed strain gauge while Figure 19 (b) highlights that a strain gauges is one long connected piece of wire (Vishay Micro-Measurements).

(a)

(b)

Figure 19: (a) Actual Strain Gauge (b) Diagram of a Strain Gauge

Strain gauges utilize the concept that an electrical conductor's resistance depends on its length and cross sectional area. When an object of interest is

elongated, the strain gauge reacts by elongating as well, which causes a change in electrical resistance of the gauge. This change in resistance can be measured by using a Wheatstone bridge setup, which is a special configuration of resistors that can be seen in Figure 20.

Figure 20: Wheatstone Bridge

What makes the Wheatstone bridge circuit unique is that when an excitation voltage, V_{EX} , is applied to the circuit, and when the resistors are equal, the output voltage, V_O , will read a value of zero (Dally and Riley). This can be proved by using voltage divider equations to get an equation for V_O , which can be seen in Equation 11.

$$V_O = \left(\frac{R_3}{R_3 + R_4} - \frac{R_2}{R_1 + R_2} \right) V_{EX} \quad 11.$$

A strain gauge can replace a resistor in this circuit and when the strain gauge's resistance changes, the output voltage, V_O , will see a non-zero voltage because of the change in the overall resistance of the circuit. The strain

gauge has a characteristic known as a gauge factor, GF , which is defined as the fractional change in electrical resistance over the fractional change in length. The GF is a set number that is unique, depending on the type of strain gauge, and is the characteristic that links the change in voltage to a strain (Reese and Kawahara).

$$GF = \frac{\frac{\Delta R}{R}}{\frac{\Delta L}{L}} = \frac{\frac{\Delta R}{R}}{\varepsilon} \quad 12.$$

3.4.1 Strain Gauge Selection

There are many different types of strain gauges available from the company Vishay Micro-Measurements and choosing the right one for a given experiment is critical in making sure you get good results. For this experiment there were a few unique factors that required proper strain gauge selection to ensure the results were valid. The different factors that went into choosing the right strain gauge will be discussed and summarized in a table at the end of the section.

3.4.2 Gauge Length

The first step when selecting a strain gauge is to choose an appropriate gauge length. For our experiment a small gauge was needed, because of the small surface area of the plate and the fact that the ball impact had to avoid the

strain gauges. A gauge length of 0.060 inch was chosen and is a gauge that offers good performance while taking up little surface area on the plate (Vishay Micro-Measurements). The plate with the strain gauges can be seen in Figure 21. Note how little surface area there was on the plate to use for strain gauges.

Figure 21: Plate with Strain Gauge Setup

3.4.3 Gauge Pattern/Grid Type

Strain gauges can be categorized into two types of grids: a single grid arrangement and a rosette gauge arrangement. A rosette gauge is a grid that has multiple gauges oriented to capture the three components of plane strain an object can undergo. They are important for individuals interested in understating the complete stress state of an object. For our experiment we wanted to compare a stress in a simulation to a single stress value on an

object, so the entire stress state was unnecessary and as a result a single grid gauge was chosen. The single grid arrangement allowed us to collect the data we needed and compare it to the results from the computer simulations.

3.4.4 Options

There are a few extra options that could be picked to complete the strain gauge set up. These included already installed solder tabs, installed lead wires, or options regarding the protection of the strain gauges. The option for a polyimide film protective coating was chosen for our strain gauges. This coating would provide protection from the contamination of fingerprints and was only 0.001 in thick. This option provided more grid protection and also made soldering easier (Vishay Micro-Measurements). The other options were unnecessary since we would be able to install the strain gauges ourselves.

3.4.5 Resistance

There are two main resistances that can be chosen for a strain gauge from Vishay Micro-measurements. These values are 120 Ω and 350 Ω . In general it is better to choose the higher resistance valued gauge because of the reduction in heat generation of the gauge. Another reason is that the lead wires coming off the strain gauge don't affect the overall resistance as much as a gauge with lower resistance (Vishay Micro-Measurements). Either resistance value was acceptable so the 120 Ω gauge was selected.

3.4.6 Temperature Issues

The testing done involved no large temperature gradients because the experiment was performed at room temperature. Despite the heat generation from the input of voltage into the resistor, the overall experiment did not cause any temperature issues for the strain gauges.

3.4.7 Test Duration

Due to the issue that there could have been unforeseen problems encountered and the fact that different types of tests would have to be setup and performed, the strain gauges had to be operable for an extended period of time. However, the gauges were used for a very limited number of runs and fatigue ended up not being an issue.

3.4.8 Installation Issues

The small amount of area on the plate meant it was important to utilize the area we had to make sure the gauges were installed properly. Using a strain gauge installation guide from Vishay Micro-Measurements and following a few attempts, the gauges were successfully installed on the plate. To prevent the strain gauges from lifting off the plate, the wiring off the gauges was connected to solder tabs which were wired to the data acquisition system.

This configuration can be seen in Figure 19 from earlier in the report.

Another important issue considered was that the intense accelerations

experienced by the plate could lead to either the solder junctions or the strain gauges themselves coming loose. If the strain gauges were even slightly unattached it could lead to some erroneous data.

Overall, there were some initial issues that were encountered when installing the strain gauges, but the solder tabs were crucial in making sure the strain gauges did not come off. The properties of the strain gauges used can be seen in Table 4 below, while the picture of the chosen strain gauge can be seen in Figure 19 from earlier in the report.

Table 4: Strain Gauge Selection

	Value
Gauge Length	0.060 in
Pattern	Linear
Gauge Series	EA
Options	Option E
Resistance	120 Ω
Strain Gauge Name	EA-13-060LZ-120/E

3.5 Data Acquisition

The LDS Analyzer is a multi-purpose data acquisition system that gathers both the stress data and acoustical data. The LDS Analyzer unit used was the MELAB180 system and was the only unit at Cal Poly that had the inputs for both strain and acoustical data collection. Luckily we were able to use this unit for an extended period of time and collect all the data we needed.

The strain input port of the LDS Analyzer is a six lead circular connection that requires the use of a cylindrical connector known as a LEMO cable. A picture of this port can be seen in Figure 22.

Figure 22: Strain Gauge Input Port (LDS-Dactron)

With the LEMO cables and the wiring diagrams from the LDS Analyzer guide the correct configuration was made for the strain gauges (LDS-Dactron). For setting up a system of strain gauges, we have to recall the Wheatstone bridge setup in Figure 20. There are three types of configurations possible in which strain gauges can replace the normal resistors in the Wheatstone Bridge circuit. These can be seen in Table 5.

Table 5: Gauge Setup Types

Setup	Number of Active Gauges
Quarter Bridge	1
Half Bridge	2
Full Bridge	4

The LDS Analyzer is only programmed to perform analysis with either a half bridge or full bridge setup. For the strain gauge work we performed, we

wanted to know the bending stresses at the point where the strain gauges were located. To do this for a half bridge configuration one gauge is placed on the top of the part and the other gauge on the bottom. When a bending moment is experienced by the gauges, one gauge undergoes tension while the other one undergoes compression. This can be seen in Figure 23.

Figure 23: Half Bridge for Strain Gauges

In this configuration one gauge will decrease its electrical resistance, while the other will increase its electrical resistance. When these resistors are placed in a Wheatstone Bridge circuit and then experience these changes in their resistance, the output voltage, V_o , becomes non-zero. Equation 11 for the Wheatstone Bridge circuit can be manipulated with the GF term and become an equation that shows what the output voltage should be in terms of strain. The equation for a half bridge configuration can be seen in Equation 13, while the equation for a full bridge circuit can be seen in Equation 14.

$$\frac{V_O}{V_{EX}} = \frac{-GF\varepsilon}{2} \quad 13.$$

$$\frac{V_O}{V_{EX}} = -GF\varepsilon \quad 14.$$

When comparing a full bridge circuit to a half bridge, you will notice that the full bridge has twice the output as the half bridge. This means the gauges will have more sensitivity and because they produce a high output there will be less noticeable noise in the signal (Vishay Micro-Measurements). When possible, it is best to use the full bridge configuration because of this fact. The different configurations can be seen in Figure 24 when used with the LDS Analyzer (LDS-Dactron).

Figure 24: (a) Half Bridge Configuration (b) Full Bridge Configuration

Once all the connections were made to the LDS Analyzer, the strain gauge settings had to be inputted so the strain data could be recorded. The actual software program used was called “RT Focus Pro” which allowed for a variety of acquisition options. For the strain gauges an excitation voltage, input voltage, and GF had to be set. Once the configuration was set it was important that the bridge was balanced and the shunt calibration tool was used. The balanced voltage value is the offset due to imperfections in the gauge and should be close to zero volts. Shunt calibration is a process in which a known resistor is placed onto a circuit to simulate a load. This changes the overall resistance of the circuit and provides a check to make sure the circuit is working properly. The calibration factor should be close to a value of one. Table 6 below shows the settings that were used for data collection.

Table 6: Data Acquisition Settings for Strain Gauges

	Value
Excitation Voltage	5 V
Input Range	100 mV
Gauge Factor	2.055
Digital Filtering	None
Balanced Voltage	0.0088 V
Stunt Resistor Calibration	0.98

A high-speed golf ball impact is a short impact of approximately 0.5 ms, which means a very high frequency sampling rate was required to gather sufficient data. Using the LDS Analyzer, the maximum frequency for taking

data that could be attained was 9615 Hz, which was an appropriate frequency for gathering the data we needed.

3.6 Strain Gauge Testing

The strain gauge is a valuable tool to use for gathering information about a system, but like the FEA, it needs to be validated to make sure it's working properly and giving appropriate results. Two simple cases were performed to see if the strain gauges were working properly, before testing the impact between the ball and the plate. The first simple case involved a simple cantilever beam loaded at the free end. The strain gauges were placed in a half bridge arrangement at a location on the beam in which the strains could be easily calculated. The other case involved looking at the strains when a force was applied to the center of the plate, with the plate in its clamped state in the impact apparatus. Using plate theory, an approximation for the strains could be found at the strain gauge locations (Ugural).

3.6.1 Strain Gauges: Simple Beam Test

To verify that the strain gauges were working properly a simple cantilever beam was analyzed with strain gauges. Using simple beam theory the bending stresses were calculated and compared with the strain data being output from the LDS Analyzer.

In Figure 25 below, the simple beam has the strain gauges attached in a half bridge configuration near the cantilevered end, where a large value of stress is experienced. The cantilevered beam and the completed circuit on the breadboard can be seen in Figure 26. For all the strain gauge testing a breadboard was used as a connection port for all the wires instead of having a buildup of wires near the strain gauges themselves.

Figure 25: Installed Strain Gauges on Simple Beam

Figure 26: Simple Beam Strain Gauge Setup

An object with a known mass was placed at the end of the cantilever beam, which yielded a concentrated force value acting on the beam. Knowing this force and the fundamentals of beam theory one can find the stresses acting on the gauges (Cook and Young). Using the following equations the stresses and strains were calculated.

$$\sigma = \frac{Mc}{I} \quad 15.$$

$$I = \frac{1}{12}bt^3 \quad 16.$$

$$\varepsilon = \frac{\sigma}{E} \quad 17.$$

In these equations, the stress is dependent on a moment, M , which is a force multiplied by a length, and the type of cross section undergoing the stress. In our case, the beam had a rectangular cross section, which had a moment of inertia, I , defined by Equation 16 with a cross-sectional width of b and thickness of t . Because this material was isotropic stainless steel, the strain can be found using the materials constitutive relationship, in the form of Equation 17.

The LDS Analyzer was used to capture the strain data. As one can see in Figure 27 (a), when the object of known mass was placed on the free end of the beam, the strain reading ramped up and then oscillated around a given

value. This oscillation was also observed visually and was due to the beam's natural frequency which was activated when it underwent a dynamic load change. This trend matched the results seen in the FEA simulations as well and intuitively made sense. Figure 27 (a) shows a picture captured from the data acquisition program, while Figure 27 (b) shows this exported data in Microsoft Excel®. From now on all data will be shown in the exported Excel® format to ease the viewing of results.

(a)

(b)

Figure 27: Strain Data of Dynamic Loading

a) Screen shot b) Exported Data in Excel®

When the beam became static with the mass resting at the free end, the strain reading was a constant value of $83 \mu\epsilon$ (microstrain). This value however did not match the theory from Equations 15-17 and can be seen visually in Figure 28.

Figure 28: Experimental vs. Theoretical Values

There are a few reasons for why this might have happened. The configuration of these gauges was a half bridge setup, which was not as ideal as the full bridge setup. The internal settings of the LDS Analyzer for completing the circuit were unknown, so it was difficult to evaluate what the circuit should have been reading. Another issue had to do with the material properties of the beam. For this analysis we were assuming the beam was a certain material with an elastic modulus. The material could have had a different elastic modulus, which would have affected the results. The clamping mechanism used to hold the beam was not a precise piece of equipment and there could have been some issues with the setup. Another issue had to do with the gauges themselves. It was possible that during the installation of these gauges, they were not properly set to the beam. As a

result it would have been undergoing less change in length and also less change in resistance. This smaller change in resistance meant the strain observed with the data acquisition system would be a smaller value.

Despite these problems, one of the positives that from this test was that the gauges were displaying real time data that made sense overall. The load increased then leveled out to a value that was in the realm of the theoretical value. However, because these results were not analogous to the theoretical value, another test was performed to make sure the gauges were working properly.

3.6.2 Strain Gauges: Central Force Applied to Plate

With the strain gauges attached to the plate and the plate clamped into the impact apparatus, a static test was performed by applying a known force to the center of the plate. Using plate theory, an approximation for the stresses was determined at the locations of the strain gauges on the plate (Ugural). To apply a known force, a load cell was used, which can be seen Figure 29. The load cell was an Omegadyne Model LC101-50 with a serial number of 213135 and range between 0-50 lbs.

Figure 29: Load Cell

The load cell was calibrated by testing an object with a known mass, and making sure the force that was being output matched the weight of the object. Once this was done, the load cell was used to apply 15 lbf to the center of the plate, and using the LDS Analyzer data was captured.

Figure 30: Center Force Setup

The results from this test can be seen in Figure 31. Multiple tests were performed and plotted versus time. Note that although the amount of time seems small, these results were the steady state values for when the force had already been applied.

Figure 31: Central Force Plate Data

The average value of the data for each run was taken and the results can be seen in Table 7.

Table 7: Data Summary of Central Force Tests

	Average Value ($\mu\epsilon$)
Take 1	9.511
Take 2	8.382
Take 3	9.478
Take 4	10.254
All Takes	9.406

For this test a full bridge configuration was used to gather the stresses.

Referring back to Figure 21, you can see there are two gauges on the front and two on the back oriented 90° apart from each other. These were all used in a full bridge setup to gather this strain data as opposed to two half bridge

setups. Because plate theory stresses are axisymmetric it doesn't matter that the gauges were so far apart from each other. As long as the gauges were radially the same distance apart from the center, their stresses would be the same. The reason the gauges were oriented as such was to make sure they would avoid the ball in an impact scenario. If the ball were to hit the strain gauges in the current configuration, it would only disable one gauge instead of two.

The theoretical analysis of this test case involved identifying the type of boundary conditions and the loading conditions of the plate. Looking at the theory, two cases were considered. They both involve a plate that is loaded with a single concentrated force at the center, but they differ in their boundary conditions. For one case, the edges are clamped and for the other the edges are simply supported. Simply supported conditions dictate that there is a translational constraint in all directions while clamped conditions mean there are translational constraints and rotational constraints in all directions. The radial stress, which is what the strain gauges are measuring, is dictated by the following equations.

$$\sigma_r = \frac{6M_r}{t^2} \quad 18.$$

$$\sigma_r = \frac{3Fz}{\pi t^3} (1 + \nu) \ln\left(\frac{a}{r}\right) \quad 19.$$

The radial stress depends on the thickness, t , and the radial moment, M_r , as seen in Equation 18. When the radial moment term is applied for each case, the radial stress for the simply supported condition, Equation 19, and the clamped condition, Equation 20, can be determined. The variables can be better understood from Figure 32.

Figure 32: Theoretical Plate Calculation Variables

Each case was applied with our conditions and the results can be seen in Table 8. The strain data looks incorrect, but as Figure 33 shows the radial stress profile varies as the radius changes and gives different values depending on the case one looks at.

Table 8: Theoretical Calculations

	Radial Moment at Strain Gauge Location (lb-in)	Strain at Strain Gauge Location ($\mu\epsilon$)
Simple Supported	-0.35692	-10.23
Clamped	0.83674	24.04

Figure 33: Radial Strain Profile of Plate

Comparing these results to the experiment we can see that the experimental results were best represented by the clamped case, but it is difficult to make conclusions from this data on whether the strain gauges were malfunctioning. The unique geometry and boundary conditions of our plate as well as the unknown role the impact apparatus was playing make it so we can't match the results perfectly with theory.

From the experimental data of these two tests we could confidently say that the strain gauges were responding properly and are within the correct magnitude of the theory. It became a matter of isolating a corrected gauge factor. Because of these issues, this data along with the impact test data will be looked at more in the section discussing the comparisons between the FEA and experimental data.

3.6.3 Strain Gauges: Impact Test

Once the strain gauges had been tested and worked properly, the impact strain data could be acquired. As mentioned in Section 3.5 of the report when discussing the data acquisition system, it is important to try and get as many points as possible when dealing with such a short time duration impact. The frequency settings were not as critical for the other tests, but for this test the sampling frequency was maximized to make sure enough points were being gathered by the system. A summary of the different settings for each test can be seen in Table 9.

Table 9: Data Acquisition Settings

	Number of Data Points Taken	Time Span (μ s)	Frequency (Hz)
Simple Beam	32768	30.6	3268
Static Plate Force	32768	30.6	3268
Ball and Plate Impact	32768	10.4	9615

For the impact tests the data acquisition settings from the last test were utilized. This included setting the gauges in a full bridge arrangement and also maintaining the same settings on the LDS Analyzer, except for the sampling frequency.

For the impact tests, different air canon pressures were set and using the air canon calibration data, the corresponding golf ball speeds were calculated. Three pressures were implemented to get a collection of strains experienced by the plate. These different pressure values can be seen in Table 10.

Table 10: Different Test Speeds

	Pressure (psi)	Speed (mph)	Speed (in/s)
Test 1	30	61.19	1076.9
Test 2	40	72.36	1273.5
Test 3	50	83.53	1470.1

Obtaining the data for these runs was more difficult than anticipated. Several issues came up that lead to problems with collecting the data. One of these issues was the fact that the strain gauges could not be triggered to capture data. Ideally, when the data acquisition needs to capture a frame, it will use a change in voltage as a trigger. Unfortunately, the strain gauge did not trigger the data collection. The only way to capture the data was to manually time a screen capture in conjunction with the firing of the air canon. This was not an easy task considering we had to capture 0.1 seconds

of data. Another issue was trying to make sure the ball hit the center of the plate, while making sure the ball didn't hit the strain gauges. Many runs were attempted, and it was a difficult task to try and get a center hit that also avoided the strain gauges. Ideally more data would have been better, but the data we did obtain was very useful when comparing it to the FEA simulations. Figure 34 on the next page shows the overall data captured for Test 1. The large impact spike generated is exactly what we wanted to see, while the remaining data dissipates quite quickly. Also notice the two distinct frequencies in the signal. This will be discussed more in the comparison section with the simulation results. The impact spike for this run can be more thoroughly examined in Figure 35. In this plot, there are numerous points that were captured in a very small time span which was helpful in preventing aliasing. Aliasing is the idea that when data isn't sampled correctly, the signal that is captured can become ambiguous when you try to interpret the data. To prevent this, many points were sampled to acquire the data, which is why a high sampling rate was chosen.

Figure 34: Strain Data for First Test at 30 psi

Figure 35: Raw Strain Gauge Impact Data

The data for all the tests were combined into Figure 36. As the figure shows, the frequencies captured were identical for all the tests, which illustrated the consistency in the data.

Figure 36: All Strain Gauge Impact Test Data

The test runs were separated out in Figure 37 to show the progression of the strain peaks. As the pressure increased, the speed increased, meaning more energy was transferred to the plate, and as a result the plate saw higher strains.

Figure 37: All Strain Gauge Impact Test Data Separated

These results at first glance looked promising, but we still didn't have a definitive stance on the strain gauge readings. Later in the report they will be compared to the FEA results from the simulations to see if they are accurate.

Chapter 4

FEA with the USGA Plate/Ball Impact

After the experimental analysis was completed, the FEA simulations involving the impact between the ball, plate, and supporting structure were analyzed. Keep in mind that all the settings mentioned in the LS-DYNA® section of the report, which was Section 2.3, were key features in the simulations that were performed. The specific settings for these simulations will be discussed followed by a presentation of the results in the comparison section with the experimental data.

4.1 The Plate

For modeling physical objects in LS-DYNA® two things were necessary. These two conditions included properly defining both the geometry of the part and the material properties. The geometry of the plate was circular, but also included a cut-out circular extrusion in the middle of the plate. This geometry can be more clearly seen in Figure 38. This plate was modeled after the plate we used for experimental testing, which is why its geometry is unique. Table 11 characterizes the physical dimensions of the plate.

Figure 38: Titanium Plate

Table 11: Plate's Geometric Properties

Plate Dimensions	Value (in)
Minimum Height	0.12
Maximum Height	0.32
Inner Diameter	3
Outer Diameter	4

For a simple isotropic material, the constitutive relationship dictates that only two mechanical properties are needed for LS-DYNA® to run a static analysis. In this case, LS-DYNA® needed to know the elastic modulus and Poisson's ratio. For a dynamic analysis, the density was also required for calculating the mass of the structure in the simulation. The plate was made of Titanium which has the properties seen in Table 12 (Matweb). My colleague, Roger Sharpe, performed a modal analysis of the plate both experimentally and computationally and in his results he recommended a reduced elastic modulus from the given modulus for pure titanium (Sharpe). This number is reflected in the table.

Table 12: Plate's Mechanical Properties

Mechanical Properties	Values	Units
Density	4.14E-04	lb _f -s ² /in ⁴
Elastic Modulus	1.45E+07	psi
Poisson's Ratio	0.34	-

4.2 The Ball

Like the plate, the ball's geometry and material properties had to be defined for the simulation to work properly. The ball we used was a Titleist Pro VI Practice Ball, a high quality golf ball used by many professionals. The picture of the ball we used can be seen in Figure 39 and shows each of the components of the golf ball mentioned in Chapter 1. For this ball, the core takes up a majority of the volume, but this can be different depending on the ball.

Figure 39: Golf Ball Section View

The golf ball's material properties are more complex than a standard material like Titanium, and as a result they had to be modeled as a hyperelastic material. A hyperelastic material is one in which the stress-strain relationship depends on a strain energy density function (Hiermaier). Normally a material can just be modeled as a linear relationship between stress and strain through the materials constitutive relationship, which is seen in Equation 21.

$$\sigma = E\varepsilon \quad 21.$$

A strain energy density function describes a more complex relationship between the stress and strain and experimental data is normally needed to characterize the relationship. Luckily, we were able to find an article that had already performed tests and analysis in determining the hyperelastic properties of a golf ball (Tanaka, Sato and Oodaira). In this paper, the Mooney-Rivlin strain energy density function was used to describe the hyperelastic behavior of the golf ball. This can be seen in Equation 22.

$$W = C10(I_1 - 3) + C01(I_2 - 3) + \frac{1}{D1}(I_3 - 1)^2 \quad 22.$$

The coefficients $C10$, $C01$, and $D1$ in Equation 22 are used for curve fitting purposes and are constrained by the following equations:

$$E = 6(C10 + C01) \quad 23.$$

$$\mu = 2(C10 + C01) \quad 24.$$

$$\mu = E/3 \quad 25.$$

Equation 25 resulted from solving equations 23 and 24, and gave us a relationship between the shear modulus, μ , and elastic modulus, E . The values of the elastic modulus were given in the Tanaka article and Table 13 shows the values as well as what the summation values for $(C10+C01)$ should be for each component.

Table 13: Elastic Moduli of the Golf Ball Components

	Elastic Modulus (MPa)	E (ksi)	E/6 (ksi)
Core	50	7.25	1208.3
Mantle	25	3.625	604.2
Cover	400	58.01	9668.3

Different combinations of $C10$ and $C01$ values were evaluated in simulations and it was found that a high $C10$ value, relative to the $C01$ value, resulted in better simulations. The values we used can be seen in Table 14.

Table 14: Golf Ball Curve Fitting Properties

	C10	C01
Core	1184	24.2
Mantle	592	12.1
Cover	9475	193.4

The next step was to define the rate dependency values of the material. As mentioned in Chapter 1, the golf ball's properties change depending on the rate at which it is stressed. The ball will act differently depending on if you are hitting a driver or a putt. The equation to determine the rate dependency values given in the Tanaka article needed to be converted to match LS-DYNA®'s equivalent form of the equation. The Tanaka article used Equation 26 while LS-DYNA® used Equation 27 to describe the rate dependencies.

$$G(\tau) = \mu \left[1 - g_1 \left\{ 1 - e^{-\frac{t}{\tau_1}} \right\} \right] \quad 26.$$

$$G(\tau) = \sum_{i=1}^n G_i e^{-\beta_i t} \quad 27.$$

The Tanaka article provided the rate dependency values according to Equation 26, which can be seen in Table 15. You'll notice that the golf ball cover does not experience the rate dependencies, but the other two components of the ball do. The shear modulus is found by using Equation 25.

Table 15: Golf Ball Hyperelastic Properties

	μ (psi)	g_1	τ_1 (sec)
Core	2416.7	0.4	4.00E-05
Mantle	1208.3	0.4	4.00E-05
Cover	19336.7	-	-

For LS-DYNA® to use the rate dependencies of the Tanaka article, a summation series needed to be generated from Equation 27 to match Equation 26. As a result, two data points were needed for the two equations to match. These points were found and can be seen in the following table.

Table 16: Golf Ball Modified Hyperelastic Properties

	G_i	B_i (1/sec)
Core - Term 1	1450.0	0.0
Core - Term 2	966.7	25000
Mantle - Term 1	725.0	0.0
Mantle - Term 2	483.3	25000

The golf ball we used and the ball used in the Tanaka article had very similar geometric properties and even though they were not the same ball, they exhibited the general properties we were looking for in our simulations. The dimensions of the golf ball were also taken by analyzing the inner portion of the golf ball. The image of the different sections can be seen in Figure 40 while the mechanical values given in the Tanaka article can be seen in Table 17.

Table 17: Golf Ball Mechanical Properties

	Density ($\text{lb}_f\text{-s}^2/\text{in}^4$)	Poisson's Ratio	Inner Diameter (in)	Outer Diameter (in)
Core	1.08E-04	0.49	-	1.53
Mantle	1.08E-04	0.49	1.53	1.61
Cover	8.90E-05	0.45	1.61	1.68

Figure 40: The Three Components of the Golf Ball

To make sure our model matched the Tanaka ball model, plots were compared in the article to ones we generated with LS-DYNA®. The article included plots of a force-time profile that resulted in a total impact time of 0.6 milliseconds with a peak force of 2000 lb_f . The comparison plot can be seen in Figure 41. The time span and peak force value of a different, older ball model we used did not match up with the Tanaka article, while the improved ball model matched quite well.

Figure 41: Force Profile at Impact

4.3 The Ball and Plate

Developing the parts and meshes for the ball and plate was an iterative process. The goal in FEA is to try and make assumptions and model the structure in the most efficient way possible. For example, it would be a poor decision to create a 3-D model of a cantilever beam to analyze the deflection at the free end, when the use of 2-D beam elements would be much more efficient. For our structure, a few different models were considered and analyzed in the modeling process. Similar models were first attempted, but more complex meshes were needed to model the system more effectively. Figure 42 shows how we started with shell elements and a simpler ball model, then progressed to solid elements, and then to the final more

geometrically accurate iteration in Figure 42 (c). Even though the cross section is not visible Figure 42 (c), it's important to note there were three layers of elements that represented the smaller thickness of the plate, which is the minimum amount of layers needed when using solid elements. For the thicker section, there were eight elements representing the thickness. The reason these values were chosen was because the ratio of the two thicknesses on the part was $3/8$ and the only way to properly tie all the elements and nodes together was to use that ratio or a higher equivalent ratio.

Figure 42: Different Ball and Plate Meshes

- a) Shell Elements
- b) Flat Plate with Solid Elements
- c) Exact Dimensioning of Plate with Solid Elements

The model of the ball progressed from a single component model to the more advanced model seen in Figure 43. Notice in Figure 43 that the outer layers

of the ball model have many solid element layers to represent the small thicknesses of the cover and the mantle.

Figure 43: Golf Ball Mesh

4.4 Mesh Convergence

The concept of a mesh convergence discussed in Section 2.3.7 of the report was applied to the different meshes created, to ensure a solution had converged. The ball and plate meshes were the primary parts analyzed for this convergence to make sure the system at its most fundamental level was converging.

Table 18 shows the different mesh densities of the ball components, the plate, and their respective totals. By using this information and the FEA data obtained we could see if a solution had converged.

Table 18: Number of Nodes for the Different Mesh Densities

	Core	Mantle	Cover	Plate	Total
Mesh 1	53	52	52	596	753
Mesh 2	321	294	294	1572	2481
Mesh 3	997	872	872	2932	5673
Mesh 4	4341	3612	3612	5828	17393
Mesh 5	7393	6062	6062	8244	27761
Mesh 6	11621	9424	9424	12772	43241

A mesh convergence was first performed with a case where a central concentrated force was applied at the center node of the circular plate. Keep in mind that the plate structure being used was the one in Figure 42 (c) and that unique boundary conditions had to be applied. Figure 44 shows that the outer ring of the plate was given clamped boundary conditions to try and simulate the experimental setup. The force applied to the center node was also 15 lb_f, the same value used for the experimental testing. The load was applied by specifying a load curve in LS-DYNA®. In this case, three points were specified for the curve, which characterized a linear increase in force followed by a constant value for the remainder of the simulation. Note the time span for this simulation is not significant, but the profile of the load curve is vital and can be seen in Figure 45.

Figure 44: Boundary Conditions for Plate

Figure 45: Load Curve for Concentrated Force Applied to Plate

To perform a mesh convergence, data at a specific point on the part was analyzed. Deflections, stresses, or other unique data values are possible candidates for this point. In this case, stress data was looked at to try and compare these results to the experimental results. As Figure 46 shows, the

points where the strain gauges were located on the plate were analyzed. The strain data was captured by analyzing the element history of the simulation. LS-DYNA® displayed the element's stress history from the simulation and this data was exported to Microsoft Excel®. Using the material's constitutive relationship for an isotropic material, the strain was found. The radial stresses at both locations were looked at in LS-DYNA® and, as expected, were the same.

Figure 46: Elements Examined for Mesh Convergence

The strain data from each mesh can be seen in Figure 47. The load increased until $4.0\text{E-}04$ seconds, when the constant force value caused the stress to fluctuate around an average value. This was very similar to the experimental data for the simple beam in which the natural frequency played an important role in the time-dependent simulation.

Figure 47: Different Meshes Strain History for Clamped Plate with Force

The results from Figure 47 were evaluated by examining the average value of the strain when the force was a constant value. This average value was plotted against the number of nodes in each mesh in Figure 48 to see if convergence occurred. This data can be summarized in Table 19.

Figure 48: Mesh Convergence Strain Plot of Clamped Plate with Force

Table 19: Mesh Convergence Strain Data for Clamped Plate with Force

	Total Number of Nodes	Average Strain Value (με)	Percent Difference Between Points (%)
Mesh 1	596	3.55	
Mesh 2	1572	6.98	96.78
Mesh 3	2932	10.52	50.57
Mesh 4	5828	12.90	22.69
Mesh 5	8244	14.22	10.22
Mesh 6	12772	14.96	5.17

One of the key aspects of the convergence data was to look at the percent difference between the values for each mesh. As Table 19 shows, this percentage was decreasing, but did not reach a conclusive convergence. One of the difficulties with the mesh convergence of a stress point is that it is difficult to capture the same physical location for every mesh. The different mesh densities cause the elements to change size which means it is the user's

responsibility to try and pick the best point to analyze the stress data.

Having said that, it was a good sign to see a general trend of convergence occurring.

The next convergence that was analyzed involved both the ball and the plate.

In these simulations, the same boundary conditions were used, but the loading came from the ball impacting the plate. The same stress location points that were used in the first case were used for these simulations as well. The results can be seen in Figure 49.

Figure 49: Different Meshes Strain History for the Ball/Plate Impact

The peak strain experienced by the plate was used for the mesh convergence plot. The total number of nodes, which included the nodes of the ball and

plate, were plotted against this peak value. The resulting plot can be seen in Figure 50, along with the data in Table 20.

Figure 50: Mesh Convergence Strain Plot of Ball/Plate Impact

Table 20: Mesh Convergence Strain Data for Ball/Plate Impact

	Total Number of Nodes	Peak Strain Value (µε)	Percent Difference Between Points (%)
Mesh 1	753	261.39	
Mesh 2	2481	662.32	153.38
Mesh 3	5673	1240.20	87.25
Mesh 4	17393	1420.40	14.53
Mesh 5	27761	1351.64	-4.84
Mesh 6	43241	1466.63	8.51

Like the other mesh convergence plot, there was difficulty establishing a constant physical location for the stress between each mesh. However, the plot does show a converging trend nonetheless. When a consistent physical location is examined, like the center node of the plate where the force is

applied, the convergence data looks even better. The normal displacement of the center node on the plate was examined and the data can be seen in Figure 51 and Table 21 for the concentrated load case.

Figure 51: Mesh Convergence Displacement Plot for Plate with Force

Table 21: Mesh Convergence Displacement Data for Plate with Force

	Total Number of Nodes	Average Displacement Value (in)	Percent Difference Between Points (%)
Mesh 1	596	-9.34E-05	
Mesh 2	1572	-1.77E-04	89.65
Mesh 3	2932	-2.71E-04	53.23
Mesh 4	5828	-2.91E-04	7.38
Mesh 5	8244	-2.97E-04	1.79
Mesh 6	12772	-3.00E-04	1.15

The plot shows visually that a convergence definitely occurred. As for the tabular data, the percentage differences between meshes were much smaller at the denser meshes, meaning the solution was converging.

The data was also examined for the ball and plate impact case, and this data can be seen in Figure 52 and Table 22.

Figure 52: Mesh Convergence Displacement Plot of Ball/Plate Impact

Table 22: Mesh Convergence Displacement Data for Ball/Plate Impact

	Total Number of Nodes	Average Displacement Value (in)	Percent Difference Between Points (%)
Mesh 1	753	-1.60E-02	
Mesh 2	2481	-2.18E-02	36.20
Mesh 3	5673	-2.40E-02	9.89
Mesh 4	17393	-2.35E-02	-2.25
Mesh 5	27761	-2.38E-02	1.49
Mesh 6	43241	-2.38E-02	0.03

Like the displacement data from the other case, this data converges nicely and further confirms the convergence of the data.

Overall, both mesh convergence tests showed a solution converged. Looking at the results, Mesh 5 appeared to be the mesh that converged, with a percent difference value below two percent, and as a result it was utilized for the rest of the simulations. After these meshes showed convergence, a few other parts were modeled to more accurately represent the entire experimental system.

4.5 The Bracket

The brackets used in the experiment were modeled in an FEA environment to provide a better representation of the system. The bracket mesh had to deviate slightly from the actual structure of the bracket system in the experiment, but the prominent features can be seen in Figure 53. The part was defeatured by removing the holes and by combining all the individual brackets into one overall structure. The general geometry of the bracket was matched to the experimental parts along with the inner slot that held the plate in place.

Figure 53: Bracket Mesh

4.6 The Supporting Structure

Similar to the progression of changes made to the ball and plate model, the supporting structure evolved from just using the bracket, to adding two T-slot beams, to a final iteration of the entire T-slot structure. The modeling of the T-slots will be discussed first, followed by the two models that use them.

The T-slots were modeled as a beam with a rectangular cross section, despite the fact that the actual part had a very unique cross section. The T-slots that were purchased came with a data sheet that had the moment of inertia term given as 0.0422 in^4 . However, to confirm this number, the cross-sectional property solver of the program SolidWorks® was used to find the moments of inertia of the cross section. Figure 54 shows the cross section as well as the

SolidWorks® solver, which gave an inertial value of 0.0454 in⁴, confirming the validity of the data sheet value. The reason the value was slightly different was because the SolidWorks® part was not an exact representation of the cross section.

Figure 54: T-slot Sectional Properties

This information gave us an insight to the overall stiffness term we could assign to the beam in the FEA simulation. When a force is applied to a beam, the amount it deflects depends on its stiffness. In our case, we were concerned with the force that was exerted on the supporting beams as a result of the impact force. Equations 28 and 29 dictate the displacement-force relationships for a given material in bending.

$$F = K\delta \quad 28.$$

$$F = \left(\frac{3EI}{L^3} \right) \delta \quad 29.$$

Equation 29 shows that the stiffness term, K , depends on the moment of inertia, I , its elastic modulus, E , and the length of the object, L . We used these equations and matched the overall bending stiffness term by using a modified elastic modulus with a standard rectangular cross section.

One of the other issues with modeling the T-slot with a rectangular cross section involved the density. The T-slots were made of aluminum, but because we modeled them as a solid cross-section, the density had to be adjusted as well. From the SolidWorks® section profile, the surface area of the T-slot cross section was 46.2% of the area taken up by a rectangular cross section of similar dimensions. By using this number we made an approximation that the density value was 46.2% of aluminum's density. Considering the difficulty of meshing a T-slot, this is the best way to try and model the supporting structure's mass.

4.6.1 The Two Beam Structure

Figure 55 shows the two-beam supporting structure. The ball was given an initial velocity and placed close to the plate so less time duration was needed

in the simulation. The interactions between the different parts will be discussed in the contact section.

Figure 55: Mesh of Two-Beam Supporting Structure

4.6.2 Contact Settings – Two Beams

The contact settings mentioned in Section 2.3.6 in the report were used for these simulations, but there were also interactions that were specific for this simulation. A contact setting was in place between the three components of the ball and the plate, which dictated the interaction when the ball impacted the plate. There was also a friction setting in place between the ball and the plate. Two articles were found that performed experimental testing to find friction values for a ball impact. The first article found values between 0.25 and 0.5 for the friction coefficients, depending on the type of ball (Ekstrom).

The other article found values in the vicinity of 0.3 for the friction values (Nakasuga and Hashimoto). As a result a conservative value of 0.3 was chosen for the static and dynamic friction coefficients between the ball and the plate. Keep in mind that because this was a normal impact between the ball and the plate, as opposed to an oblique impact, friction ended up not playing a significant role in the simulation. There was also a contact setting between the plate and bracket. This simulated the clamping effect of the bracket without actually fusing the two parts by tying the different nodes together. The T-slot beams, however, were tied together with the bracket to simulate the experimental system. To recall the experimental system, see Figure 18.

4.6.3 Loading Conditions – Two Beams

The plate was loaded in the normal direction due to the head-on impact between the golf ball and the plate. The golf ball was modeled in LS-DYNA® with different speeds to test different loading conditions that the plate encountered, just like in the experiment.

4.6.4 Boundary Conditions – Two Beams

The plate was constrained by the contact definition mentioned in Section 4.6.2, so the ends of the T-slots became the source of the boundary nodes. All of the nodes located on the face of the ends of each T-slot were given a “clamped” boundary condition. This meant these nodes were not able to

translate or rotate in all directions. This simulated the effect of the beams being constrained by the other parts of structure. Figure 56 visually shows the boundary conditions for this structure with the blackened visual marks.

Figure 56: Boundary Conditions for Two-Beam Support Structure

4.6.5 The Entire Structure

The last finite model involved the entire structure and yielded the best representation of the experimental system. The system is very similar to the two beam supporting structure, except in this model more of the impact apparatus was modeled. Figure 57 shows the entire structure. The reason that simpler beam elements were not used for the entire support structure was because we needed the acoustic data that came from the sound emitted from the surface of the solid elements of the structure. All the parts in the system contributed to the acoustics generated from the impact so solid elements had to be used.

Figure 57: Entire Supporting Structure Mesh

4.6.6 Loading/Boundary/Contact Conditions – Entire Structure

The loading conditions and the contact settings for the entire structure were the same as they were for the two-beam support system. The boundary conditions, however, were different due to the added supports. Recall that in the experimental setup, which can be seen in Figure 14, clamps were placed on both sides of two of the supporting T-slots. This meant the nodes at the points where the structure was clamped needed to be constrained. These nodes were given the same type of condition of no rotation or translation in all directions like the bounded nodes for the two beam support structure.

Figure 58 shows the boundary conditions for the entire structure with the indicated black marks.

Figure 58: Boundary Conditions for Entire Supporting Structure

The mechanical properties and the overall geometries of the supporting parts can be seen in Table 23 and Table 24. Notice the difference in the elastic moduli of the two parts because of the cross section adjustment that needed to be made.

Table 23: Supporting Parts Mechanical Properties

	Density (lbf-s ² /in ⁴)	Poisson's Ratio	Elastic Modulus (psi)
Bracket	2.48E-04	0.33	1.00E+07
T-slot	1.145E-04	0.33	5.03E+06

Table 24: Supporting Parts Geometric Properties

	Length (in)	Height (in)	Width (in)
Bracket	6	6	1
T-slot (1 unit)	24	1	1

The number of nodes and elements in the supporting structures were set to try and best represent the experiment while making sure the computational time required to complete the entire simulation remained reasonable. From the mesh convergence we already knew that our ball and plate meshes were valid and as a result the following mesh values were set, which can be seen in Table 25.

Table 25: Number of Nodes and Elements for the Supporting Structures

	Nodes	Elements
Bracket	4500	3264
Two Beam Support Structure	7938	5952
Entire Support Structure	16944	11702

4.7 Section Properties

The simulations involving the ball, plate, and supporting structure were modeled with solid elements. These elements were also assigned quadratic elements and fully integrated. The quadratic element improves upon the assumption of a linear interpolation of the DOF in-between nodes and instead assumes a quadratic profile. In order to profile a quadratic curve

mathematically, three points are needed, so the quadratic element adds a node in-between the already existing nodes on an element. Figure 59 shows the regular elements next to their quadratic element counterpart. The quadratic elements improve upon the accuracy of the solution but take more computational resources to use.

Figure 59: Quadratic Elements

4.8 Model Check

After all of the different parts in the simulations had been created, the models were examined by performing a quality check of the elements. A high quality element is one that maintains its shape so that it represents the “parent” element, like those seen in Figure 59. On the other hand, poor elements are ones which deviate from this geometry. LS-DYNA® has a built-in tool for checking the quality of different finite elements. By using the entire support system mesh, all the parts would be included in the element

check. One key characteristic of having a quality element is to have a low aspect ratio, which is the ratio between the long side and short side of the element. The many thin layers of the mantle and cover, which can be seen in Figure 43, created this problem with the quality of the elements, but in general it is better to have many solid elements to represent these layers. Some of the other elements of the T-slot supporting structure also had a poor aspect ratio, but these elements were able to maintain the hexahedral shape of the “parent” element. These results can be visually seen in Figure 60. In this figure the colors represent a fringe pattern which has a scale on the right side of the figure. The brighter colors represent poor aspect ratios, while the darker colors represent better ratios.

Figure 60: Aspect Ratio of All Parts for Validation Simulations

One of the other element checks that can be made involves the angles of the elements. Ideally it is important to try and maintain the geometry of the original element, the solid hexahedral. However, when you have circular geometries, this is difficult to achieve, and in the end some elements will have a different shape. The modeling of the more complex geometries was performed in TrueGrid® by Roger Sharpe and his meshing techniques can be better understood by examining his report (Sharpe). The golf ball in Figure 43 and plate in Figure 46 show how the circular geometries are manipulated to give a structured and well-defined mesh. This is one of the better ways to discretize a circle and establishes high quality elements in the middle of the plate where the impact takes place. As a result of this circular mesh a few poor angles were established, but alternative meshing arrangements would have yielded even poorer results. The different angles that these solid elements underwent can be seen in Figure 61. The poorer angles are highlighted by the red colors and the proper angles can be identified by the blue colors. As both Figure 60 and Figure 61 show, using a fringe pattern is an easy and quick way to visualize the results.

Figure 61: Maximum Angle of All Parts for Validation Simulations

Despite these issues with the elements, the overall structure contained no tetrahedral elements, which are generally poorer elements, and considering the very unique geometries that had to be modeled, the mesh was able to emulate the geometry of the system to match the experimental setup.

Chapter 5

Comparing FEA to Experimental Results

The Experimental testing and FEA simulations have been performed. Now it's time to look at the results and compare them.

First, the experimental results for the plate in the clamped impact apparatus will be revisited to make sure the strain gauges values were reasonable.

Following this, the three impact tests speeds with experimental results will be compared to the different FEA simulations of the plate, two beam supporting structure, and entire structure at the same speeds. Refer to Table 10 to recall the different speed values for each pressure.

5.1 Clamped Plate with a Concentrated Force

Using the different meshes that were created, the experimental setup and test from the central concentrated force were simulated with LS-DYNA®.

The reason the mesh convergence case of a concentrated force applied to the center node was performed was because it mimicked the experimental setup and provided a basis to compare the two. When the other meshes which

included the supporting elements were also analyzed, the same boundary conditions that were applied for the impact case applied to this case as well. The results can be seen in Figure 62 and Table 26.

Figure 62: Comparison of Concentrated Force Test

Table 26: FEA data for Concentrated Force Case

	Average Strain (με)
Plate	14.22
Two Beam	11.35
Entire	11.49
Experimental	9.32
Simply Supported Theory	-10.23
Clamped Theory	24.04

The overall results looked good and by including the more advanced meshes the finite element results showed a similarity to the experimental results and the theory. It is difficult to arrive at a concrete conclusion about the strain

gauges from these results. The actual experimental values varied from an average of 8.4 to 10.25 $\mu\epsilon$, so it is difficult to say that the average value is a definitive number with so much deviation of the different attempts. This deviation was unfortunately due to the poor resolution of the load cell. One point that could be argued is that the strain gauges should have been reading slightly higher values because of the results from the simple beam test as well as the results that were just presented. Deciding what the exact correction factor should have been was another question. As a result of this dilemma, no correction factor was applied, but these results should be kept in mind when viewing the results of the impact tests.

5.2 Impact Tests

In Chapter 3 of the report the three impact tests were presented and here they will be compared to the finite element cases.

5.2.1 Results

In Figure 63, Figure 64, and Figure 65, we see the data for the three cases of different chamber pressures, which correspond to three different speeds of the golf ball.

Figure 63: Comparison of 30 psi Results

Figure 64: Comparison of 40 psi Results

Figure 65: Comparison of 50 psi Results

One of the first things to notice was the overall similarity in magnitude between the cases. As the different figures show, the “plate only” mesh ball led to high stresses due to the fact that none of the energy in the impact was being dissipated into the impact apparatus. The other meshes, which included supporting structures, had lower strains than the “plate only” case because of the energy that was absorbed by the supporting components. Another noticeable characteristic of these plots is that the FEA results of the two beam supporting mesh and the entire mesh are very similar. By adding the two beams, the visual FEA results showed that the beams deflected and absorbed some of the energy from the impact. This visual aspect was seen in both the entire mesh and the two beam mesh. In the entire mesh, however, you can see there just wasn’t much energy being absorbed by the whole

structure. The modified properties and settings for the T-slots are probably causing this issue. However, because of the unique geometry of the T-slots, there was little that could be done to correct this. It is also important to remember the results from earlier in this chapter, and that the strain gauges could be under-predicting the stresses slightly.

Although a sensitivity analysis was not formally performed for the different FEA settings in the simulation, one aspect to note was the sensitivity of the different elements stress values. The stresses depend largely on the radial location as theoretical calculations in Section 3.6.2 showed. The choice of the stress element had to be made, but did not perfectly represent the stress location of the strain gauge. This is another reason that the experimental strains didn't match up with the simulations.

Another feature to look at is the impact times for the different cases. When we look at each of these figures, we see that the impact time duration matched up well between the experiment and FEA results. This meant that the ball model, which was taken from the Tanaka article, was properly representing the ball we used for the experimental results.

When looking at the 40 psi plot in Figure 64 it does unfortunately show that some of the experimental data was cut off. However, visually it can be seen

in a close up analysis that the curve has some concavity at the top and that the data could be interpolated. This interpolation can be seen in Figure 66.

Figure 66: Interpolated Data from 40 psi run

5.2.2 Impact Hand Calculations

Like the central force test case, it was important to have an idea for what the theoretical strain should have been. A dynamic impact problem however, is much more complicated than a static problem. An impact involves classic mechanics, contact stresses, and 3-D elastic wave propagation (Witteman and Faik). To simplify the problem, a rough approximation of the strains was found just focusing on classical mechanics.

When we used the high-speed camera we filmed a drop test of the ball hitting the plate in the impact apparatus and bouncing back up. A picture from this

video can be seen in Figure 67. In the drop test, the ball was dropped from about 19 inches above the plate and the video was recorded. Using the software program that had already been used for the air canon data, the incoming velocity and outgoing velocity of ball was found which would allow for the calculation of the COR. Two videos were shot and software was used to gather four sets of velocity data seen in Table 27. The drop test video data also gave us an insight into the overall rigidity of the structure. It is clearly visible from the video that the T-slots are flexing and not providing the rigidity we wanted.

Figure 67: Drop Test Picture

Table 27: Drop Test Results

	V1 (mph)	V2 (mph)	COR
Run 1	-6.25	5.02	0.803
Run 2	-6.167	5.07	0.822
Run 3	-6.082	4.785	0.787
Run 4	-6.098	4.773	0.783
Average	-	-	0.799

Knowing the COR, classical mechanics was used along with the strain data time duration of impact to get an estimation of the peak impact force that acted in the impact between the ball and the plate. This can be seen in Equation 30.

$$\Delta G = \int_{t_1}^{t_2} F(t)dt \quad 30.$$

In this equation, the linear change in momentum, ΔG , is equal to the integral of the force-time profile. The force calculation can be seen in Appendix B and makes an assumption that the force-time profile is a parabolic shape just like in the experimental results. Using the linear change in momentum from the impact, the peak force was found. This peak force profile was also compared to the simulation to see if the forces were similar and as Figure 68 shows they were.

Figure 68: Force Profile of Impact for 30 psi

With this information a very rough approximation of the stress was carried out using the same circular plate theories mentioned in Section 3.6.2. The peak strain values and theoretical calculations were compared for all in the cases in Table 28.

Table 28: Summary of Strain Results

	30 psi	40 psi	50 psi
Plate Only - Peak Strain ($\mu\epsilon$)	1352	1603	1864
Two Beam - Peak Strain ($\mu\epsilon$)	1146	1370	1610
Entire - Peak Strain ($\mu\epsilon$)	1143	1349	1615
Plate Only - Peak Force (lb_f)	1592	1905	2243
Two Beam - Peak Force (lb_f)	1530	1850	2182
Entire - Peak Force (lb_f)	1496	1816	2175
Experimental - Peak Strain ($\mu\epsilon$)	1360	1440	1480
Exp. Contact Time for Peak Force (sec)	4.946E-04	4.715E-04	4.615E-04
Theory - Peak Force(lb_f)	1613.6	2001.5	2360.8
Theory SS - Peak Strain ($\mu\epsilon$)	-1103.3	-1367.6	-1614.20
Theory C – Peak Strain ($\mu\epsilon$)	2586.5	3208.3	3784.3

The similarity between the entire structure and two beam structure can be more closely seen in the tabular data. One of the major reasons in the discrepancy between the experimental data and the simulations is the differences in the peak strains between the cases. The experimental data shows about a 50 $\mu\epsilon$ difference between cases while the experimental and theoretical cases have a difference of about 200 $\mu\epsilon$. The peak forces are also listed in the table and show that they have a direct correlation with the peak strains, which is expected. It would be very interesting to know what the force profile was for the actual experimental test. A view of the force/strain relationship for each case can be seen in Table 29. Because the forces relate so well to the strains it can be concluded that something in the experimental setup is causing the energy to dissipate away from the plate and unfortunately this is not captured in the FEA models. The problems with the T-slot modeling mentioned earlier are again confirmed with this data. Note that the theoretical peak strains represent the two different boundary condition cases and that the theoretical results for the statically loaded case showed our conditions were somewhere in-between the two cases.

Table 29: Force/Strain Ratio for Each Case

	30 psi	40 psi	50 psi
Plate Only - Force/Strain	1.178	1.189	1.204
Two Beam - Force/Strain	1.335	1.351	1.356
Entire - Force/Strain	1.309	1.347	1.347
Theory SS - Force/Strain	1.463	1.464	1.463
Theory C – Force/Strain	0.625	0.624	0.624

Despite some of these issues, the FEA models and the theoretical calculations show that the simulations and the theory are matching the values seen in the experimental data.

A quick side note involves mentioning some of the contact mechanics occurring in the problem. When curved objects come into contact, the objects involved will deform slightly and induce stresses which depend on the material properties and geometries of the two objects in contact. In 1882, Heinrich Hertz developed the Hertz Contact Theory between bodies which specifically deals with these unique cases (Witteman and Faik). The case of a ball impacting a half space is one of cases under the Hertz theory of impact and obeys the following relationship from Hertz's findings.

$$F = K_H \delta^{3/2} \quad 31.$$

In this equation, the force, F , depends on the deformation of the bodies, δ , and a constant K_H which depends on the bodies' material and geometric properties. When a golf ball, a spherical object, impacts a plate or the face of a driver, this will induce a contact stress. However, the golf ball is not an elastic isotropic material, and therefore Hertz's theory should not apply, but research has shown that the Hertz Theory can be applied to golf balls (Jones). With actual experimental testing of the golf ball we used some more insight into the forces involved could be gathered to better understand the problem.

5.2.3 Frequency Results

Another characteristic of the plots to examine is the different frequencies. You can see how the plate and ball model only had one natural frequency, while the other models show a clash of frequencies. The main frequency the “plate only” mesh displays, along with the other meshes, matches up quite well with one of the frequencies of the experimental data. These frequencies were measured and are presented in Table 30.

Table 30: Frequencies of Cases for the 30 psi Chamber Pressure

	Frequency 1 (Hz)	Frequency 2 (Hz)	Average (Hz)
Plate Only	5000.0	5263.2	5131.6
Two Beam Support	5263.2	5555.6	5409.4
Entire Support	4545.5	4347.8	4446.6
Experimental	4800.0	4800.0	4800.0

The frequencies can be noticed even more in Figure 69, in which an increased time duration for the simulation is presented and compared to the experimental results.

Figure 69: Longer Time Duration Comparison

The FEA mesh used was the two beam support structure and this mesh exhibited some overall damping but not to the extent of the experimental data. This plot also makes more apparent the presence of the other frequency in the experimental data. This frequency tends to dominate the experimental data and is almost nonexistent in the FEA results. The presence of this frequency is clearly an issue with the entire structure because we know that the higher frequency belongs to the plate. One of the reasons this frequency was not being well represented was because of the modeling of the T-slots. Recall that rectangular cross sections were used with modified properties, which could be influencing the simulation results. Other issues have to do with the fact that the connection between the bracket and the T-slots is not as stiff in real life as it is in the simulated FEA model. All

of the nodes between the T-slot and the bracket are tied, which isn't what is happening in the experimental setup. In the experimental setup there are numerous screws connecting the T-slots to other T-slots and the brackets to one another.

Overall the entire validation process was very useful in determining the proper way to model a real structure in LS-DYNA®. These results confirm that the proper techniques and settings were applied in LS-DYNA® to try and model the real world system.

Chapter 6

The Driver Head/Ball Impact

After the FEA simulations for the USGA plate and ball were validated, simulations were run and results for the impact between the driver head and the golf ball were obtained. For modeling the impact between the ball and the driver head, many of the settings from the USGA plate and ball impact model were used. However, a driver head model had to be implemented for these simulations. Our advisor, Dr. Tom Mase, has acquired years of experience in the golf industry and let us use one of his basic 350 cm³ driver head models seen in Figure 70. Even though this model does not represent the modern driver head of today, it provided us with a proper foundation to modify and simulate.

Figure 70: Driver Head FEA Model

The driver head model had eight parts with each part representing a unique component of the driver head. Typically, each of these components will have their own specific material and geometric properties. When all of the components are meshed together they work cohesively to create an all-in-one unit capable of generating very high golf ball speeds when involved in an impact. For this model, all of the components were given cast titanium alloy

properties and standard thicknesses (Matweb). Titanium is a great material to use for driver heads because of its good strength to weight ratio.

Generally, one alloy, 15V-3Sn-3Cr-3Al, is used for the face component while a different alloy, 6V-4Al, is used for the rest of the structure. In the case of 15-3-3-3 Titanium, the name implies 76% Titanium, 15% Vanadium, 3% Tin, 3% Chromium, and 3% Aluminum. All the components of the driver head are casted, except for the face, which is forged. Because of this fact, the face of the driver head was given a thickness of 2.75 mm, a value in-between the maximum, 3 mm, and minimum, 2.4 mm, thickness values used for driver head faces. To simplify the analysis, a casted titanium alloy was used as the material for the all the driver components with its properties seen in Table 31.

Table 31: Properties of Components of Driver Head FEA Model

	Density (lb _f -s ² /in ⁴)	Elastic Modulus (psi)	Poisson's Ratio	Thickness (in)
Face	5.28E-04	1.03E+07	0.375	0.10826
Crown	-	-	-	0.035
Sole	-	-	-	0.035
Heel	-	-	-	0.035
Toe	-	-	-	0.035
End	-	-	-	0.035
Skirt	-	-	-	0.09
Hosel	-	-	-	0.035

6.1 LS-DYNA® Settings

Like the ball and plate simulation, the driver simulation had its own unique features that had to be modeled in LS-DYNA®.

6.1.1 Section Properties

The golf ball's sectional properties remained the same, but because a driver head was being used instead of the titanium plate, some sectional changes needed to be made. The driver head was composed of shell elements, unlike the titanium plate. Considering the driver head is hollow on the inside and that these simulations were very computationally intensive, this element choice was an ideal one. For our simulations we used fully integrated shell elements for the driver head and fully integrated solid elements for the ball.

6.1.2 Contact Settings

For the driver head and golf ball impact the only contact that occurred was the contact between the golf ball and the face of the driver head. The same contact definition that was used for the ball and plate impact was used for the ball and driver head model.

6.1.3 Rigid to Deformable

One of the settings in LS-DYNA® that was utilized to save computational time was the rigid to deformable switch. With this setting, the program could switch the material properties of an object at certain instances in time. The advantage with this setting was that a material that is normally deformable and requires analysis of the DOF at each node could be switched to a rigid material which undergoes no DOF changes. For the acoustic simulations, the time span required to generate a real sound was much greater than the time span for the impact itself. Because of this, the ball would impact the plate and then rebound and continue into the free space while the driver remained in an energized state from the impact. With the rigid to deformable switch, the ball could be switched to a rigid material before and after the impact takes place, and as a result reduce the computational time required to solve the entire simulation.

6.2 Acoustics Using Boundary Element Method

For the USGA Plate and Ball impact I validated the stress data while the acoustical data was validated by Roger Sharpe. For the club head and golf ball impact the acoustic settings that were verified in the validation phase were applied to the club head and ball simulation.

The acoustic outputs generated were driven in LS-DYNA® by using a Boundary Element Method (BEM) that was modified with custom settings for our simulation. Like all the other commands in LS-DYNA, the BEM uses a keyword command, which is called “BEM_ACOUSTIC.” The BEM is much more efficient than FEA for an acoustic application. When a structure like a driver head becomes energized from the impact with the ball, it will vibrate and give off a sound from these vibrations. The BEM focuses on gathering data concerning just the boundary of that part, and from this, an acoustic profile at a point in free space can be found (Huang and Souli). The acoustic wave propagation is governed by what is known as the Helmholtz equation, seen below.

$$\nabla^2 P + k^2 P = 0 \quad 32.$$

In this equation, P is the pressure at any point in the acoustic medium, k the wave number equals ω/c_{sound} , where c_{sound} is the speed of sound, and ω is a frequency in radians. This equation can be transformed using Green’s theorem to an integral equation that dictates the pressure at any point in the acoustic medium can be found with the knowledge of the pressure and velocity on the boundary. This equation, called the Helmholtz integral equation, takes all the information along the entire surface to find the pressure at any point in the medium. With the pressure known at this point, the acoustic profile can be determined. By using the BEM, this integral can

be broken up by discretizing the boundary of the object of interest. A visualization of these ideas can be seen in Figure 71.

Figure 71: Boundary Element Method

The main advantage of the BEM is that the entire acoustic medium does not have to be discretized, only the boundary of the object of interest. If an FEA approach was taken, the entire domain of the acoustic medium would have to be discretized, a task that would be very computationally intensive. For our impact problem, LS-DYNA® first performed the mechanical analysis of the problem and stored all the velocity data generated into a file which was accessed for the BEM analysis. In this analysis, a Fast Fourier Transform (FFT) was performed on this velocity data to transform it into the frequency domain to create boundary conditions for the BEM (Alia and Souli). The BEM analysis needed to also know the pressure values on the boundary to have all of the boundary conditions needed for the analysis. LS-DYNA®

solved for these pressures by discretizing the integral equation and using a numerical technique called the collocation method (Huang and Souli). The integral equation was applied to each node and a set of equations was solved that yielded the pressures on the boundary. Because each frequency has to be solved for using the BEM, an iterative process solves all the pressures in the given range to find the profile at a given point in the acoustic field. The “BEM_ACOUSTIC” keyword also allows for two other less accurate, but computationally quicker methods to be used for an acoustic analysis. One is called the Rayleigh Method, which only uses the velocities for the boundary conditions and the other is the Kirchhoff method which couples with the FEM analysis and uses an alternate keyword “MAT_ACOUSTIC” (Huang and Souli). For our analysis we did not use these less accurate methods.

6.3 Acoustic Settings in LS-DYNA

The keyword “BEM_ACOUSTIC” needed to be adjusted with specific settings from our simulation. Our acoustic field point was set to a location just behind the driver head’s smile, in a location that would not interfere with the impact area. The acoustic boundary chosen included all the shell elements of the driver head. The acoustic program needs a surface, not a volume, for the boundary element method, so the shell elements again proved to be the better choice of elements for the driver head.

Our simulation also needed inputs for the environment in which the acoustic waves were emitted as well as the frequency range to be examined. A frequency range was specified that was audible to the human ear and the simulation took place in air at standard pressure and temperature. To generate an acoustic sound a longer time duration for the entire simulation was needed. As a result, the time duration was set to 0.25 seconds, which is a large jump from the impact time of 0.6 ms, but this long time duration was needed to capture all the possible frequencies. The acoustic settings were chosen accordingly and can be seen in the Table 32.

Table 32: Acoustic Settings

	Value	Units
Medium	Air	-
Density	1.20E-07	lb _f -s ² /in ⁴
Speed	33450	in/s
Minimum Freq	20	Hz
Maximum Freq	20000	Hz
Time Duration	0.25	seconds

6.4 Loading Conditions

Similar to the ball and plate impact, the driver head was loaded from an impact with the golf ball. The same ball model was used and was set to a speed of 1800 in/s (102.27 mph) which is a speed typical of driver head/ball impacts.

6.5 Boundary Conditions

For the acoustic program to work properly, the driver head had to be constrained so that the fixed nodal acoustic point was in proximity of the acoustic output being generated by the impact between the club head and the ball. As a logical and intuitive choice, the nodes located where the hosel meets the golf shaft were chosen as the constrained nodes and given a “clamped” condition. This allowed the club to remain in a relatively fixed spot but still experience changes in motion due to the energy that it gained from the impact. Sonic holography tests of driver heads show that even with the hosel constrained different modes of the driver heads can be activated. With these activated nodes, unique surface velocities and pressures can be developed which can lead to a proper acoustic profile. The bounded nodes on the hosel can be seen in Figure 72.

Figure 72: Driver Head Boundary Conditions

6.6 Meshing/Model Check

The mesh of the driver head can be looked at closely in Figure 70. The same tool for checking the elements of the other meshes was used for the driver head mesh. The tool was much more robust at checking shell elements than solid elements, so a thorough analysis of the driver head was undertaken. Table 33, Table 34, and Table 35 show a summary of the data that LS-DYNA® generated concerning the driver head mesh.

Table 33: Mesh Density of Components of Driver Head

	Number of Nodes	Number of Elements
Face	437	397
Crown	1299	1242
Sole	606	567
Heel	206	178
Toe	206	178
Smile	400	350
Skirt	92	66
Hosel	79	61
Total	3325	3039

Table 34: Types of Elements in Driver Head Mesh

	Number of Elements
Quad Elements	3034 (99.8%)
Tri Elements	5 (0.165%)

Table 35: Shell Element Quality Checks

	Allowed	Min Value	Max Value	Violated
Aspect Ratio	10	1.01	4.74	0(0%)
Warpage	10	0	25.1	47 (1.55%)
Minimum Quad Angle	45	28.8	89.9	5(0.165%)
Maximum Quad Angle	135	90.1	152	14(0.461%)
Minimum Tri Angle	30	48	52.1	0(0%)
Maximum Tri Angle	120	69.1	83.1	0(0%)
Taper	0.7	0.000731	0.494	0(0%)
Skew	45	0.000187	54.4	1(0.0329%)
Jacobian	0.6	0.49	0.999	9(0.296%)
Time Step	1.00E-06	2.02E-07	1.37E-06	2837(93.4%)

These three tables gave a lot of data about the driver head mesh. The driver head had a reasonable mesh density, as seen in Table 33, which helped keep the computational time reasonable. Table 34 shows there were only 5 elements that were triangular, compared to the majority of quadrilateral elements. Similar to 3-D elements, 2-D quadrilateral elements generally perform better than the 2-D triangular elements. The LS-DYNA® tool was able to thoroughly check the element quality and some of the same issues with the solid elements of the other models can be seen in the shell elements. However, as the far right column of Table 35 shows, for the most part there were very few elements that had poor quality. One of quality issues of having a high aspect ratio can be visually seen in Figure 73. As the figure shows, a few elements in the Skirt component of the driver were poor quality, but nothing too dramatic.

Figure 73: Aspect Ratio Values for the Driver Head Mesh

Another component looked at was the warpage of the elements. As mentioned earlier, the driver head cannot be perfectly modeled, and as a result there were a few elements near the hosel that were warped in order to conform to the geometry of the structure. This can be seen in Figure 74.

Figure 74: Warpage Values for the Driver Head Mesh

Another element check involved looking at the computational integration involved with the shell elements. One of the aspects of integrating elements is that in order to perform the integration, the geometry has to be transformed to the “parent” form, the 2-D quadrilateral. This is done through a term called the Jacobian, which should have a value of one if the element is identical to the “parent” element. In our mesh, there were a few scattered elements with lower Jacobians, which can be seen in Figure 75, but nothing too drastic.

Figure 75: Jacobian Values for the Driver Head Mesh

One of the more interesting checks that LS-DYNA® performed involved looking at the minimum time step that was required for each element. As mentioned in the Section 2.3.2 of the report, the time step is dependent on the element, and LS-DYNA® was able to show which elements required an

increase in the minimum time step. Not surprisingly, the elements that have already been mentioned in the other checks are the ones with the time step issues. The time step plot can be seen in Figure 76.

Figure 76: Required Time Step Values for the Driver Head Mesh

6.7 Mechanical Results of the Simulation

The simulation was run with all these settings already mentioned, except the acoustics, to look at the mechanical results involved and to examine how the driver head/ball interaction was taking place. As Figure 77 shows the ball undergoes a lot of deformation, which is exactly what we want to see. In high-speed videos of a golf ball impact with a driver head this is typical. Visually, the ball model appears to be working properly but does have some

excessive wave propagation amongst the elements that would not occur in a real ball. The deformation of the club head face is also noticed due to the intense loading from the high-speed golf ball.

Figure 77: Driver Head and Ball Impact

Chapter 7

Golf Industry Driver Head Sound Data

Acoustics is a difficult field to grasp analytically, but with real data a basis can be derived to help understand the acoustics of the system being analyzed.

Through some sources in the golf industry we were able to obtain some actual driver head sound data from three different driver heads (NDA). Each driver had a modern shape with a volume of 460 cm³. All three conformed to the standards set for drivers by the USGA and R&A. In each test, the driver head was placed in an anechoic chamber and impacted while five microphones in the chamber captured the sound through a change in voltage sensed by the microphone. The actual time span of the impact was quite short and the cropped audio data can be visually seen in Figure 78. The plots were created by analyzing the “.wav” file audio data in the program MATLAB™.

Figure 78: Wave File Data for the Different Driver Heads

(a) Ping G5 (b) Cleveland Launcher DST (c) Cobra LD

The actual sounds produced by the drivers are somewhat difficult to describe, however, brief verbal explanations will be presented. The sense of sound is a subjective quality and different people might have different impressions of the sounds they hear. Having said that, the audio file was shared amongst different golfers who agreed on the sounds they preferred between the three drivers. The Ping G5 driver sounded tinny and gave off an impression of a weak hit. This is a sound that most people wouldn't prefer for their driver.

The Cleveland Launcher DST sound was also tinny, but sounded more solid than the Ping G5. The Cobra LD had the deepest sound, which also sounded harsh and abrasive. This club was by far the worst sounding of the three. In order to analytically look at the frequency domain of the sound data an FFT was implemented. The results can be seen in Figure 79.

Figure 79: FFT Data for the Different Driver Heads

(a) Ping G5 (b) Cleveland LauncherDST (c) Cobra LD

This data was further manipulated by looking at the region between 20-14,000 Hz, where all of the data of interest was located. This can be seen in Figure 80.

Figure 80: Specific FFT Regions for the Different Driver Heads

(a) Ping G5 (b) Cleveland LauncherDST (c) Cobra LD
 (d) Cobra LD region of interest

The data from Figure 80 is very significant for understanding the types of sounds that a driver head can produce. The purpose of having the data in the frequency domain was to look at which frequency points led to the largest

responses in amplitude. The data analytically represented the type of sound being produced. One thing to notice is the data for the worst sounding club, the Cobra LD. The plot in Figure 80 (d) shows how much larger the peak frequency amplitude is compared to the other plots. Another noticeable feature of the Cobra LD is in Figure 80 (c) where the various peaks in the plot are all concentrated in the lower frequency range, which is why the sound data sounded so harsh and deep. The next plot to notice is that of the Ping G5 in Figure 80 (a). The Ping G5 had three distinct peaks, and it can be concluded from the data and audible sound that it probably gets the tinny sound from the higher frequency peak near 8000 Hz, because its frequencies are very similar to the better sounding Cleveland Launcher DST, except for the frequency near 8000 Hz. The Cleveland Launcher DST had the most favorable sound, with the peaks in the 4000 Hz range and only one other peak in the 6000 Hz range. These are the characteristics that will be discussed more when looking at the simulation acoustic data. Table 36 shows the values of these frequencies and the accompanying magnitude in the plots of Figure 80.

Table 36: Peak Frequency Values for the Three Driver Heads

	Main Peak Frequency Values (Hz) / Magnitude	Minor Peak Frequency Values (Hz) / Magnitude
Ping G5	4372.5 / 87.7	6371.5 / 67 - 8089.2 / 41
Cleveland Launcher DST	4090 / 93.7 - 4180.8 / 85.1	6070 / 60.2
Cobra LD	2824 / 268.3 - 3078.7 / 147 3569.3 / 111	1860.7 / 59.5 - 2224 / 71.2

This data will be compared with our model's results to see if our driver head was exhibiting a favorable sound profile.

Chapter 8

Optimization

Now that results have been obtained for the original driver head configuration, it's time to look at how we can improve the design and make the club produce optimal results.

One area in which engineering has advanced in the design process is through the development of the concept of optimization. Before the concept of optimization, engineering had been performed by building, testing, rebuilding and so forth until a proper design was reached. Optimization can be loosely defined as “achieving the best outcome of a given operation while satisfying certain restrictions” (Stander, Roux and Goel). In engineering the goal is to make things faster, cheaper, lighter, and better. With optimization we can say we want to maximize the performance of a part, but also constrain the weight to a certain amount. This sometimes creates a tradeoff in the design, and by optimizing the problem, the best design can be found.

8.1 Defining an Optimization

The idea of optimization has its roots in mathematics. The mathematical definition of an optimization can be stated as:

$$\begin{aligned} &\text{Find } \mathbf{x}, \text{ that minimizes } f(\mathbf{x}) \\ &\text{subject to } g_j(\mathbf{x}) \leq 0 ; j = 1, 2, \dots, m \\ &h_k(\mathbf{x}) = 0 ; k = 1, 2, \dots, l \end{aligned}$$

In this mathematical description, $\mathbf{x}$ represents the vector of design variables and f is the objective function. The functions g and h represent the constraints in the problem, with indices that represent multiple types of constraints in a given problem (Stander, Roux and Goel).

8.1.1 Design Variable

A design variable is a variable that is changeable in the optimization to produce different results. For our optimization the thicknesses of the different components of the driver head were the design variables. The thickness design variable is considered a continuous variable in which the value can be manipulated precisely, as opposed to a discrete variable which must have a set number, like the number of columns used to support a structure (Papalambros and Wilde). These variables generally have bounds that must be obeyed for real world applications.

8.1.2 Objective

The objective of an optimization problem is a value that you are trying to maximize or minimize. When you have a problem with more than one objective function, the problem becomes a multi-objective or multi-criteria problem (Papalambros and Wilde). In the multi-objective optimization problem, the feasible values for the multiple functions constitute an attainable set. To simulate a single objective optimization subjective weights are used for each objective and then summed to create an overall objective function. When the solutions can be reduced down into many attainable sets and then combined, a Pareto set can be created. Once a Pareto set is identified, the optimal solution, called the Pareto optimal point, can be found. “A point in the design space is a Pareto optimal point if no feasible point exists that would reduce one criterion without increasing the value of one or more of the other criteria” (Papalambros and Wilde). The designer can look at this Pareto set visually and see the different possible designs and then select the point on the curve that meets subjective trade-off preferences. This visual can be seen in Figure 81.

Figure 81: Pareto Front

8.1.3 Constraint

A constraint is a condition in the design that must be met for the design to be considered feasible (Papalambros and Wilde). Typically they are natural constants that cannot be influenced by the user. In our case, the weight of the structure must be constrained to a certain value for the club to be reasonable design. This value cannot be changed by the user.

8.1.4 Problem Solution

An optimization problem is solved using specific mathematical optimization techniques. These techniques will be discussed in the next section.

8.2 LS-OPT[®]

LS-OPT[®] is a program developed by LSTC that acts as an additional tool for LS-DYNA[®] and specializes in optimizing complex problems. LS-OPT[®] provides powerful optimization schemes as well as probabilistic analysis and reliability design options.

LS-OPT[®] 4.0 was the version used for our optimal schemes and has a GUI interface that is fairly user-friendly and incorporates multiple settings for different simulation runs. Like LS-DYNA[®], the input files are text based and can be edited, modified, and manipulated by the user. The GUI has multiple categories for defining the settings of the optimization.

8.2.1 Solvers

The solver setting allows the user to decide which solver packages to use. This includes which pre-processor as well as which post-processor to be utilized. LS-OPT[®] can be interfaced with other programs like TrueGrid[®] and HyperMorph[®], but is primarily designed to work with LS-DYNA[®].

8.2.2 Optimization Methods

To solve an optimization problem, mathematics must be implemented. One of the ways to solve an optimization problem is to use is a gradient-based

solver. This solver uses the first derivatives of the components functions to solve the problem. As a result, these functions must be continuous with continuous first-derivatives (Stander, Roux and Goel). For our simulation, we have a non-linear dynamic analysis that involves an impact scenario. A simulation like this one results in the derivatives of the response functions to be discontinuous because of the chaotic nature of the simulation. This issue with the gradient method has lead to the development of Response Surface Methodology (RSM), which is a statistical method for constructing smooth approximations to functions in multi-dimensional space (Stander, Roux and Goel).

In RSM, a metamodel-based method is used to create and optimize an approximate model of the design. The metamodel is a design space of the all the possible designs that fall within the constraints. The metamodel is also mathematically based and can be modeled as linear, quadratic, or as other more complex functions. The instability of the gradient method is eliminated by the smooth response of the metamodel. The metamodel also captures the entire problem globally and avoids local minima and maxima from noisy responses. Another advantage of the metamodel is that it can be used to find the optimal points and the Pareto Front for multi-objective cases (Stander, Roux and Goel). In Figure 82 a visual representation can be seen that

illustrates the metamodel concept (Stander and Goel, LS-OPT Training Class).

Figure 82: Metamodel Visual Representation

There are a few different types of metamodels that can be used in LS-OPT®. The basic metamodels are simple mathematical functions using polynomials which allow for quicker simulations and work well for basic problems. The more advanced metamodels are based on neural networks, which are complicated representations of a data set that is generated. To get a proper neural network representation, a large data set is required, meaning lots of computational time. The two neural network choices in LS-OPT® are the

Feedforward Network (FF) and the Radial Basis Function Network (RBF). The FF network has a nonlinear regression analysis for modeling while the RBF network uses a linear regression technique (Stander, Roux and Goel). In general the RBF is a better neural network to use because of the shorter computational time needed due to the linear regression. The polynomial models are useful, but the user must choose the type of fit and in more complicated designs the polynomial has difficulties fitting the entire design space. The neural networks provide a global basis and a more extensive view of the possible designs.

8.2.3 Algorithms

Using the metamodels, an optimization algorithm can be used to find the optimum within the metamodel. One of the main algorithms is the Leap Frog Optimizier for Constrained Optimization (LFOCP). This is a gradient-based solver that uses penalty formulations for the constraints (Stander, Roux and Goel). The other main algorithm used is the genetic algorithm (GA) which finds the optimum based on a “survival of the fittest” approach. One of the advantages of the GA is that because of its unique algorithm the solution that is found is global and optimal solutions won’t get caught in local minima and maxima. For multi-objective optimization problems the GA is a useful method because many trade-off solutions can be

found in a single simulation run. One of the disadvantages, however, is that more computational time is required to find the optimal solution.

8.2.4 Strategy

For the metamodel to be found a strategy must be implemented that automates finding the best design set. In LS-OPT[®], there are three types of strategies that can be taken: a single stage, sequential, and sequential reduction with domain reduction (SRSM). The single stage strategy is useful when numerous points are pre-specified before the simulation in order to create a thorough metamodel. The sequential strategy finds a small number of design points for each iteration and then ceases once the solution has converged. These two methods listed work best with non-polynomial metamodels, due to the ability of other models to adjust more to the numerous points sampled. The SRSM method is similar to the sequential strategy, but can be applied to polynomial metamodels due to the fact that the strategy is able to reduce the domain and deal with more manageable sample sets (Stander, Roux and Goel).

8.2.5 Optimization Settings in LS-DYNA[®]

There were a few settings that had to be made for LS-OPT[®] to be able to interact with the LS-DYNA[®] solver. One of the most important settings involved making sure LS-OPT[®] could interpret what the design variables

were in the input file. LS-DYNA® handled this by using a "PARAMETER" keyword which assigned a string of characters to the numerical settings that were supposed to be the design variables. These design variables must be numerical values for the program to properly work. Another setting LS-DYNA® requires involves how LS-OPT® accesses the simulation data when certain variables change. For LS-OPT® to work, the simulation history must be stored in an output database history file so the data can be accessed by the program.

8.3 Driver Head COR Optimization

The goal of this project was to try and understand an acoustical optimization of the sound of the driver head through computer simulations. To do this, the optimization of just the speed of the ball after impacting the driver head, or essentially the COR, was analyzed. The new structural geometry from this optimization was taken and an acoustic analysis was performed on the new configuration. This acoustical data was compared to the original configuration to see the potential for an optimization. An actual multi-objective optimization of the acoustics and the COR will be a project attempted in the future.

8.3.1 Driver Head LS-DYNA settings

The driver head model settings that were mentioned in Chapter 6 of the report were the same settings that were used for the driver head in these simulations. The loading conditions, with the ball traveling at a speed of 1800 in/s, boundary conditions, and contact settings remained the same.

8.3.2 Objectives/Constraints

For this optimization problem the objective was to maximize the rebound velocity of the golf ball after it had impacted the driver head. The club head's eight different components each with their own thickness value were the design variables of the problem. The design variables were given lower bounds that represented the manufacturing feasibility of cast titanium and upper bounds that represented a far enough point in the design to be considered inefficient. The design constraint of this problem was that the mass of the head could not exceed 200 g, or 0.001139 lb_f -s²/in (Mase).

8.3.3 Metamodel Type

The metamodel system used for this optimization was the RBF neural network. Because the future goal is to perform a multi-objective optimization, the RBF neural network metamodel was chosen as the metamodel type. The advantages of this metamodel, mentioned in Section 8.2.2, made it a good choice for this type of optimization problem.

8.3.4 Algorithm

Keeping the goal of an eventual multi-objective optimization in mind, the genetic algorithm was chosen instead of the LFOPC gradient-based algorithm. The genetic algorithm has promise for multi-objective optimization runs in the future.

8.3.5 Histories/Response

The histories and responses represent the actual output data that LS-DYNA® generates. In LS-OPT® you specify which data you want to analyze that will lead to meeting the objectives and constraints. Velocity data was taken from a node near the tail end of the golf ball core and then stored as a response. This node was chosen because it was one of the nodes that best represented the velocity of the overall ball. The mass of the driver, which included all eight components driver head, was also specified in the response.

8.3.6 Optimization Results

LS-OPT® 4.0 has improved from the older versions of the program to have a robust set of visual tools to examine the results of an optimization. First the objectives, constraints, and design variables will be examined.

The results of the objective of the optimization can be seen in Figure 83. Five iterations were implemented to make sure a feasible design was met. We can see the overall trend of the plot is that the magnitude of the velocity is increasing. The negative sign of the velocity just indicates the direction, and as a result it would be incorrect to say the speed of the ball was minimized. Keep in mind that the velocity units for this plot are inches per second. In this plot the black line represents the predicted values of the simulations while the red squares indicate the computed values at the start of each iteration.

Figure 83: Velocity Maximization of Driver Head Optimization

For the optimization to truly find the optimum, the constraint of the mass of the club head had to be met. As Figure 84 shows, the mass was minimized to the given constraint value. Note the units for the mass values in this plot are $\text{lb}_f \cdot \text{s}^2/\text{in}$, the unit requirement from Table 2.

Figure 84: Mass Constraint of Driver Head Optimization

Table 37 shows a summary of the results and how the mass decreased while the velocity increased.

Table 37: Optimization Results for the Objective and Constraint

	Pre Opt Values	Post Opt Values	Percent Difference (%)
Mass	0.00122694	0.0011398	-7.10
Velocity	-1133.94	-1453.34	28.17

For these values to have been met the design variables had to undergo changes and the summary of these changes after the five iterations of the optimization can be seen in Table 38.

Table 38: Optimization Results for the Design Variables

	Pre Opt Values (in)	Max/Min Value (in)	Post Opt Values (in)	Percent Difference (%)
Face	0.10826	0.0885804/0.11811	0.0885804	-18.18
Crown	0.035	0.03/0.06	0.0300008	-14.28
Sole	0.035	0.03/0.06	0.0300003	-14.28
Heel	0.035	0.03/0.06	0.0598747	71.07
Toe	0.035	0.03/0.06	0.0397544	13.58
Smile	0.035	0.03/0.06	0.0433227	23.78
Skirt	0.09	0.04/0.09	0.0400122	-55.54
Hosel	0.035	0.03/0.06	0.0300039	-14.27

The percent difference values on the far right of the table really reveal how the club's geometry changed. In general, more perimeter weighting was placed on the smile, toe, and heel of the club while the face and skirt saw a decrease in thickness. In terms of an overall new design for the club, there are still issues that a true club designer might have with the structure. The thickness of the face was minimized to a value that could present some fatigue issues after lots of use, but by moving the weight from the face to other parts of the club the center of gravity, or CG, of the club is moved

toward the back of the club. As mentioned in Chapter 1 of the report, the driver is designed to generate high speeds and low spins to maximize ball flight. Through the principle of the conservation of angular momentum, a further back CG, relative to the face, leads to less spin imparted on the golf ball. The large difference in thicknesses between the heel and toe could also be considered an issue. Ideally you want the CG in-line with the center of the face, and the different thicknesses would cause a shift in the CG to make it out-of-line with the center of the face.

Besides the main results LS-OPT® provides, there are other components of the optimization that can be looked at with one of these components being the metamodel itself as seen in Figure 85.

Figure 85: Metamodel of Driver Head Optimization

This plot displays the design space of three of the components of the driver head: the velocity of the ball, the thickness of the face, and the thickness of the end. Metamodels were generated for all the relationships and a designer has the ability to explore other design options in the visual tools package of LS-OPT®. Most of the metamodels gave a linear pattern with a slight curve like the one shown here. But as one feasibility plot in Figure 86 shows there was a limited set of design points that were feasible (the green blocks) and many infeasible points (red blocks) for the optimization. With more design points, a better metamodel could've been created, but would've required more computational time.

Figure 86: Feasibility Distribution for the Optimization

Another analysis LS-OPT[®] performs involves the comparison between the computed and predicted values of variables in the optimization. As Figure 87 shows, for the most part there is a linear relationship between the two quantities and that this linear relationship does not deviate in many of the design cases.

Figure 87: Metamodel Accuracy for the Velocity Objective

Overall this optimization was successful at demonstrating the capabilities of LS-OPT[®] and the potential for more advanced optimizations in the future.

Chapter 9

Comparison of Acoustic Simulation Results

After the optimization had been performed to maximize the geometric properties of the driver, the acoustical results needed to be compared. The industry driver head data will be used as a basis for analyzing these results. To compare the data sets, LS-DYNA® was able to output the different plots we needed through a command in the “BEM_ACOUSTIC” keyword.

9.1 Acoustic Results

Figure 88: Amplitude vs. Time for Driver Simulations

The raw sound data generated from LS-DYNA® has some positive aspects along with some negative ones. The overall trend is logical and resembles a typical sound output with an increase to a maximum amplitude followed by the waning of the sound. The amplitude values are also similar to the amplitudes that were seen in the industry driver data. However, the industry driver head data set showed consistently that the sound was an immediate spike in amplitude and no initial buildup was generated. As mentioned in Section 6.3 of the report, a long time duration needed to be set to gather the necessary data. However, the industry driver head data only lasted an average of 0.05 seconds. The reason for the discrepancy had to do with the way the BEM solver works in LS-DYNA®. The program dictates that the longer you let the sound last the more detailed the frequencies will be analyzed. The 0.25 second run was a compromise between what frequencies were analyzed and the computational time required for the simulation. Shorter time duration simulation runs were attempted for the un-optimized case and as Figure 89 shows the results did not represent the system as well as the longer duration runs. The data is cut off at the endpoints and as a result a longer time duration was needed to properly capture the data.

Figure 89: Short Time Duration Acoustics

Similar to the industry driver head data, the raw sound data was taken and an FFT was performed to transform it to the frequency domain. LS-DYNA® was able to generate these plots, but the FFT data was also verified with analysis done in MATLAB™. The results from LS-DYNA® can be seen in Figure 90 while the results from MATLAB™ can be seen in Figure 91.

MATALB™ was also used to generate the sound from the raw sound data in Figure 88. The actual sound produced from the data did not sound like a driver head impact. It sounded very clean and monotonal. The FFT data however provides a useful comparison setting for the acoustic data.

Figure 90: FFT Data from LS-DYNA® for Driver Simulations

Both figures have the exact same FFT analysis, but the MATLAB™ plots on the next page are actually more useful than the LS-DYNA® plots. In general, the acoustics were similar between the two optimized and un-optimized cases with the optimized case having higher amplitudes for some of the frequencies.

Figure 91: FFT Data from MATLAB™ for Driver Simulations

(a) Optimized Results (b) Optimized Results Region of Interest (c) Un-optimized Results

Because MATLAB™ was used to analyze the industry driver heads, for comparison purposes it makes sense to use the same FFT analysis, so the magnitudes can be compared. The magnitudes of the LS-DYNA® plot are completely different and difficult to compare to the industry data. The

MATLAB™ data also shows the frequencies in the higher ranges all the way to 20,000 Hz.

When the optimized and un-optimized plots are compared a few key differences can be seen. Figure 91 (a) shows that the optimized case has much higher amplitudes compared to the un-optimized case. When the same amplitude scale is applied to Figure 91 (b) and Figure 91 (c) a more direct comparison can be made that shows the general higher magnitudes in the optimized case.

A modal analysis of an object can often give a key insight into the acoustics as well. Roger Sharpe performed a modal analysis of the driver head model and found that in general the modal frequencies matched up to the peaks seen in the acoustic optimization (Sharpe). One of the main peaks in both the optimized and un-optimized case, at 364 Hz, is the first major mode of the driver head which is unfortunately a rigid mode generated from the boundary condition. Ideally we were hoping the primary mode shape on the face of the club was the relevant mode in the acoustics because of the impact that takes place. The major peak at 10,800 Hz is not a frequency we expected to see. It did not show up prominently in the modal analysis and is a much higher frequency than ones seen compared to the industry cases. The industry cases also, in general, had lower amplitudes and less high frequency spikes.

Looking back at Chapter 7 and specifically Figure 79, there is very little

going on after 10,000 Hz in all the cases. Our cases have much more prominence in the higher frequencies and this fact would be something that would have to be analyzed more in the simulations. This would include looking at different boundary conditions or moving the acoustic point to different locations. A sensitivity analysis of the BEM method settings could also be performed along with different BEM test cases.

In our acoustic analysis we only looked at the driver head. This did not include any information about acoustics generated from the ball and did not account for the role a golf shaft might play. Each of the concepts could be explored in related projects to better understand the overall acoustics of the problem.

9.2 The Feasibility of an Acoustic Optimization

After looking at this data there is promise of the possibility of an acoustic optimization. The industry driver head data showed us that high magnitudes and higher frequencies can lead to poor sounds. In this case, the optimized driver head gave similar frequencies to the un-optimized case, but had higher magnitudes, especially in the high frequency range. The goal in the optimization is to establish analytical constraints and objectives. This comparison data shows that when an optimization is performed the sound

quality suffers in the form of an increase in magnitudes and more prominence of higher frequencies in the FFT data. Incorporating these characteristics into LS-OPT® would be difficult, but not out of the realm of possibility. A routine created by the user would have to be written that would tell LS-OPT® to perform an acoustic analysis and then apply magnitude and frequency constraints on the FFT data. Another approach would be to run different types of optimizations without acoustics, then run an acoustic analysis and compare the results of a large data set of simulations. One could see what frequencies are triggered by certain changes in the design. This approach would require lots of data for the given system, but would avoid a dependence on LS-OPT® to do all the optimization schemes.

Overall, the results for the potential of an acoustical optimization were promising, but more work would need to be done in the simulation realm to make sure that the data is valid. This would include testing and a more thorough investigation of the acoustic settings in the simulation.

Chapter 10

Discussion and Recommendations

After performing all the tests and analyses there were some aspects that worked well and some that could be improved. The foundations established in this project could provide a great basis for future work to be done on a related project.

10.1 Experiment

The experimental setup had some good aspects, but also some poor ones. The acoustic foam did a great job in isolating the sound and preventing any reverberations. The digital data acquisition was great at taking lots of data in a short time span, but the fact that the strain gauges could not be triggered was very detrimental. The air canon was a functional design and gave us reasonable speeds, but a redesigned version with better housing, mobility capabilities, and improved chamber capacity would be a great improvement. Having the entire experiment take place in the enclosed golf net provided a safe environment when shooting the golf ball at such high speeds. If more testing is done, repeating it in the golf net would be an

appropriate and safe setting. With the acoustic environment and an improved canon many types of projects, even unrelated to golf, could be tested for impact sounds.

One of the big problems with the experiment was trying to design an impact apparatus that was rigid yet would provide a good acoustic environment for testing. We decided to use T-slots for the convenient connectivity options and the customization they could provide. Unfortunately, they were not rigid enough for the impact. This meant, instead of having to just model the plate, the entire structure had to be modeled, which was not what we intended to do at the start. Similarly, the clamping system of the plate could have been designed with more rigidity in mind to reduce the amount of FEA modeling required.

10.2 Computation

One of the issues with using computers is the large amount of computational time they require to run a complex problem. Even though the impact is a short duration event, to get all the acoustic data we wanted, the entire simulation had to run for a much longer time. This was a very difficult process, to have to wait for a 150 hour long simulation to finish and then see the results. After a long simulation a minor mistake could not be realized

until the entire simulation finished. Use of the computer cluster on campus was considered, but in the end we were left running our own simulations. In the future if the cluster computers were used, the optimization simulations would be an ideal candidate for such a system of computers. The cluster computers are designed for parallel computing and when an optimization problem runs numerous test cases, the parallel computers would be able to work on many simulations at once.

Some of the modeling done in LS-DYNA® was good but could be improved upon. Modeling the golf ball with the use of an online article helped us get a basic baseline model. To get a more accurate representation of the ball, some experimental testing of the actual ball we used would be a good improvement. By using the tensile tester machine on campus, the ball's properties could be modeled better with actual experimental data. In terms of the validation structure, a more complicated mesh of the T-slots could be attempted, but would probably increase the computational time greatly. A mesh of the driver head with solid elements representing the face of the club would also be worth pursuing. The solid face mesh could be compared with the shell element face to try and understand which elements work best for the driver head. Altering the driver mesh to fix some of the element quality issues would also be worth pursuing to try and improve upon the modeling of the

driver. With more computational power finer meshes could be used for the driver to model it even more accurately.

10.3 Acoustics

The overall acoustic results from the FEA analysis were not quite what we were hoping for at the beginning of the project. Ideally we were hoping that the audio sound produced from our simulations would be similar to actual driver head data. In terms of improving the acoustic results, a better driver head model could be implemented. The driver head model we were using was a basic model and did not represent an actual driver head well enough. As Figure 94 later in this chapter shows, the actual structure of a driver head has very complicated geometries and shapes. If a golf industry company were to use the acoustics methods that have been utilized in this project they may have more success. By using the more advanced driver head models with detailed dimensioning a more accurate model could be made. The ability to test the different sounds of their driver head models within the company and run the simulations on more powerful computers would also be a great way to improve the acoustic results.

However, the FFT plots did provide a great way to compare the simulation results to the industry driver head data and with our results, along with the

driver head data, we were able to establish a guide as to what frequencies should be avoided and which ones should be studied. Acoustics in general is a difficult field to comprehend, but by using an FFT analysis different designs could be analytically examined and compared.

The different driver head sound files that we received were very useful in determining the types of frequencies that real driver heads can produce. This data set gave us a glimpse into how real driver head's sound and the analytical interpretations that can be made with such data. However, to truly understand the driver head profile, a larger sampling group of drivers would be needed. By testing many different drivers, a robust foundation for the acoustical design of a driver head could be established.

10.4 Optimization

The optimization program LS-OPT[®] is limited, but there are some possible improvements in the program that could lead to some unique design possibilities for the driver head. Getting back to the computational issue, the optimization schemes that we ran needed lots of data, and in order to get that set of data many computations needed to be run. Our facility just didn't have the capabilities to run an intense optimization. With the cluster computing

potential, much better optimization schemes could be run and better metamodels could be produced.

The program LS-OPT[®] shows a lot of promise for being able to perform multi-objective optimizations as well. As Figure 92 shows, LS-OPT[®] is capable of producing a Pareto Frontier, which can be used to find the optimal design point. The case in the in Figure 92 is an example that found the Pareto optimal point for its optimization case (Stander, Roux and Goel, New Developments in LS-OPT 4).

Figure 92: Pareto Fronts Using LS-OPT[®]

LS-OPT[®] does have a few drawbacks, however, and is limited in the fact that the variables that can be chosen are discrete. There is some work being done with LS-OPT[®] to give it the capability to perform topological optimizations

(Stander, Roux and Goel, New Developments in LS-OPT 4). A topological optimization would optimize a part by completely altering the geometry of a solid element mesh structure and would turn it into a part with complicated geometric features. This would be a very nice tool for trying to determine the optimal geometries of all of the components of the golf club. An example case from a presentation about the new developments in LS-OPT® can be seen in Figure 93. In this case a solid block underwent an impact scenario and the entire geometry was optimized for that specific impact (Stander, Roux and Goel, New Developments in LS-OPT 4). After 37 iterations a new much more complicated design was found.

Figure 93: Topological Optimization Example

A club head that has had its crown removed can be seen in Figure 94. This club represents a standard club head design with very intricate mass

distribution of the different components of the club. With the topological optimization these complicated shapes could be found to produce the best possible club designs.

Figure 94: Actual Club Head Design Dimensions

10.5 Future Plans

This project provided a strong basis for acoustical and structural design of driver heads. The experimental setup can be used to do more acoustic testing and gather more information about drivers as well as other golf clubs. LS-DYNA® and LS-OPT® offer many tools for analysis using FEA and with many of the suggestions mentioned in this section there is the potential for some very innovative design work to be done in the golf industry and other disciplines as well.

Chapter 11

Conclusion

This project was a great experience and gave me a tremendous opportunity to tackle a unique engineering problem. The goal of the project was to better understand the acoustics involved in a golf ball and driver head impact, and without a doubt that goal was met. The process to get to that point was a long and difficult task, but as a team obstacles were engaged and goals were met.

A year ago, the basics of the program LS-DYNA® were learned and simple simulations were attempted. An experiment was proposed to make sure the FEA simulations could be verified. In this experiment, a golf ball was fired out of an air canon and into a titanium plate that was fitted with strain gauges and placed in an anechoic chamber so proper acoustic data could be obtained. Unfortunately, in this validation phase due to improper experimental boundary conditions we were unable to hone in on a precise comparison with the finite element analysis. However, this general validation gave us the proper techniques to model the driver head impact. For the acoustical simulation data from the driver head impact, we did not

obtain ideal results, but we did establish a unique methodology for producing distinct sounds from the impact between the driver head and golf ball. The concept that different driver head designs created distinct sounds generated a trade-off between the optimized and un-optimized designs. This trade-off established that with proper analysis an ideal design with a high quality sound profile and mechanical attributes could be obtained.

In terms of a future direction for this project, there are two paths that could be taken. One would be to give these results to a golf industry company, which could improve upon certain aspects of the project because of their corporate standing. With their own driver head models, acoustic facilities, and improved computational potential they could perfect their own methodology process. Another direction to take for this project would be to build on this foundation and explore possible improvements that could be made. Some of these improvements would include access to computational efficiency, implementing the topological optimization when it becomes available, and attaining of a larger set of industry driver acoustic data. For the optimization routine, the improved computational power and topological method could lead to larger data sets and the ability to analyze and improve other design aspects of the driver head. Other overall enhancements for the project would be improving the FE modeling of the driver head by upgrading the models and meshes as well as trying different settings for the acoustic

routine. Improving the understanding and settings in the BEM acoustic routine would help to solidify the acoustic simulation data.

Either of these directions would be a great way to build on the strong basis that has been created for this project. After completing the project, a solid foundation has been set in acoustical optimization for designing driver heads and hopefully with this work more progress can be made in the future to design the best possible driver heads in the industry.

References

- Alia, Ahlem and Mhamed Souli. Acoustic and Vibroacoustic Modeling in LS-DYNA Based on Variational BEM. Report. Livermore, CA: LSTC, 2005.
- Cook, Robert D. and Warren C. Young. Advanced Mechanics of Materials, Second Edition. New Jersey: Prentice Hall, Inc., 1999.
- Dally, James W. and William F Riley. Experimental Stress Analysis. New York: McGraw-Hill, 1991.
- Ekstrom, Erik A. "Experimental Determination of Golf Ball Coefficients of Sliding Friction." Farrally, M.R. and A.J. Cochran. Science and Golf III. United Kingdom: Human Kinetics, 1998. 510-518.
- Encyclopaedia Britannica Online. Golf. 2009. 16 November 2009
<<http://search.eb.com/eb/article-222219>>.
- Fox, Robert W, Alan T McDonald and Philip J Pritchard. Introduction to Fluid Mechanics: Sixth Edition. Hoboken, N.J.: John Wiley & Sons, Inc. , 2004.
- Golf Digest. The Shapes: Evolution of Improvement. February 2009.
November 2009
<http://www.golfdigest.com/equipment/2009/02/hotlistevolution_0902>.

- Hiermaier, Stefan. Structures Under Crash and Impact: Continuum Mechanics, Discretization and Experimental Characterization. New York: Springer, 2008.
- Huang, Yun and Mhamed Souli. Simulation of Acoustic and Vibro-Acoustic Problems in LS-DYNA Using the Boundary Element Method. Report. Livermore, CA: LSTC, 2009.
- Jones, I.R. "Is the Impact of a Golf Ball Hertzian?" Thain, Eric. The Science of Golf: IV. Routledge, 2002. 501-514.
- Jorgensen, Theodore P. The Physics of Golf. New York: Springer-Verlag, 1994.
- LDS-Dactron. RT Pro Dynamic Signal Analysis User Guide v6.3. Manual. Middleton, WI: LDS-Dactron, 2007.
- Livermore Software Technology Corporation. LS-DYNA Keyword User's Manual. Manual. Livermore, CA: Livermore Software Technology Corporation, 2007.
- MacDonald, Bryan J. Practical Stress Analysis with Finite Elements. Dublin, Ireland: Glasnevin, 2007.
- Mase, Dr. Tom. Golf Industry Basics Nickolai Volkoff-Shoemaker. July 2009.
- Matweb. Titanium Material Properties. 2000. November 2009
<<http://www.matweb.com/search/DataSheet.aspx?MatGUID=66a15d609a3f4c829cb6ad08f0dafc01>>.

- Nakasuga, M. and R. Hashimoto. "Measurement of Tangential Force of Golf Ball Impact." Proceedings of the Japanese Society of Mechanical Engineers (1997): 42-45.
- NDA, Golf Industry Member. Golf Industry December 2009.
- Papalambros, Panos Y. and Douglass J. Wilde. Principles of Optimal Design: Modeling and Computation, Second Edition. New York: Cambridge University Press, 2000.
- R&A and USGA. "Rules of Golf." Guide. 2007.
- Reese, Robert T. and Wendell A. Kawahara. Handbook on Structural Testing. Liburn, GA: The Fairmont Press, Inc., 1993.
- Sharpe, Roger. A Prediction of the Acoustical Output of a Golf Driver Head Using Finite Elements. Master's Thesis. San Luis Obispo, CA: Cal Poly, 2009.
- Stander, Nielen and Tushar Goel. LS-OPT Training Class. Presentation. Livermore, CA: LSTC, 2009.
- Stander, Nielen, Willem Roux and Tushar Goel. New Developments in LS-OPT 4. Presentation. Livermore, CA: LSTC, 2009.
- Stander, Nielsen, et al. LS-OPT User's Manual. Manual. Livermore, CA: Livermore Software Technology Corporation, 2009.
- Tanaka, K, et al. "Construction of the finite-element models of golf balls and simulations of their collisions." Proceedings of the Institution of

- Mechanical Engineers, Part L: Journal of Materials: Design and Applications (2006): 13-21.
- Ugural, Ansel C. Stresses in Plates and Shells: Second Edition. New York: McGraw-Hill, 1999.
- Vishay Micro-Measurements. Strain Gauge Selection - Criteria, Procedure, Recommendations. Manual. Toronto: Vishay Micro-Measurements, 2005.
- Volkoff-Shoemaker. ME 346 Lab Experiment: Polytropic Blowdown. Lab Report. San Luis Obispo, 2008.
- Wittteman, Holly and Salah Faik. Modeling of Impact Dynamics: A Literature Survey. Guide. Toronto, Canada: Schneider Electric, 2000.

Appendix A

SolidWorks® Drawings and BOM

In order to design the entire experimental setup the CAD program SolidWorks® was used to create the parts and drawings necessary for the system to come together. The following drawings include the parts and concepts that were drawn up and then produced.

Thesis

Ckd By:	Init:	Drawing By: N. Volkoff-Shoemaker	Init:
Material:	Units: Inches	Drawing #:	
Tolerance:	Scale: 1:12	Title: Air Canon Exploded View	
Next Assy:	Date:	Group:

Thesis

Ckd By:	Init:	Drawing By: N. Volkoff-Shoemaker	Init:
Material:	Units: Inches	Drawing #:	
Tolerance:	Scale: 1:16	Title: Impact Apparatus	
Next Assy:	Date:	Group:	

Bill of Materials for Experiment

Air Canon Parts	Manufacturer	Quantity	Part Number
4" Chamber	Home Depot	1	Stock
Chamber 4" to 2" Reducer	Home Depot	1	Stock
PVC 2" Pipe	Home Depot	1	Stock
PVC 1.5" Pipe	Home Depot	1	Stock
PVC 1" Pipe	Home Depot	1	Stock
1" inch male adapter	http://www.flexpvc.com/	4	436-010
1.5" to 1" reducing bell	http://www.flexpvc.com/	4	429-211
1" reducing couple	http://www.flexpvc.com/	1	429-249
Air compressor Fittings	Home Depot	1	Stock
Air Pressure Gauge	Home Depot	1	Stock
PVC Cement	Home Depot	1	Stock
Solenoid Valve	Mcmaster	1	4738K704
T-slots 1" x 1"	Mcmaster	2	47065T123
T-slots 2" x 1"	Mcmaster	2	47065T126
Polycarbonate Sheet - 48" x 24" x 0.236"	Mcmaster	1	8574K23
T slot connectors	Mcmaster	16	47065T142
90 degree angle connectors	Mcmaster	16	47065T175
Aluminum 12" x 6" x 1/2"	Mcmaster	1	8975K442
Brackets for connecting plate to structure:	Mcmaster	8	47065T177

Appendix B

Hand Calculations

Hand calculations were necessary for the theoretical analysis of the air canon system and the analysis of the forces acting on the plate during impact.

Some of the calculations had to be done by hand while other parts could be done by using the software program MATLAB™. The hand calculations can be seen on the following pages.

Define: A Golf Ball is launched from an air canon by the release of pressure from a chamber through a solenoid Valve. The ball undergoes work through 2 feet of the release tube. The chamber pressure is 30 psi while the ambient pressure is STP.

Find: Net Work done on ball with polytropic assumption

Solution

For polytropic case: $PV^n = \text{const}$

Assuming $n = 1.4$ $PV^{1.4} = \text{const}$, using chamber values:

$$(30 \text{ psi}) \left(\frac{\pi}{4} (4 \text{ in})^2 (15 \text{ in}) \right)^{1.4} = 45977$$

Using the first law of thermodynamics for a closed system:

$$\Delta KE + \Delta PE + \Delta U = Q - W$$

Assuming no ΔPE , ΔU , or Q the equation reduces to:

$$\Delta KE = -W$$

where

$$\Delta KE = \frac{1}{2} m v^2 \text{ of the ball and}$$

$$W = \int_1^2 P dV$$

National Brand

Define:

A golf ball strikes a plate with an incoming velocity of 27.36 m/s, and from video analysis an outgoing velocity of -21.885 m/s. From strain data $t_{\text{contact}} = 0.4946 \text{ ms}$

$$m = 0.0459 \text{ kg}$$

Find:

Max Force exerted on plate

Solution:

Momentum Equation:

$$\Sigma F = m \dot{v} \Rightarrow \frac{d(mv)}{dt} \Rightarrow \Sigma F = \dot{G}$$

$$\int_{t_1}^{t_2} \Sigma F dt = \int \dot{G}$$

$$\int_{t_1}^{t_2} \Sigma F dt = G_2 - G_1$$

$$\int_{t_1}^{t_2} \Sigma F_x dt = (mv_x)_2 - (mv_x)_1$$

$$\Sigma F_x = F_{\max} \sin(\omega t) \quad F_{\max}$$

$$\omega = \frac{\pi}{t_{\text{contact}}} = 6352 \text{ rad/sec}$$

$$G_2 - G_1 = \int_{t_1}^{t_2} F_{\max} \sin(\omega t) dt = \left(\frac{F_{\max}}{\omega} \cos(\omega t) \right) \Big|_{t_1}^{t_2}$$

$$= \frac{F_{\max}}{\omega} \left(\cos(\omega t_2) - \cos(\omega t_1) \right) \quad \text{when } t_2 = 0.495 \text{ ms} \\ t_1 = 0 \text{ sec}$$

$$= \frac{F_{\max}}{\omega} \left(\cos(\pi) - \cos(0) \right)$$

$$G_2 - G_1 = \frac{F_{\max}}{\omega} \begin{pmatrix} -1 & -1 \end{pmatrix}$$

$$(mv_x)_2 - (mv_x)_1 = \frac{F_{max}}{\omega}(-2)$$

$$F_{max} = m \omega (v_{x1} - v_{x2})$$

$$F_{max} = \frac{(0.0459 \text{ kg})(6352 \text{ rad/s})^2}{2} (27.36 \text{ m/s} - (-21.885 \text{ m/s}))$$

$$F_{\max} = 7178,1 \text{ N} \Rightarrow F_{\max} = 1613,6 \text{ lbf}$$

Problem : Derive the expressions for V_o/V_{Ex} for a half bridge and full bridge circuit starting from the following relationship:

Solve:

$$V_o = V_{Ex} \left(\frac{R_3}{R_3 + R_4} - \frac{R_2}{R_1 + R_2} \right)$$

Solution:

$$GF = \frac{\Delta R}{R} \Rightarrow \Delta R = R \epsilon GF$$

for a half bridge: $R_2 = R_1$ while

$$R_3 = R_4 = R_{Gauge} \text{ and } \Delta R = R_G \epsilon GF$$

when a stress is applied to the gauge its overall resistance becomes: $R_G + \Delta R$

$$V_o = V_{Ex} \left(\frac{(R_G - \Delta R)}{(R_G + \Delta R) + (R_G - \Delta R)} - \frac{R_2}{R_1 + R_2} \right)$$

$$V_o = V_{Ex} \left(\frac{R_G - (R_G \epsilon GF)}{2R_G} - \frac{1}{2} \right)$$

$$V_o = V_{Ex} \left(\frac{R_G(1 - \epsilon GF)}{2R_G} - \frac{1}{2} \right)$$

$$\frac{V_o}{V_{Ex}} = \frac{1}{2} \left((1 - \epsilon GF) - 1 \right)$$

$$\frac{V_o}{V_{Ex}} = -\frac{\epsilon GF}{2}$$

Similarly for a full bridge

$$\frac{V_o}{V_{Ex}} = \frac{1}{2} \left((1 - \epsilon GF) - (1 + \epsilon GF) \right)$$

$$\frac{V_o}{V_{Ex}} = -\epsilon GF$$

Appendix C

LS-DYNA® Keyword Example File

As mentioned in the report, LS-DYNA® uses a text-based input file that contains all the information about the finite element analysis to be performed. Provided here in the appendix is an example input file of a plate and ball impact simulation. The code specifies all nodal locations, element assignments, and everything else the LS-DYNA® solver needs to run the file. For the solver to be able to find all the data, the rows and columns must line up so input data can be properly accessed. Some of the nodal and elemental information has been cut short to be more efficient and is indicated by the bracketed text. In the more refined meshes the nodal and elemental data takes up hundreds of pages of data.

Example LS-DYNA Code – Ball and Plate Impact Model (Mesh 1)

```

$# LS-DYNA Keyword file created by LS-PREPOST 2.4 - 30Sep2009(10:00)
$# Created on Nov-12-2009 (21:01:47)
*KEYWORD
*TITLE
$# title
Titanium Plate Modal
*CONTROL_TERMINATION
$#   endtim      endcyc      dtmin      endeng      endmas
      0.002000      0      0.000      0.000      0.000
*DATABASE_GLSTAT
$#   dt      binary      lcur      ioopt
      1.0000E-5      0      0      1
*DATABASE_NODOUT
$#   dt      binary      lcur      ioopt      dthf      binhf
      1.0000E-5      0      0      1      0.000      0
*DATABASE_RCFORC
$#   dt      binary      lcur      ioopt
      1.0000E-5      0      0      1
*DATABASE_BINARY_D3PLOT
$#   dt      lcdt      beam      npltc      psetid
      1.0000E-5      0      0      0      0
$#   ioopt
      0
*BOUNDARY_SPC_SET
$#   nsid      cid      dofx      dofy      dofz      dofrx      dofry
dofrz
      2      0      1      1      1      1      1
1
*SET_NODE_LIST_TITLE
Bound1
$#   sid      da1      da2      da3      da4
      2      0.000      0.000      0.000      0.000
$#   nid1      nid2      nid3      nid4      nid5      nid6      nid7
nid8
      596      307      316      325      334      343      408
417
      426      435      500      509      518      527      578
587
      591      302      311      320      329      338      403

      ...
      ...
[ ALL THE BOUNDARY NODES ]
      ...
      ...

      399      390      381      372      298      289      280
271
      233      542      537      532      455      450      445
440
      363      358      353      348      253      248      243
238
      263      254      561      552      543      483      474
465

```


	456	391	382	373	364	290	281
272							
	221	225	201	193	185	177	17
13							
	9	5	1	61	73	85	97
217							

```

*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_ID
$#      cid
title
      1Contact
$#      ssid      msid      sstyp      mstyp      sboxid      mboxid      spr
mpr
      1          1          2          3          0          0          0
0
$#      fs      fd      dc      vc      vdc      penchk      bt
dt
      0.300000  0.300000      0.000      0.000      0.000          0
0.0001.0000E+20
$#      sfs      sfm      sst      mst      sfst      sfmt      fsf
vsf
      1.000000  1.000000      0.000      0.000  1.000000  1.000000  1.000000
1.000000
*SET_PART_LIST_TITLE
Ball
$#      sid      da1      da2      da3      da4
      1      0.000      0.000      0.000      0.000
$#      pid1      pid2      pid3      pid4      pid5      pid6      pid7
pid8
      5          6          7          0          0          0          0
0
*PART
$# title
material type # 1 (Elastic)
$#      pid      secid      mid      eosid      hgid      grav      adpopt
tmid
      1          1          1          0          0          0          0
0
*SECTION_SOLID
$#      secid      elform      aet
      1          3          0
*MAT_ELASTIC
$#      mid      ro      e      pr      da      db      not used
      1  4.1400E-4  1.6100E+7  0.340000  0.000  0.000          0
*PART
$# title
Ogden Rubber
$#      pid      secid      mid      eosid      hgid      grav      adpopt
tmid
      5          5          5          0          0          0          0
0
*SECTION_SOLID
$#      secid      elform      aet
      5          3          0
*MAT_HYPERELASTIC_RUBBER
$ DEFINITION OF MATERIAL      4
$#      mid      ro      pr      n      nv      g      sigf

```

```

5 1.0760E-4 0.490000 0 2 0.000 0.000
$# c10 c01 c11 c20 c02 c30
1200.0000 24.200001 0.000 0.000 0.000 0.000
$# gi betai
0.600000 0.000
$# gi betai
0.400000 25000.000
*PART
$# title
Ogden Rubber
$# pid secid mid eosid hgid grav adpopt
tmid
6 6 6 0 0 0 0
0
*SECTION_SOLID
$# secid elform aet
6 3 0
*MAT_HYPERELASTIC_RUBBER
$ DEFINITION OF MATERIAL 5
$# mid ro pr n nv g sigf
6 1.0760E-4 0.490000 0 2 0.000 0.000
$# c10 c01 c11 c20 c02 c30
600.00000 24.200001 0.000 0.000 0.000 0.000
$# gi betai
0.600000 0.000
$# gi betai
0.400000 25000.000
*PART
$# title
Ogden Rubber
$# pid secid mid eosid hgid grav adpopt
tmid
7 7 7 0 0 0 0
0
*SECTION_SOLID
$# secid elform aet
7 3 0
*MAT_HYPERELASTIC_RUBBER
$ DEFINITION OF MATERIAL 6
$# mid ro pr n nv g sigf
7 8.9000E-5 0.450000 0 0 0.000 0.000
$# c10 c01 c11 c20 c02 c30
9670.0000 24.200001 0.000 0.000 0.000 0.000
*MAT_RIGID
$# mid ro e pr n couple m
alias
4 4.1400E-4 1.7110E+7 0.340000 0.000 0.000 0.000
$# cmo con1 con2
0.000 0 0
$# lco or a1 a2 a3 v1 v2 v3
0.000 0.000 0.000 0.000 0.000 0.000
*INITIAL_VELOCITY
$# nsid nsidex boxid irigid
1 0 0 0
$# vx vy vz vxr vyr vzr
0.000 0.000-1077.0000 0.000 0.000 0.000

```

*SET_NODE_LIST_TITLE

Ball

\$#	sid	da1	da2	da3	da4		
	1	0.000	0.000	0.000	0.000		
\$#	nid1	nid2	nid3	nid4	nid5	nid6	nid7
nid8							
	597	598	599	600	601	602	603
604							
	605	606	607	608	609	610	611

...

...

[ALL THE BALL NODES FOR THE VELOCITY KEYWORD]

...

...

	645	646	647	648	649	650	651
652							
	653	654	655	656	657	658	659
660							
	661	662	663	664	665	666	667
668							
	669	670	671	672	673	674	675
676							
	677	678	679	680	681	682	683
684							
	685	686	687	688	689	690	691
692							
	693	694	695	696	697	698	699
700							
	701	0	0	0	0	0	0

0

*ELEMENT_SOLID

\$ ELEMENT CARDS FOR SOLID ELEMENTS

\$#	eid	pid	n1	n2	n3	n4	n5	n6
n7	n8							
	1	1	1	21	25	5	2	22
26	6							
	2	1	21	41	45	25	22	42
46	26							
	3	1	5	25	29	9	6	26

...

...

...

[ALL THE ELEMNT ASSIGNMENTS]

...

...

	478	7	668	694	701	675	669	695
697	671							
	479	7	670	696	698	672	675	701
700	674							
	480	7	675	701	700	674	671	697
699	673							

```

*NODE
$#   nid           x           y           z           tc
rc
      1      -1.0606600      -1.0606600      -0.1600000           0
0
      2      -1.0606600      -1.0606600      -0.1200000           0
0
                                ...
                                ...
                                [   ALL THE NODE LOCATIONS   ]
                                ...
                                ...
      860      2.0000000      1.5000000      -0.5000000           0
0
      861      2.0000000      1.5000000           0.000           0
0
      862      2.0000000      2.0000000      -0.5000000           0
0
      863      2.0000000      2.0000000           0.000           0
0
*END

```

Appendix D

MATLAB™ Code Calculations

The robust engineering software program MATLAB™ was used to perform calculations for different analyses in the report. Some of these analyses include the air canon software validation, stress calculations, and acoustic plotting used in the report.

```

I:\School\Thesis\MATLAB\AirCanonCalcs.m
%Air Canon Calculations
%Thesis
%List of Variables
% P_chamb - chamber pressure
% P_amb - ambient pressure
% d_ball - diameter of the ball
% A_ball - surface area of the ball
% F_net - net force acting on the ball
% m_ball - mass of the ball
% L - length of release tube
% W - work
% v - speed of the golf ball
% L_chamb - length of the pressure chamber
% V_chamb - volume of the pressure chamber
% d_chamber - diamter of the pressure chamber
% d_tube - diamter of the release tube
% V_tube - volume of the release tube
% L_other - length of the middle section of air canon
% V_other - volume of the middle section of air canon

clc
clear

%Pressures - Ambient and Chamber in SI and English Units
P_chamb_psi = 30; %psi
P_chamb_Pa = P_chamb_psi*6894.7573; %Pa
P_amb_psi = 14.7; %psi
P_amb_Pa = P_amb_psi*6894.7573; %Pa

%Diameter of the Golf Ball in SI and English Units
d_ball = 1.68; %in
d_ball_m = 1.68*0.0254; %m

%Golf Ball Surface Area in SI and English Units
A_ball_in2 = pi*0.25*d_ball^2; %in^2
A_ball_m2 = pi*0.25*d_ball_m^2; %m^2

%Net Pressure Force in SI and English Units
F_net_lb = (P_chamb_psi - P_amb_psi)*A_ball_in2; %lb
F_net_N = (P_chamb_Pa - P_amb_Pa)*A_ball_m2; %N

%Mass of the Golf Ball in SI and English Units
m_ball_kg = 1.62*0.0283495231; %kg
m_ball_slug = 1.62*0.00194255939; %slug

%Acceleration of the Golf Ball in SI and English Units
a_ball_ins2 = F_net_lb*12/m_ball_slug; %in/s^2
a_ball_ms2 = F_net_N/m_ball_kg; %m/s^2

%Length of the Release Tube in SI and English Units
L_ft = 2; %ft
L_in = 2*12; %in
L_m = 2*0.3048; %m

```

```

%Work Done on Ball by Pressure in SI and English Units
W_lbft = F_net_lb*L_ft; %lb-ft
W_lbin = W_lbft*12;
W_J = F_net_N*L_m; %N-m

%Speeds of golf ball from Work-Energy and Particle Kinematics
v_ins = 12*sqrt(2*W_lbft/m_ball_slug); %in/s
v_ms = sqrt(2*W_J/m_ball_kg); %m/s
vf_ins = sqrt(2*a_ball_ins^2*L_in); %in/s
vf_ms = sqrt(2*a_ball_ms^2*L_m); %m/s
vf_mph = vf_ms*2.23693629;

%Chamber Length, Diameter, and Volume in SI and English Units
L_chamb_in = 15; %in
L_chamb_m = 15*0.0254; %m
d_chamb_in = 4; %in
d_chamb_m = 4*0.0254; %m
V_chamb_in3 = pi*0.25*d_chamb_in^2*L_chamb_in;
V_chamb_m3 = pi*0.25*d_chamb_m^2*L_chamb_m;

%Constant Value for the polytropic case
const = P_chamb_Pa*V_chamb_m3^1.4;
const_eng = P_chamb_psi*V_chamb_in3^1.4;

%Diameter, Volume of the release tube in SI and English Units
d_tube_in = 1.7; %in
d_tube_m = 1.7*0.0254; %m
V_tube_in3 = pi*0.25*d_tube_in^2*L_in; %in^3
V_tube_m3 = pi*0.25*d_tube_m^2*L_m; %m^3

%Volume and Lengths of the middle section of air canon in SI and English Units
L_other_in = 4; %in
L_other_m = 4*0.0254; %m
V_other_in3 = pi*0.25*d_tube_in^2*L_other_in; %in^3
V_other_m3 = pi*0.25*d_tube_m^2*L_other_m; %m^3

%Net Volume of chamber in English and SI
V_net_in3 = V_tube_in3 + V_other_in3 + V_chamb_in3;
V_net_m3 = V_tube_m3 + V_other_m3 + V_chamb_m3;

%Net volume change for golf ball being fired
V_work_m3 = (V_net_m3 - (V_chamb_m3 + V_other_m3));
V_work_in3 = (V_net_in3 - (V_chamb_in3 + V_other_in3));

%Ambient Work done on ball with Polytropic Case
W_J_amb = P_amb_Pa*V_work_m3;
W_lbfin_amb = P_amb_psi*V_work_in3;

%From hand calculations
W_net_lbfin = 1287.75 - W_lbfin_amb;
W_net_lbfft = W_net_lbfin/12;

%New speeds values with polytropic assumption
v_ins_polytropic = 12*sqrt(2*W_net_lbfft/m_ball_slug); %in/s
v_mph_polytropic = v_ins_polytropic*0.0568181818;

```

```
%Significant Solutions
%
% vf_mph =
%
% 141.5643
%
% v_mph_polytropic =
%
% 109.4956
```



```

I:\School\Thesis\MATLAB\momentum.m

%Momentum Caluclations

%Nickolai Volkoff-Shoemaker
%Thesis
%Variables
% m - mass
% v1 - speed at 1
% v2 - speed at 2
% t_contact - time duration of contact
% w - frequency
% Fmax - max force
% Fmax_lb - max force in pounds force
% COR - coeffictient of restitution

clc
clear

%Mass
m = 0.0459; %kg

%Velocities, V1 set to 30, 40, 50 psi chamber value
v1 = [-27.356 -32.347 -37.34] ; %m/s (1077 in/s, 1273.5 in/s, 1470.1
in/s)

COR = 0.80;
v2 = -v1*COR; %m/s

%Time of contact used from the Strain Gauge Impact Data for 30 psi Test
Run
t_contact = [0.49459e-3 0.4715e-3 0.46145e-3]; %s

%Number of points in data set
n = length(v1);

for i = 1:n

    %Frequency for the half period sine wave plot
    w(i) = pi/t_contact(i);

    %Through integration a max force can be found in SI and English
    Units
    Fmax_N(i) = m*w(i)*(v1(i)-v2(i))/2;
    Fmax_lb(i) = Fmax_N(i)*0.2248;

end

% Significant Solutions
%
% Fmax_lb =
%
% 1.0e+003 *
%
% -1.6136 -2.0015 -2.3608

```

```

I:\School\Thesis\MATLAB\speedcalc.m
%Speed of Air Canon Hand Calculation to Validate Software from
%High Speed Camera
%Nickolai Volkoff-Shoemaker
%Thesis
%Variables
% fps - frames per second
% fr1 - frame number at 1
% fr2 - frame number at 2
% d - distance between points
% t - time duration
% v - speed
% v_mph - speed in mph
% t1 - time at 1
% t2 - time at 2
% dt - delta time
% dalt - alternate distance
% valt - alternate speed

clc
clear

%Frame information
fps = 4000;
fr1 = 1228;
fr2 = 1185;

%Distance
d = 12;
%Time
t = (fr1-fr2)/fps;

%Speed
v = d/t;
v_mph = v*3600/(12*5280)
%Alternate Calculation for speed
t1 = 9000e-6;
t2 = 13500e-6;
%delta t
dt = t2 - t1;

%distance in program
d_alt = 5.3;
%alternate speed
v_alt = d_alt/dt;
v_mph_alt = v_alt*3600/(12*5280)

%Significant Solutions
%
% v_mph =
%
% 63.4249
%
% v_mph_alt =
%
% 66.9192

```

```

I:\School\Thesis\MATLAB\forcesPlateReportSingleImpact.m
%Force in Center of the Plate

%Nickolai Volkoff-Shoemaker
%Thesis
%Variables
% E - Elastic Modulus
% v - poissons Ratio
% D - constitutive relationship
% P - Force applied to center of plate
% t - thickness
% a - radius
% delta - deflection
% Mr - radial moment
% Mth - angular moment
% c - distance to stress point
% sigma - stress
% eps - strain
% eps_micro - strian in microstrain

clc
clear

%Mechanical and Geometric Properties
E = 14.5E6;
v = 0.34;
t = 0.120;
z = t/2;
D = E*t^3/(12*(1-v^2));

%Force
P = 1613.6;

%Radius of Plate
a = 1.5;

%Radial Point of Strain Gauge
r = 1.2;

%Deflection of center of plate
delta = (P/(16*pi*D))*(2*r^2*log(r/a)+a^2-r^2);
delta_center = P*a^2/(16*pi*D)
delta_ss = (P/(16*pi*D))*(2*r^2*log(r/a)+((3+v)/(1+v))*(a^2-r^2));
delta_ss_center = (P*a^2/(16*pi*D))*((3+v)/(1+v));

%Moments at r = a for clamped case
Mr_c_edge = -0.0796*P;
Mth_c_edge = -0.0239*P;

%Moments at Strain Gauge Location for Both Cases
Mr_sgc = (P/(4*pi))*((1+v)*log(a/r) - 1 );
Mth_sgc = (P/(4*pi))*((1+v)*log(a/r) - v );
Mr_ss = (P/(4*pi))*((1+v)*log(a/r));
Mth_ss = (P/(4*pi))*((1+v)*log(a/r) + 1 - v );

```

```

%Stress and Strain
%Clamped Edges Stresses
sigma_th_c_edge = 6*Mth_c_edge/t^2;
sigma_r_c_edge = 6*Mr_c_edge/t^2;

%Clamped Edges Strains
eps_th_c_edge = sigma_th_c_edge/E;
eps_r_c_edge = sigma_r_c_edge/E;
eps_r_c_micro_edge = eps_r_c_edge*10e5;

%Simple Supported Stresses for Simply Supported at Strain Gauge
Location
sigma_th_ss = 6*Mth_ss/t^2;
sigma_r_ss = 6*Mr_ss/t^2;

%Simple Supported Strains for Simply Supported at Strain Gauge Location
eps_th_ss = sigma_th_ss/E;
eps_r_ss = sigma_r_ss/E;
eps_r_ss_micro = eps_r_ss*10e5;

%Stresses at Strains for Clamped at Strain Gauge Location
sigma = 6*Mr_sgc/t^2;
eps = sigma/E;
eps_micro = eps*10e5;

%Stresses Calculated with Pre-set Formulas
sigma_r_ss_form = ((3*P*z)/(pi*t^3))*(1+v)*log(a/r);
eps_r_ss_form = sigma_r_ss_form/E;
eps_r_ss_micro_form = eps_r_ss_form*10e5
sigma_r_c_form = ((3*P*z)/(pi*t^3))*((1+v)*log(a/r)-1);
eps_r_c_form = sigma_r_c_form/E;
eps_r_c_micro_form = eps_r_c_form*10e5

%Significant Solutions
%
% eps_r_ss_micro_form =
%
% 1.1033e+003
%
%
% eps_r_c_micro_form =
%
% -2.5865e+003

```

I:\School\Thesis\MATLAB\StressCalcsSimpleBeam.m

```
%Stress Calcs for Simple Beam
%Nickolai Volkoff-Shoemaker
%Thesis
%Variables
% L - length to gauge from point load
% L_tot - total length of unclamped portion
% m - mass of object
% a - acceleration
% F - force
% F_lb - force in pounds
% E - Elastic Modulus
% b - width of cross section
% h - height of cross section
% I - moment of inertia of cross section
% del - deflection
% M - moment
% c - distance to stress point
% sigma - stress
% eps - strain

clc
clear
%Lengths
L = 3.875; %in
L_tot = 4.125; %in
%Mass of object
m = 87.37/1000; %kg
a = 9.81; %m/s^2
F = m*a; %N
F_lb = F*0.2248;
%Modulus for Stainless Steel
E = 30E6; %psi
%Cross Sectional Dimensions
b = 0.864; %in
h = 0.037; %in
I = (1/12)*b*h^3; %in^4
%Deflection at End of Beam
del = F_lb*L_tot^3/(3*E*I); %in
%Moment at Strain Gauge Location
M = F_lb*L;
c = h/2;
%Stresses and Strains at Strain Gauge Location
sigma = M*c/I;
eps = sigma/E;
eps_micro = eps*10E5;

%Significant Solution
%
% eps_micro =
%
% 126.2444
```

```

I:\School\Thesis\MATLAB\forcesPlateReportSingle.m
%Force in Center of the Plate
%Nickolai Volkoff-Shoemaker
%Thesis
%Variables
% E - Elastic Modulus
% v - poissons Ratio
% D - constitutive relationship
% P - Force applied to center of plate
% t - thickness
% a - radius
% delta - deflection
% Mr - radial moment
% Mth - angular moment
% c - distance to stress point
% sigma - stress
% eps - strain
% eps_micro - strian in microstrain

clc
clear

%Mechanical and Geometric Properties
E = 14.5E6;
v = 0.34;
t = 0.120;
z = t/2;
D = E*t^3/(12*(1-v^2));

%Force
P = 15;

%Radius of Plate
a = 1.5;

%Radial Point of Strain Gauge
r = 1.2;

%Deflection of center of plate
delta = (P/(16*pi*D))*(2*r^2*log(r/a)+a^2-r^2);
delta_center = P*a^2/(16*pi*D)
delta_ss = (P/(16*pi*D))*(2*r^2*log(r/a)+((3+v)/(1+v))*(a^2-r^2));
delta_ss_center = (P*a^2/(16*pi*D))*((3+v)/(1+v));
%Moments at r = a for clamped case
Mr_c_edge = -0.0796*P;
Mth_c_edge = -0.0239*P;
%Moments at Strain Gauge Location for Both Cases
Mr_sgc = (P/(4*pi))*((1+v)*log(a/r) - 1 );
Mth_sgc = (P/(4*pi))*((1+v)*log(a/r) - v );
Mr_ss = (P/(4*pi))*((1+v)*log(a/r));
Mth_ss = (P/(4*pi))*((1+v)*log(a/r) + 1 - v );

%Stress and Strain
%Clamped Edges Stresses
sigma_th_c_edge = 6*Mth_c_edge/t^2;
sigma_r_c_edge = 6*Mr_c_edge/t^2;

```

```

%Clamped Edges Strains
eps_th_c_edge = sigma_th_c_edge/E;
eps_r_c_edge = sigma_r_c_edge/E;
eps_r_c_micro_edge = eps_r_c_edge*10e5
%Simple Supported Stresses for Simply Supported at Strain Gauge
Location
sigma_th_ss = 6*Mth_ss/t^2;
sigma_r_ss = 6*Mr_ss/t^2;

%Simple Supported Strains for Simply Supported at Strain Gauge Location
eps_th_ss = sigma_th_ss/E;
eps_r_ss = sigma_r_ss/E;
eps_r_ss_micro = eps_r_ss*10e5;

%Stresses at Strains for Clamped at Strain Gauge Location
sigma = 6*Mr_sgc/t^2;
eps = sigma/E;
eps_micro = eps*10e5;

%Stresses Calculated with Pre-set Formulas
sigma_r_ss_form = ((3*P*z)/(pi*t^3))*(1+v)*log(a/r);
eps_r_ss_form = sigma_r_ss_form/E;
eps_r_ss_micro_form = eps_r_ss_form*10e5;
sigma_r_c_form = ((3*P*z)/(pi*t^3))*((1+v)*log(a/r)-1);
eps_r_c_form = sigma_r_c_form/E;
eps_r_c_micro_form = eps_r_c_form*10e5;

%Significant Solutions
%
% eps_r_ss_micro_form =
%
% 10.2563
%
% eps_r_c_micro_form =
%
% -24.0443

```

```

I:\School\Thesis\MATLAB\forcesPlateAGAINPLOTS.m
%Force in Center of the Plate
%Nickolai Volkoff-Shoemaker
%Thesis
%Variables
% E - Elastic Modulus
% v - poissons Ratio
% D - constitutive relationship
% P - Force applied to center of plate
% t - thickness
% a - radius
% delta - deflection
% Mr - radial moment
% Mth - angular moment
% c - distance to stress point
% sigma - stress
% eps - strain
% eps_micro - strian in microstrain

clc
clear

%Properties
E = 14.5E6;
v = 0.34;
t = 0.120;
z = t/2;
D = E*t^3/(12*(1-v^2));

%Force
P = 15;

%Radius of Plate
a = 1.5

%Deflection of center of plate
delta = (P/(16*pi*D))*(2*r^2*log(r/a)+a^2-r^2);
delta_center = P*a^2/(16*pi*D)
delta_ss = (P/(16*pi*D))*(2*r^2*log(r/a)+((3+v)/(1+v))*(a^2-r^2));
delta_ss_center = (P*a^2/(16*pi*D))*((3+v)/(1+v));

%Moments at r = a for clamped case
Mr_c_edge = -0.0796*P;
Mth_c_edge = -0.0239*P;

%Moments at Strain Gauge Location for Both Cases
Mr_sgc = (P/(4*pi))*((1+v)*log(a/r) - 1 );
Mth_sgc = (P/(4*pi))*((1+v)*log(a/r) - v );
Mr_ss = (P/(4*pi))*((1+v)*log(a/r));
Mth_ss = (P/(4*pi))*((1+v)*log(a/r) + 1 - v );

%Stress and Strain
%Clamped Edges Stresses
sigma_th_c_edge = 6*Mth_c_edge/t^2;
sigma_r_c_edge = 6*Mr_c_edge/t^2;

```



```

%Clamped Edges Strains
eps_th_c_edge = sigma_th_c_edge/E;
eps_r_c_edge = sigma_r_c_edge/E;
eps_r_c_micro_edge = eps_r_c_edge*10e5

%Simple Supported Stresses for Simply Supported at Strain Gauge
Location
sigma_th_ss = 6*Mth_ss/t^2;
sigma_r_ss = 6*Mr_ss/t^2;
%Simple Supported Strains for Simply Supported at Strain Gauge Location
eps_th_ss = sigma_th_ss/E;
eps_r_ss = sigma_r_ss/E;
eps_r_ss_micro = eps_r_ss*10e5;

%Stresses at Strains for Clamped at Strain Gauge Location
sigma = 6*Mr_sgc/t^2;
eps = sigma/E;
eps_micro = eps*10e5;

%Settings for Plotting different radial values
r = linspace(0.01,1.5);
n = length(r);

%Finding Stresses and Strains for different radial values
for i = 1:n
sigma_r_ss_form(i) = ((3*P*z)/(pi*t^3))*(1+v)*log(a/r(i));
eps_r_ss_form(i) = sigma_r_ss_form(i)/E;
eps_r_ss_micro_form(i) = eps_r_ss_form(i)*10e5;
sigma_r_c_form(i) = ((3*P*z)/(pi*t^3))*((1+v)*log(a/r(i))-1);
eps_r_c_form(i) = sigma_r_c_form(i)/E;
eps_r_c_micro_form(i) = eps_r_c_form(i)*10e5;
end

%Plotting
figure(1)
plot(r,eps_r_ss_micro_form)
hold on
plot(r,eps_r_c_micro_form, 'r')
xlabel('Radial Value (in)')
ylabel('Strain (ue)')
legend('Simply Supported','Clamped')
figure(2)
plot(r,sigma_r_ss_form)
hold on
plot(r,sigma_r_c_form, 'r')
xlabel('Radial Value (in)')
ylabel('Stress (psi)')

```

```

I:\School\Thesis\MATLAB\FFT s...\wavfileanalysisIndustryDrivers.m
%Wav file Analysis of Industry Drivers
%Nickolai Volkoff-Shoemaker
%List of Variables
% wave - wave file data
% fs - frequency
% t - time
% n - number of points
% wavefft - fft data

clc
clear

%data collection
[wave1,fs1] = wavread('I:\School\Thesis\Wav
files\PingG5CompletelyIsolated');
[wave2,fs2] = wavread('I:\School\Thesis\Wav
files\LauncherDSTcompleteiso');
[wave3,fs3] = wavread('I:\School\Thesis\Wav files\CobraLDCompleteIso');

%time steps for plotting
t1 = 0:1/fs1:(length(wave1)-1)/fs1;
t2 = 0:1/fs2:(length(wave2)-1)/fs2;
t3 = 0:1/fs3:(length(wave3)-1)/fs3;

%Plotting
figure(1)
subplot(3,1,1)
plot(t1,wave1);
title('(a)');
ylabel('Amplitude');
xlabel('Time (seconds)');

subplot(3,1,2)
plot(t2,wave2, 'b');
title('(b)');
ylabel('Amplitude');
xlabel('Time (seconds)');
subplot(3,1,3)
plot(t3,wave3, 'r');
title('(c)');
ylabel('Amplitude');
xlabel('Time (seconds)');

n1 = length(wave1)-1;
n2 = length(wave2)-1;
n3 = length(wave3)-1;

%frequencies
f1 = 0:fs1/n1:fs1;
f2 = 0:fs2/n2:fs2;
f3 = 0:fs3/n3:fs3;

%FFT analysis
wavefft1 = abs(fft(wave1));
wavefft2 = abs(fft(wave2));

```

```

wavefft3 = abs(fft(wave3));

%Plotting
figure(2);
subplot(3,1,1)
plot(f1,wavefft1);
title('(a)');
xlabel('Frequency in Hz');
ylabel('Magnitude');

subplot(3,1,2)
plot(f2,wavefft2, 'b');
title('(b)');
xlabel('Frequency in Hz');
ylabel('Magnitude');

subplot(3,1,3)
plot(f3,wavefft3, 'r');
title('(c)');
xlabel('Frequency in Hz');
ylabel('Magnitude');
figure(3);

subplot(4,1,1)
plot(f1,wavefft1);
title('(a)');
xlabel('Frequency in Hz');
ylabel('Magnitude');
xlim([20 14000])

subplot(4,1,2)
plot(f2,wavefft2, 'b');
title('(b)');
xlabel('Frequency in Hz');
ylabel('Magnitude');
xlim([20 14000])

subplot(4,1,3)
plot(f3,wavefft3, 'r');
title('(c)');
xlabel('Frequency in Hz');
ylabel('Magnitude');
xlim([20 14000])
ylim([0 100])

subplot(4,1,4)
plot(f3,wavefft3, 'r');
title('(d)');
xlabel('Frequency in Hz');
ylabel('Magnitude');
xlim([20 14000])

figure(4);
plot(f1,wavefft1);
xlabel('Frequency in Hz');
ylabel('Magnitude');

```

```
xlim([20 14000])  
hold on  
plot(f2,wavefft2, 'b');  
plot(f3,wavefft3, 'r');  
legend('Ping G5','LauncherDST' , 'Cobra LD')
```

```

I:\School\Thesis\MATLAB\FFT stuff\wavfileanalysisDrivers.m
%Wav file Analysis of our FEA driver Acoustic Data
%Nickolai Volkoff-Shoemaker
%List of Variables
% wave - wave file data
% fs - sampling frequency
% t - time data for plotting
% n - number of points
% wavefft - fft data

clc
clear

data = xlsread('I:\School\Thesis\MATLAB\FFT
stuff\OVERALLACOUSTICDRIVERRESULTS', 'Pa v t',
'B2:L8193');

%wave and time data from excel

%optimized
time1 = data(4095:8192,3);
wave1 = data(4095:8192,5);

%optimized
time2 = data(:,1);
wave2 = data(:,4);

%not optimized
time3 = data(4095:8192,9);
wave3 = data(4095:8192,11);

%not optimized
time4 = data(:,7);
wave4 = data(:,10);

%max time
T1 = max(time1);
T2 = max(time2);
T3 = max(time3);
T4 = max(time4);

%total amount of time
N1 = length(time1);
N2 = length(time2);
N3 = length(time3);
N4 = length(time4);

%differential frequency
df1 = 1/T1;
df2 = 1/T2;
df3 = 1/T3;
df4 = 1/T4;

%Sampling Frequency
fs1 = N1*df1;
fs2 = N2*df2;

```

```

fs3 = N3*df3;
fs4 = N4*df4;

%number of points
n_1 = length(wave1)-1;
n_2 = length(wave2)-1;
n_3 = length(wave3)-1;
n_4 = length(wave4)-1;

%frequenices for plotitng
f1 = 0:fs1/n_1:fs1;
f2 = 0:fs2/n_2:fs2;
f3 = 0:fs3/n_3:fs3;
f4 = 0:fs4/n_4:fs4;

%FFT data
wavefft1 = abs(fft(wave1));
wavefft2 = abs(fft(wave2));
wavefft3 = abs(fft(wave3));
wavefft4 = abs(fft(wave4));

%Plotting
figure(1)
subplot(4,1,1)
plot(time1,wave1,'r');
title('(a)');
ylabel('Amplitude');
xlabel('Time (seconds)');

subplot(4,1,2)
plot(time2,wave2,'r');
title('(b)');
ylabel('Amplitude');
xlabel('Time (seconds)');

subplot(4,1,3)
plot(time3,wave3,'b');
title('(c)');
ylabel('Amplitude');
xlabel('Time (seconds)');

subplot(4,1,4)
plot(time4,wave4,'b');
title('(d)');
ylabel('Amplitude');
xlabel('Time (seconds)');
figure(2)

subplot(4,1,1)
plot(f1,wavefft1,'r');
title('(a)');
ylabel('Amplitude');
xlabel('Time (seconds)');

subplot(4,1,2)
plot(f2,wavefft2,'r');

```

```
title('(b)');  
ylabel('Amplitude');  
xlabel('Time (seconds)');  
  
subplot(4,1,3)  
plot(f3,wavefft3,'b');  
title('(c)');  
ylabel('Amplitude');  
xlabel('Time (seconds)');  
  
subplot(4,1,4)  
plot(f4,wavefft4,'b');  
title('(d)');  
ylabel('Amplitude');  
xlabel('Time (seconds)');  
  
pause  
sound(wave2,fs2)  
pause  
sound(wave4,fs4)
```