PREDICTING THE ACOUSTIC RESPONSE OF THE GOLF CLUB &
BALL IMPACT USING FINITE ELEMENTS AND THE
BOUNDARY ELEMENT METHOD

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

Scott Moreira

December 8, 2011

© 2011

Scott Moreira

ALL RIGHTS RESERVED

COMMITTEE MEMBERSHIP

TITLE: PREDICTING THE ACOUSTIC RESPONSE OF
THE GOLF CLUB & BALL IMPACT USING
FINITE ELEMENTS AND THE BOUNDARY
ELEMENT METHOD

AUTHOR: Scott Moreira

DATE SUBMITTED: December 8, 2011

COMMITTEE CHAIR: Dr. Tom Mase
Mechanical Engineering Professor

COMMITTEE MEMBER: Dr. Jim Meagher
Mechanical Engineering Professor

COMMITTEE MEMBER: Tom Preece
Vice President of Club R&D – Cobra PUMA Golf

ABSTRACT

PREDICTING THE ACOUSTIC RESPONSE OF THE GOLF CLUB & BALL IMPACT USING SOLID FINITE ELEMENTS AND THE BOUNDARY ELEMENT METHOD

Scott Moreira

An improved and repeatable method for meshing golf club heads using finite elements in TrueGrid® was developed. Using solid brick elements through the thickness of the club head instead of shell elements better represents the many thickness variations throughout each section of a club head. This method also results in a high quality mesh at the center of the club head sections while still maintaining high quality at the edges. A simulation procedure was also developed to predict the acoustic pressure at a designated point in an acoustic medium of a golf club and ball impact using the BEM and Rayleigh methods in LS-DYNA®. The simulation time and computing power required for the impact are modest, while the acoustic simulation time and computing power are much greater. The Rayleigh method provides an alternative which can greatly reduce these requirements. The simulation of sound produced from the ball and a USGA COR plate, generic driver, and hybrid impact was accomplished with reasonable results.

Experimental testing was performed using a USGA plate to validate the plate result. A simple tap test and an air cannon test were performed to record the acoustic response with a microphone. A Fast Fourier Transform was performed to obtain the frequency response. These two tests correlated with each other, indicating that air cannon procedures could be

negated in favor of a much simpler tap test during prototype testing for acoustics. The simulation frequency responses showed similar results to the experimental tests, demonstrating that the procedure developed in this project can be a viable and effective method for determining the acoustic response of the golf club and ball impact.

Table of Contents

List of Tables	ix
List of Figures	x
Nomenclature.....	xiv
1. Introduction	1
1.1 Golf Ball Development	1
1.2 Metal Wood Development	3
1.3 Motivation	7
2. Acoustics Theory	8
2.1 Boundary Element Method	9
2.2 Rayleigh Method	11
3. Computer Simulation Process	13
3.1 CAD Models	14
3.2 TrueGrid®	15
3.3 LS-DYNA®	16
3.3.1 The Boundary Element Method with LS-DYNA®	16
3.3.2 Acoustic Simulation Settings	18
4. Finite Element Meshes	20
4.1 Simplified Generic Driver Mesh	21
4.1.1 Surface Definitions	21
4.1.2 Reference Curves	21
4.1.3 Part Creation & Topology	22
4.1.4 Mesh Techniques & Refinement	24
4.2 Generic Driver Mesh	28

4.2.1	Reference Curves	29
4.2.2	Basic Part Topology	29
4.2.3	The Governing Corner	31
4.2.4	Crown.....	32
4.2.5	Face.....	33
4.2.6	Sole	35
4.2.7	Hosel	36
4.3	Hybrid Mesh	38
4.3.1	Reference Curves	39
4.3.2	Basic Part Topology.....	40
4.3.3	Sole	41
4.3.4	The Governing Corner	46
4.3.5	Face.....	47
4.3.6	Crown.....	48
4.4	Ball Mesh.....	50
4.4.1	Cover.....	50
4.4.2	Core.....	51
4.5	USGA Plate Mesh	52
5.	Acoustic Simulations	54
5.1	TrueGrid® Settings	55
5.1.1	Impact Settings	55
5.1.2	Ball Material Properties	55
5.1.3	Club Head & Plate Material Properties	56
5.2	LS-DYNA® Settings.....	57
5.2.1	Listening Node	57
5.2.2	Boundary Elements.....	57
5.3.3	BEM Keyword.....	58

6. Results	60
6.1 USGA Plate Results	60
6.2 Generic Driver Results	63
6.3 Hybrid Results	65
6.4 BEM & Rayleigh Comparison	68
7. Validation	72
7.1 Tap Testing	72
7.2 Air Cannon Testing	74
7.3 Results	76
8. Conclusions	79
Bibliography	82

List of Tables

Table 1. *BOUNDARY_ELEMENT_METHOD_ACOUSTIC card variables	18
Table 2. Sizes and commands used to create parts.....	23
Table 3. Material properties defining the golf ball Mooney-Rivlin material model	56
Table 4. Material properties of club heads and USGA plate	57
Table 5. Air properties at standard temperature and pressure.....	58
Table 6. *BOUNDARY_ELEMENT_METHOD_ACOUSTIC card variables used	59
Table 7. LS-DYNA® simulation statistics for the USGA COR plate	61
Table 8. LS-DYNA® simulation statistics for the generic driver head	64
Table 9. LS-DYNA® simulations statistics for the fully featured hybrid head.....	66

List of Figures

Figure 1. Modern three-piece golf ball showing the inner core, middle mantle layer, and outer cover (www.golf.com)	2
Figure 2. Metal wood component terminology used throughout the paper	4
Figure 3. Some examples of recent club head designs. (www.specialpricegolf.com & www.golfclubskingdom.com)	6
Figure 4. Visualization of the boundary integral equation	10
Figure 5. Visualization of the half space boundary integral equation	11
Figure 6. Process flow of programs for acoustic computer simulation	14
Figure 7. Process flow of LS-DYNA® acoustic solver	17
Figure 8. Reference curves used to separate and outline each part	22
Figure 9. Basic part topology template used	24
Figure 10. Butterfly mesh technique used for the crown	25
Figure 11. Driver crown illustrating the results of the butterfly meshing technique	26
Figure 12. Transitional block boundary interface of the face and hosel illustrating the use of the trbb command	27
Figure 13. Final mesh the simplified generic driver	28
Figure 14. Reference curves used to govern the band of parts wrapping around the club head	29
Figure 15. Example of a strip of parts wrapped around the entire club	30
Figure 16. The governing corner part showing its three partitions	31

Figure 17. The crown topology showing the three locations of the partitioned butterfly mesh parts.....	33
Figure 18. The face topology showing the two locations of the partitioned butterfly mesh parts.....	34
Figure 19. Butterfly mesh of 6x18x3 part in the face section.....	35
Figure 20. The sole topology showing the three locations of the partitioned butterfly mesh parts.....	36
Figure 21. Hosel portions of the crown (left), face (middle) and sole (right) sections	37
Figure 22. Final mesh of the generic driver	38
Figure 23. Reference curves used to govern the band of parts wrapping around the hybrid club head.....	39
Figure 24. The inner surface and sole section of the outer surface of the hybrid. The large gap represents a thick section	40
Figure 25. Sole inner surface and the curves used to outline extruded section on sole. ...	43
Figure 26. Topology of extruded section on sole of hybrid.....	45
Figure 27. Final hybrid sole mesh indicating nine element thick parts and butterfly meshes	46
Figure 28. The governing corner part showing its three partitions	47
Figure 29. The face topology showing the two locations of the partitioned butterfly mesh parts.....	48
Figure 30. The hybrid crown topology showing the three locations of the partitioned butterfly mesh parts.....	49
Figure 31. Final mesh of the hybrid	49

Figure 32. Partitioned block part used for the cover of the ball.....	50
Figure 33. Partitioned block part used for the core of the ball.....	51
Figure 34. Final mesh of golf ball used in impact simulations	52
Figure 35. USGA COR plate finite element mesh	53
Figure 36. Experimental modal analysis results of the USGA plate.....	61
Figure 37. Acoustic response of USGA COR plate impact simulation using the Boundary Element Method.....	62
Figure 38. Acoustic response of USGA COR plate impact simulation using Rayleigh's Method	63
Figure 39. Acoustic response of the driver impact simulation using the Boundary Element Method.....	64
Figure 40. Acoustic response of the driver impact simulation using the Rayleigh Method	65
Figure 41. Acoustic response of the hybrid impact simulation using the Boundary Element Method.....	67
Figure 42. Acoustic response of the hybrid impact simulation using the Rayleigh Method	68
Figure 43. USGA plate impact simulation using the BEM and Rayleigh methods	69
Figure 44. Driver impact simulation using the BEM and Rayleigh methods	70
Figure 45. Hybrid impact simulation using the BEM and Rayleigh methods	71
Figure 46. Frequency response of USGA plate impact hammer tap test.....	73
Figure 47. Air cannon impact test setup	74
Figure 48. USGA plate free boundary condition for air cannon impact testing	75

Figure 49. Frequency response of USGA plate air cannon impact test.....	76
Figure 50. Combined results of the USGA plate tap and air cannon impact tests	77
Figure 51. Combined results of the USGA plate LS-DYNA® and air cannon impact tests.....	78

Nomenclature

<i>.IGES</i>	Initial Graphics Exchange Specification; solid CAD model file type
<i>A</i>	user defined constant in Mooney-Rivlin material model
<i>B</i>	user defined constant in Mooney-Rivlin material model
<i>bb</i>	TrueGrid® command: block boundary definition and assignment
<i>BEM</i>	Boundary Element Method
<i>block</i>	TrueGrid® command: create a block part
<i>C</i>	constant in Mooney-Rivlin material model
$C(P)$	jump term resulting from treatment of a singular integral with Green's function
<i>c</i>	speed of sound
<i>CAD</i>	Computer-Aided Design
<i>cc</i>	cubic centimeters
<i>COR</i>	Coefficient of Restitution
<i>curs</i>	TrueGrid® command: a 3D curve has component mesh edges attached
<i>D</i>	constant in Mooney-Rivlin material model
<i>dB</i>	decibels
<i>dei</i>	TrueGrid® command: delete regions of the mesh by index progression
<i>delsds</i>	TrueGrid® command: delete a list of surface definitions
<i>FEA</i>	Finite Element Analysis
<i>FFT</i>	Fast Fourier Transform
$G(r, r_y)$	full space Green's function
$G_H(r, r_y)$	half space Green's function
<i>GUI</i>	Graphical User Interface

I_1	principal invariants related to the right Cauchy-Green tensor
i	imaginary number
k	wave number
$lsdyopts$	TrueGrid® command: LS-DYNA® output options
$lsdymats$	TrueGrid® command: LS-DYNA® material models
$mseq$	TrueGrid® command: change the number of nodes
$\vec{n}$	normal vector
n_y	normal vector on vibrating structure's surface
r	field point
r_h	distance between reflected field point and respective point on vibrating structure's surface
r_y	position vector of a source point located on vibrating structure's surface
P	field point in a fluid medium
P_y	field point in a fluid medium reflected over plane S_h
P_{ref}	reference pressure to obtain sound pressure level
p	pressure at a field point
pb	TrueGrid® command: position vertices of a block
$R\&A$	Royal & Ancient Golf Club at St. Andrews
S_y	vibrating structure's surface
S_h	reflection plane
sd	TrueGrid® command: surface definition
sds	TrueGrid® command: combine surfaces into one surface
sfi	TrueGrid® command: project nodes to a surface by index progression
$trbb$	TrueGrid® command: transition block boundary definition and assignment

<i>USGA</i>	United States Golfers Association
ν	poisson's ratio
v_c	club velocity before impact
v'_b	ball velocity after impact
v'_c	club head velocity after impact
v_n	normal structural velocity on vibrating structure's surface
W	strain energy function
w	pulsation frequency

1. Introduction

The first authentic historical reference to the game of golf came on March 6, 1457 when the Parliament of King James II banned the game from Scotland (Grimsley). Although the exact origin of the game is hazy, it is mostly agreed that golf was popularized by the Scottish and has since spread across most of the world. As the game of golf has evolved over the years, even more so has the equipment, providing players with technologically advanced golf balls and clubs that help them hit the ball farther and straighter. While the advancement in technology has been mostly beneficial to the player, some problems have also arisen, particularly the acoustic response of the golf ball and metal wood impact. Players and manufacturers have both objectively identified sounds that are unfavorable of certain clubs and between sweet spot and “mis-hits” on the club face. Due to the complexity of the current designs of golf balls and metal woods there are many factors that influence the acoustics of their impact.

1.1 Golf Ball Development

The first golf ball design consisted of a tightly packed core of goose feathers wrapped into a sphere of horse or cow hide and was called the “Featherie.” Over 200 years later, a new ball called the “Guttie” was introduced and was constructed by heating and shaping the rubber-like sap from the tropical Gutta tree into a sphere. This ball didn’t fly as far as

the Featherie because it was smooth, so its surface was eventually patterned to emulate the scores on the Featherie. At the end of 19th century the first one-piece ball was produced, containing a solid rubber core wrapped with rubber thread within a gutta material sphere. It was also during this time that the surface pattern was experimented with on Gutties and the one-piece balls until dimples were applied to the surface.

Dimples maximize lift while reducing drag for a more aerodynamic ball flight. Although more experimental ball designs continued, the dimpled pattern in combination with the one-piece ball performed best and is the first representation of the modern golf ball design.

Modern golf balls now range from two to four-piece designs typically containing a dimpled core, cover, and possibly a mantle, shown in Figure 1.

Figure 1. Modern three-piece golf ball showing the inner core, middle mantle layer, and outer cover (www.golf.com)

The core material is typically a polybutadiene rubber blend with a complex non-linear material response to achieve maximum performance. The cover is usually an ionomer or polyurethane material and the mantle can be an ionomer or polybutadiene (Nesbitt et al.

1999). In two-piece balls the cover is most likely a hard ionomer to increase distance. Three-piece balls are generally a soft cover (polybutadiene) over a hard mantle (ionomer) to increase the spin without sacrificing much distance. Certain rules and restrictions were put into place to protect the integrity and tradition of the game of golf that dictate the modern golf ball design as well. The United States Golfers Association (USGA) and The Royal & Ancient Golf Club at St. Andrews (R&A) together write the rules of golf and equipment restrictions. The materials in golf ball construction are used mostly due to their performance but also so they do not substantially differ from the traditional and customary form and make, as stated within the Rules of Golf. There is also a maximum weight limit of 1.620 ounces and minimum ball diameter restriction of 1.680 inches (R&A and USGA 2007). Each golf ball design will have a different effect on the acoustic response of the club/ball impact, right down to the depth of the dimples (Tai et al. 2007).

1.2 Metal Wood Development

The first reference to what is now the modern driver came in the 1600s and was called the “longnose.” At the time, all clubs were hand made out of various types of wood, including the shaft and head. With the invention of the Guttie ball the longnoses were replaced with “bulgers” that resembled the current shape of modern woods. Around 1900, club designers began adding metal inserts to the face or sole and grooves on the club face to increase backspin. At the same time, hollow steel shafts were slowly replacing wooden shafts. After World War II golf club development expanded greatly with research into

synthetic and composite materials and the introduction of casting for club manufacturing. This led to graphite shafts and the first metal wood in the 1970s.

The modern metal wood is now a complex piece of technology made of advanced materials and a hollow head consisting of components with intricately designed surfaces, as seen in Figure 2. Typically, metal woods are made of titanium and almost all have composite graphite shafts attached. During production the club is casted and a titanium face insert is welded into the rest of the body. Titanium is used in metal woods for its low weight and high strength material properties, allowing club designers to create larger club heads with thinner walls that remain the same weight as their older counterparts.

Figure 2. Metal wood component terminology used throughout the paper

Due to these great technological advancements, the USGA has placed restrictions on golf club design in addition to golf ball design to continue to protect the integrity of the game.

The first major restriction is on club head size. The size of driver heads has increased greatly over recent years with the popularity and advancement of metal wood development. Larger heads will increase the “sweet spot” on the club face and increase forgiveness for off center hits. The USGA decided on a volume limit of 460 cubic centimeters (cc) for club heads to prevent overly large club head designs (R&A and USGA 2007).

Another major restriction involves the interaction between the golf ball and club face on impact. Thinner walls on the face of the club head allow for more deformation, increasing the efficiency of energy transfer during impact and creating a spring-like effect. This measurement of elasticity is called the Coefficient of Restitution (COR) and its equation relating to a golf ball and club impact is shown below.

$$COR = \frac{v'_b - v'_c}{v_c} \quad (1)$$

The COR is basically the ratio of ball velocity after impact (v'_b) to ball velocity before impact. In the case of a golf swing the ball velocity before impact is zero. The USGA decided to limit the COR of club heads to a maximum value of 0.830 (R&A and USGA 2007).

Club manufacturers continue to produce new metal woods that attempt to make the golf ball fly farther, faster, and straighter, but are now forced to design them within the restraints set forth by the USGA. The basic design goals are to produce a lighter and more aerodynamic club with state-of-the-art materials that achieves the aforementioned ball performance while feeling and sounding good to the player. The vast array of club designs released to the market in recent years indicates manufacturers are experimenting

with the geometry of the club head, with designs ranging from a triangular head (TaylorMade Burner SuperFast) to a more square head (Nike SQ Sumo2) as shown in Figure 3. Some manufacturers have neglected certain design goals to focus solely on performance and have released clubs that were scrutinized for their poor acoustic response. Of course the sound a club makes is a completely objective characteristic that will vary between players, yet still an important one to the consumer. When paying top dollar for a state-of-the-art club, many consumers seek positive feedback from their shots in the form of the club's sound and may not be willing to pay the money for a club with an unappealing acoustic response.

Figure 3. Some examples of recent club head designs. (www.specialpricegolf.com & www.golfclubskingdom.com)

1.3 Motivation

There are currently a couple methods for testing club heads during the design phase for their acoustic performance, including field testing, direct testing and computer modeling (Sharpe 2009). With field testing, a company will release a new club design to the market and rely on player testimony to get feedback on its performance. This will usually come in the form of touring professionals, as they are the most influential, but could be consumer based as well. This is the most risky and least cost effective test procedure, as any poor testimony could result in low sales for this product and require a complete redesign for the next year. Direct testing involves producing a prototype of a new design and testing it with a swing robot. The acoustic response can be captured with a microphone and optimized for later revisions during the design phase. This method could require multiple prototypes and design revisions.

Computer modeling and simulation may be the most efficient and cost effective method for acoustic testing. Typically this can be done using Finite Element Analysis (FEA) in conjunction with a Boundary Element Method (BEM) to numerically solve the acoustic analysis. These simulations could be run at any point in the design phase and possibly negate the need for acoustical prototype testing.

The goal of this project is to improve the method for modeling golf clubs using solid brick elements instead of shell elements through the club's thickness in FEA and develop a simulation procedure using the BEM to accurately predict the acoustic response of the golf ball and club impact.

2. Acoustics Theory

The BEM can be used to analyze the radiation of sound emitted by a vibrating object in an unbounded acoustic medium. In the case of the golf ball/club impact, the club is energized upon impacting the golf ball, which will cause it to vibrate. These vibrations cause fluctuations in the air pressure surrounding the boundary of the club head and radiate into the acoustic medium, which in this case is the surrounding air. These pressure changes are what the human ear interprets as sound, and it will vary in frequency and intensity depending on the location and distance from the vibrating structure. This variance can be attributed to the principle of superposition. Acoustic phenomena are typically of small amplitude, which can be approximated well using linear algebra and linear differential equations. Therefore the principle of superposition can be applied to linear acoustics. Waves of each frequency will propagate independently, allowing them to be analyzed separately and their results combined. This also relates to a complicated sound field composed of several sources, where the overall sound is regarded as the sum of each field from the individual sources (Rossing 2007). This can be applied to a golf club head, as the different parts composing the club head (face, crown, sole etc.) each have their own acoustic field (Sharpe 2009).

The Hemholtz equation governs the acoustic wave propagation in an ideal fluid medium and is defined as follows:

$$\Delta p + k^2 p = 0 \tag{2}$$

p is the pressure at any field point and k is the wave number defined as:

$$k = \frac{w}{c} \quad (3)$$

w is the pulsation frequency ($w = 2\pi f$) and c is the speed of sound in the fluid medium.

2.1 Boundary Element Method

In many cases, integral equations are necessary to determine the acoustic field rather than partial differential equations. In this case, Green's theorem is necessary to transform Eq. 2 into an integral equation so the BEM can be applied. Now the pressure at any field point P in the fluid medium can be expressed as an integral of pressure and velocity over a surface shown below (Huang and Souli):

$$C(P)p(P) = \int_{S_y} [G(r, r_y) \frac{\partial p(r)}{\partial n_y} - p \frac{\partial G(r, r_y)}{\partial n_y}] dS_y \quad (4)$$

The full space Green's function is defined below:

$$G(r, r_y) = \frac{e^{-ik|r-r_y|}}{4\pi|r-r_y|} \quad (5)$$

Figure 4 illustrates the boundary integral equation (Eq. 4), where n_y is the normal vector on the vibrating structure's surface S_y , $p(r)$ is the pressure at a field point r , and r_y is the position vector of a source point located on S_y . $C(P)$ is the jump term resulting from the treatment of the singular integral involving Green's function (Alia and Souli 2005) and is

equal to one for any point in the acoustic domain and $\frac{1}{2}$ for a point on a smooth boundary (Herrin et al. 2003).

Figure 4. Visualization of the boundary integral equation

The pressure is related to the structural velocity by:

$$\frac{\partial p}{\partial n} = -i\omega p v_n \quad (6)$$

At this point the BEM can be applied to find the pressure at any point for the golf ball/club impact. The main advantage of the BEM is that only the vibrating acoustic surface needs to be meshed and not the entire acoustic medium as with typical finite element methods, because only the surface pressure and velocity is needed. This is a very computationally intensive process that requires iteration to solve the system at every frequency, so the simulation time can be extremely long (Huang and Souli 2008). Using the BEM with LS-DYNA® will be described in further detail in a later section.

2.2 Rayleigh Method

An alternative, time-saving method to solving an acoustic problem such as this for the pressure at a field point is Rayleigh's method. Using a Rayleigh integral approximation is widely dismissed as an unreliable method for predicting sound pressure, despite some recent research suggesting otherwise (Herrin et al. 2003). The Rayleigh method works under the assumption that each element vibrates independently and is assembled to a plane surface mounted in an infinite rigid plane, shown in Figure 5 (Huang and Souli 2008).

Figure 5. Visualization of the half space boundary integral equation

P_y is the mirror image of the field point P reflected over plane S_h . r_h is the distance between the reflected field point and the respective point on the vibrating structure's surface. The half space formulation for obtaining the Rayleigh integral equation, as described by Herrin, begins with the half space Green's function:

$$G_H(r, r_y) = \frac{e^{-ikr}}{4\pi r} + \frac{e^{-ikr_h}}{4\pi r_h} \quad (7)$$

If the vibrating surface lies on the half space plane, Green's function reduces to:

$$G_H(r) = 2G(r) \quad (8)$$

and the partial derivative with respect to the normal reduces to:

$$\frac{\partial G_H(r)}{\partial n_y} = 0 \quad (9)$$

By applying this half space assumption and Eqs. 7-9, Eq. 4 reduces to the Rayleigh integral equation:

$$C(P)p(P) = \int_{S_y} 2G(r) \frac{\partial p(r)}{\partial n_y} dS_y \quad (10)$$

Now the sound pressure at any field point can be obtained from the surface velocity only.

This does not require any matrices to be formed or systems of equations to be solved.

Therefore, the Rayleigh method requires significantly less computer resources than the BEM which results in a shortened simulation time.

Most real world problems are not of this type, but the Rayleigh integral can provide a good approximation for a relatively flat surface as long as the higher amplitude vibration results from the central portion of the surface (Herrin et al. 2003). Due to the construction of metal woods having thin surfaces that get thicker at the edges it is reasonable to use the Rayleigh method as a faster, preliminary analysis. Therefore, both the Rayleigh method and BEM were used for comparison during this project.

3. Computer Simulation Process

The first step in the simulation process is to virtually construct the club head. This begins by creating a 3D Computer-Assisted Design (CAD) model of the club head, typically of just the outer and inner surfaces rather than a solid model. This CAD model is then imported into a mesh generator or finite element software program, such as TrueGrid®, to build a mesh of the club head. A CAD model of the ball is not necessarily required and can instead be created and meshed within TrueGrid®. These meshes are brought into a pre-processor where the impact and acoustic analysis parameters are defined. The simulation is then run using a dynamic finite element analysis solver. Finally, the impact and acoustic results are analyzed using a post-processor. LS-DYNA® provides a pre- and post-processor along with its FEA software package for running and analyzing these types of dynamic problems. Figure 6 outlines the process flow for the club head/ball impact and acoustic simulation.

Figure 6. Process flow of programs for acoustic computer simulation

3.1 CAD Models

The two CAD models of the clubs used in this project were designed using the NX CAD program from Siemens PLX Software and contained only the inner and outer surfaces of the club heads instead of a solid model. The models provided were in the Initial Graphics Exchange Specification (.IGES) format. This is a universal CAD file type with a small file size compared to the native file types used in CAD software. Its small file size and nature of its format allow it to be easily exchanged between CAD systems, making it the ideal file type to import into the mesh generator, TrueGrid®.

3.2 TrueGrid®

TrueGrid® is a mesh generator and pre-processor for FEA, licensed by XYZ Scientific Inc. This program was the ideal choice for meshing the golf club heads and ball because of its ability to create quality meshes and output them to the LS-DYNA® keyword format used for the acoustic impact simulation. The user has complete control over the entire mesh through a graphical user interface (GUI) or a text file, while typically only using a few, simple commands to actually create the structure. Its projection method also allows the creation of meshes on non-standard geometry imported from a CAD part file, such as the .IGES files of the club heads. Once the part file is imported, a block mesh is created in space and partitioned to define the number of elements in each of the x-, y- or z-directions. If necessary, portions of the mesh can even be removed to match its topology to the structure being meshed. From here, a number of techniques can be used to map the mesh to the structure, including moving pieces of the mesh, positioning edges along curves, and projecting it to the structure's surface. Multiple parts can be attached together at their interfaces and common nodes merged by specifying a tolerance to complete the object's overall mesh. The specific techniques and commands used to mesh the club heads and ball are described in further detail in a later section 4. Upon mesh completion, TrueGrid® can output the mesh to a LS-DYNA® keyword file needed to create the impact and acoustic analysis within LS-PrePost®.

3.3 LS-DYNA®

LS-DYNA® is a transient dynamic finite element solver licensed by Livermore Software Technology Corporation (LSTC), capable of simulating very complex real world engineering problems. This explicit/implicit solver was primarily designed to analyze varieties of impact loading and is used in automotive, aerospace, military, and bio-engineering industries. The keyword format of the software provides a simple, organized database of cards used to activate each of the functions provided. An extensive material database is also built in that is especially beneficial for modeling non-linear materials, such as golf balls. LS-DYNA® comes packaged with LS-PrePost®, a pre- and post-processor, with a GUI that allows the user to activate all the keywords required for the simulation and analyze the results upon completion. This is beneficial compared to other FEA software as it gives the user easy access to the inner workings of the program to fully customize the analysis.

3.3.1 The Boundary Element Method with LS-DYNA®

Recently, a new card was developed to solve acoustic problems based on the BEM called *BOUNDARY_ELEMENT_METHOD_ACOUSTIC. This allows for a vibro-acoustic simulation without discretizing the entire acoustic medium, as long as the vibrating structure isn't affected by the acoustic propagating waves (Huang and Souli 2008).

The transient structural response of the impact is first computed to obtain the surface velocity data necessary for the BEM analysis and stored in a binary file. The transient solution is performed in the temporal domain, but the BEM requires the velocity data in the frequency domain to create the boundary conditions. This is accomplished by

performing a Fast Fourier Transform (FFT) on the velocity data to transform it into the frequency domain. The surface pressures are also required for the analysis and are obtained by using the collocation method. The BEM is used to discretize the integral equation (Eq. 4) and apply it at each node on the boundary. This creates a set of equations at each frequency that needs to be solved to obtain the pressure data. This requires an iterative process to solve the system at each frequency, which is the main contributor to the length of the simulation. Once the velocity and pressure data is obtained in the frequency domain the BEM can be applied to obtain the acoustic pressure at any field. Figure 7 below summarizes the *BOUNDARY_ELEMENT_METHOD_ACOUSTIC solver process flow.

Figure 7. Process flow of LS-DYNA® acoustic solver

3.3.2 Acoustic Simulation Settings

The *BOUNDARY_ELEMENT_METHOD_ACOUSTIC card can be accessed and defined using LS-PrePost®. Table 1 below shows the parameters for this card that need definition, where *F* indicates a floating point number is required and *I* indicates an integer. The default values are shown as well.

Table 1. *BOUNDARY_ELEMENT_METHOD_ACOUSTIC card variables

Card 1	1	2	3	4	5	6	7	8
Variable	Dens	Sp-sound	Min_Freq	Max_Freq	NFreq	Dt-Output	Start_Time	Ref_Pres
Type	F	F	F	F	I	I	F	F
Default	None	None	None	None	0	0	0	0

Card 2	1	2	3	4	5	6	7	8
Variable	Ext_Pres	Type_Ext	Int_Pres	Type_Int	FFT_Wind			
Type	I	I	I	I	I			
Default	0	0	0	0	0			

Card 3	1	2	3	4	5	6	7	8
Variable	IBEM_Met	Max_Iter	Res	NDD				
Type	I	I	F	I				
Default	0	100	10 ⁻⁶	0				

Card 4	1	2	3	4	5	6	7	8
Variable	SSID	SSTYPE	Norm	BEM_Type	Restart			
Type	I	I	I	I	I			
Default	0	0	0	0	0			

The fluid density and speed of sound of the acoustic medium are input by the user. A range of output frequencies is required, as well as the total number of frequencies for which the acoustic pressure will be calculated. Time parameters can also be defined, including the interval between which the velocity and pressure data is written to the binary file, and the time to begin recording this data. If interested, the user can input a

reference pressure to obtain the pressure at the field points in decibels (dB), but is not required otherwise.

The boundary of the vibrating structure and the field points of interest can be defined from a keyword input file in LS-PrePost®. The boundary is specified by *SET_SEGMENT, *SET_PART, OR *PART and can be selected using the GUI. In a similar fashion, the field points can be defined using *SET_SEGMENT or *SET_NODE. The boundary and field point type and definition need to be specified in the cards once created.

The options relating to the type of BEM used also need to be chosen. This keyword provides three different methods to solve the problem. The exact BEM can be used, or the faster Rayleigh and Kirchhoff methods can be implemented. Each of these methods requires the use of an FFT, and the windowing type needs to be specified. If the exact BEM is used, several other parameters can be defined, including the maximum number of iterations for the iterative solver and its residual value. Memory can also be saved by using a number of domain decompositions.

4. Finite Element Meshes

The first portion of this project consisted of learning techniques and developing a repeatable process to efficiently mesh driver heads. The biggest challenge and goal of this method is to use solid brick elements through the thickness of the club head in order to later perform an acoustic analysis on the mesh. Typically, for thin-walled structures, shell elements are used when creating a finite element mesh. Shell elements are two dimensional and do not physically have a thickness in the third direction, but are instead assigned a thickness value. Although metal woods are thin-walled, the thickness varies greatly over the entire club head. To obtain a model with an accurate thickness, meshing using shell elements is not advantageous, and solid brick elements were used instead. Three finite elements meshes were created using the TrueGrid® mesh generator: two using a simplified, generic model of a driver, and the last using a model of a hybrid. The purpose of the first model was mainly to learn how to use the TrueGrid® program to develop various techniques and processes for future models. The second model was more in depth with a higher mesh density to obtain a basic part topology of the club and a method for meshing it. The third model was a refined mesh that included all the artwork and intricacies of the hybrid club which were absent in the first two driver models. A golf ball and a titanium plate used for validation simulations were also meshed.

4.1 Simplified Generic Driver Mesh

The first model was not an exact replica of the driver head. All the artwork, grooves and other intricacies of the head were removed, leaving a basic club head with smooth surfaces on all sides. The .IGES file was imported into TrueGrid® and a binary file of the model was then created. A binary file significantly reduces the processing time when importing the solid model every time the mesh is opened for editing.

4.1.1 Surface Definitions

TrueGrid® imports .IGES files as a set of surfaces that make up the model rather than solid parts. Certain CAD programs export extremely large planar surfaces to the .IGES file that define the extreme limit of the entire model, as was the case with this club model. These large planar surfaces were removed from the model using the `delsds` command, leaving just the driver head. The club was a set of surfaces that needed to be separated into an inner and outer surface. This was done by listing and selecting all the visible surface definitions and combining them using an `sd 1000 sds` command to make a single surface definition for the outer surface. This surface was then removed from view, leaving just the inner surface definitions. The process was repeated with the remaining surfaces to create a single surface definition for the inner surface. These definitions were crucial later on during the meshing process for projection and viewing purposes.

4.1.2 Reference Curves

For this preliminary model, the club was separated into ten parts: face, crown, sole, toe skirt, back skirt, heel skirt, and four parts that comprise the hosel. In order to map each part appropriately over the surfaces of the club, reference curves were created to act as

boundaries for the edges of each part to attach to. These curves also helped interface each part with its surrounding parts. Twenty-four curves were created separating and outlining each part as shown in Figure 8.

Figure 8. Reference curves used to separate and outline each part

These curves were created using the spline tool from the 3D curves menu. The spline tool requires a list of points input from the user to comprise the curve. The Z-Buffer option was chosen from the pick menu and points were hand selected by following the edges of the imported surfaces, with the exception of the hosel. The points running down the hosel were selected so the curves divided the hosel into four equal sections. The curves on the outer surface were defined first and the process was repeated for the inner surface.

4.1.3 Part Creation & Topology

A 3x1x1 partitioned block part was first made to represent the face of the club head using a block 1 11 21 31; 1 16; 1 4; command. This indicates the part consists of 30 elements in

the x-direction, with 10 per partition, 15 elements in the y-direction, and three elements through the thickness in the z-direction. To map the part onto the surface of the club face, the eight corner nodes of the part were moved to the corresponding eight corners of the face on the inner and outer surfaces created by the reference curves. The pb command was used and the eight intersection points on the reference curves were selected using the label option from the pick menu. From there, the eight edges of the part were attached to the appropriate reference curves outlining the face using the curs command. Lastly, the appropriate two surfaces on the part were projected to the inner and outer surfaces of the club face using the sfi command and the surface definitions defined earlier. In each of the three steps necessary to map the part onto the surfaces the computational window GUI was used to select the required nodes, edges or surfaces of the part. Now that the first part was fully defined and mapped onto the surfaces of the club, the boundary interfaces needed to be defined using the bb command. All four boundaries of the face served as master boundaries since it was the first part created. These master boundaries will control how each of the surrounding parts will interface and attach to this part. This process was repeated correspondingly for the other nine parts using the block commands in Table 2.

Table 2. Sizes and commands used to create parts

Part	Block Command
Face	block 1 11 21 31; 1 16; 1 4;
Crown	block 1 11 21 31; 1 4; 1 10 20 30;
Sole	block 1 11 21 31; 1 4; 1 10 20 30;
Sole Skirt	block 1 4; 1 16; 1 10 20;
Back Skirt	block 1 11; 1 16; 1 4;
Heel Skirt	block 1 4; 1 16; 1 11;
Crown Hosel	block 1 4; 1 8; 1 4;
Face Hosel	block 1 8; 1 7; 1 4;
Sole Hosel	block 1 4; 1 8; 1 10;
Heel Skirt Hosel	block 1 16; 1 8; 1 4;

Figure 9 shows the basic part topology layout used for this model. This view shown represents the club head being laid flat without the sole. Despite not being shown in this view, the sole is just beneath the crown and has the exact same layout as the crown. The toe skirt is at the top of this view, the back skirt is at the right, and the heel skirt is at the bottom.

Figure 9. Basic part topology template used

4.1.4 Mesh Techniques & Refinement

The sole and crown parts required a modified butterfly mesh technique to achieve an appropriate topology. This technique is used when projecting a block element to a curved surface. The elements near the corners of a block element projected to a curved surface do not have the proper angles for a quality mesh. The outer two surfaces of the corner

element have to be folded to an angle close to 180 degrees in order to conform to the contour, and this problem is magnified as the mesh is refined. To prevent this, a block part is partitioned and the corner partitions are deleted using the dei command wherever they would be projected to a curved surface. The resulting interior surfaces exposed are then stitched together using a bb command. An example of this technique used for the crown is shown in Figure 10. Instead of one block part, the part was partitioned into a 3x3 array and the top two corner partitions were deleted. The top left corner has been stitched together and will project to the curved surface appropriately.

Figure 10. Butterfly mesh technique used for the crown

Figure 11 shows the finished crown and the results of this technique. The angles of the elements in the corner region remain close to 90 degrees and preserve mesh quality. The butterfly mesh technique was also used on the sole in similar locations as the crown.

Figure 11. Driver crown illustrating the results of the butterfly meshing technique

Another technique used in creating this mesh is the transition block boundary (trbb). In order to use the standard block boundary command (bb), the number of elements of two adjacent parts must be equal at the interface. The transition block boundary is assigned when the number of elements on two adjacent parts differs at their interface by a factor of three. This block boundary converts the elements of the slave boundary to match the number of elements of the master boundary. This trbb command is most often used when

adjacent parts are progressively getting smaller and fewer elements are required, or vice versa. In this model, the trbb command was used to interface each of the four hosel parts to their adjacent counterparts. An example of the face and hosel interface is shown below in Figure 12.

Figure 12. Transitional block boundary interface of the face and hosel illustrating the use of the trbb command

The hosel width is much narrower than the face, so fewer elements were used. Six elements were used on the hosel compared to the 18 on the face at this interface, thus complying with the trbb requirements.

The mesh was then further refined by using the pb command to move non-corner nodes of the partitioned parts to different locations in order to get the most even mesh possible.

The goal is to keep the elements at a consistent size and each of their four angles at 90 degrees. The final result of this model is shown in Figure 13.

Figure 13. Final mesh the simplified generic driver

4.2 Generic Driver Mesh

The second club head mesh was used to develop a basic topology using smaller elements and parts in preparation for the final mesh. This will be a more representative process of meshing a fully featured club head. It was also used to run preliminary acoustic tests.

This mesh was modeled using the generic driver head with just the smooth surfaces as used in the first model. The .IGES file was imported once again using a binary file, and the inner and outer surface definitions were created using the process described in section 4.1.1.

4.2.1 Reference Curves

Since this part is comprised of 118 smaller parts, compared to the ten used in the first model, the club head was divided into three sections: the crown, face, and sole. Each of these sections was divided by a narrow band of parts that ran along the edges of the crown and face and continued up to the top of the hosel. This band serves as a reference point and interface for the small parts in each section. Twelve reference curves were created using the same method in the previous model to govern this reference band, shown in Figure 14.

Figure 14. Reference curves used to govern the band of parts wrapping around the club head

4.2.2 Basic Part Topology

There were two basic methods for the part topology in this model. One method was to create parts along the edges that follow the curvature of the club head. This would lead to

quality meshed parts along the edges of the club head, but cause problems in the interior of the sections where the parts would eventually interface. This method is not ideal for this mesh. The other method is to wrap the club using straight strips of parts in each direction, as shown in Figure 15. This creates a grid of consistent square parts in the interior while projecting the parts along the edges to the edge curvature. Now there will be mesh quality issues along the edges, but they can be controlled with the butterfly meshing technique. The higher quality mesh will be located in the interior of the sections, which serves best for this mesh.

Figure 15. Example of a strip of parts wrapped around the entire club

Once again, solid brick elements were used with three elements through the thickness of the club. The majority of parts used were 18x18x3 blocks for consistency in covering the three sections evenly based on the size and coordinates of the club head. They also

allowed for the eventual use of the transition block boundary technique with six and two elements at the interfaces. All the parts were projected to the club surfaces using the same method described in section 4.1.3, with the narrow band of parts along the edges containing all the master block boundaries.

4.2.3 The Governing Corner

The model began by creating the part in the band at the corner where the crown, face and sole sections meet, as this part governed most of the topology of the entire mesh. This part defines the width of the band along the crown, the width of the band along the lower edge of the face, and dictates the starting part sizes for each main section. A partitioned part was created using a block 1 10 13 19; 1 7; 1 4; command and is outlined in black in Figure 16.

Figure 16. The governing corner part showing its three partitions

The part is 18 elements long to interface with the partitioned 18x18x3 part above it on the crown and six wide to define the crown band width. The middle partition is three wide to define the lower face edge band width. Other features resulting from this part will be referenced in the following sections.

4.2.4 Crown

The crown of the club head was the first section to be meshed. The basic method for creating the topology was to create parts at the governing corner and move away towards the opposite corner. The first part made was an 18x18x3 part, as seen in Figure 16. This part was critical because it interfaced with the governing corner and began the master block boundaries for the rest of the 18x18x3 parts on the crown. A 2x2 partition butterfly mesh was required to interface with the governing corner part. It was important when making the governing corner part to properly define the master block boundaries to interface to this part. The two partitions of the governing corner that were three and six elements long needed to be combined into one master block boundary to interface appropriately with one of the nine element long butterfly partitions of this part.

Once this part was completed the rest of the crown was mapped out moving towards the opposite corner, as mentioned prior, always beginning with the thin parts in the band. Each part in the band was 18x6x3 and contained all the master block boundaries, while the interior parts covering the crown were all 18x18x3. Two more butterfly mesh parts identical to the first one were necessary at the corners of the crown in similar locations as the first model, shown in Figure 17. The parts in the band interfacing with these two butterfly mesh parts needed to have two 9x6x3 partitions in order to define the two

necessary master block boundaries. The pb command was used to locate nodes in order to achieve a mesh with consistent and even elements.

Figure 17. The crown topology showing the three locations of the partitioned butterfly mesh parts

4.2.5 Face

The face section was meshed following the crown. The method was similar to the crown, as the mesh began at the governing corner and moved towards the opposite corner of the face. The band parts along the lower edge of the face were all $18 \times 3 \times 3$ to interface properly with the middle partition of the governing corner, while the majority of the

interior parts remained at $18 \times 18 \times 3$. Once again all the band parts contained the master block boundaries.

Two butterfly meshes were required, as shown in Figure 18, each with different partition sizes. The lower left corner is the same 2×2 partition used for each of the butterfly meshes used in the crown section.

Figure 18. The face topology showing the two locations of the partitioned butterfly mesh parts

The other is part of a smaller interior strip of two parts that are $6 \times 18 \times 3$ in size. This strip of parts is a result of the six element wide partition in the governing corner part. The butterfly mesh is separated into two $3 \times 15 \times 3$ and two $3 \times 3 \times 3$ partitions. One of the $3 \times 3 \times 3$ partitions was deleted and gives decent mesh quality at a sharp corner, shown in Figure

19. As before, the adjacent thin band part was partitioned to interface correctly to this part. The pb command was used to locate nodes as needed to achieve a mesh with consistent and even elements.

Figure 19. Butterfly mesh of 6x18x3 part in the face section

4.2.6 Sole

The sole was the most straightforward section to mesh because all of the parts in the bands were previously created and, therefore, all the master block boundaries were defined before starting this section. Like each section prior, the parts were made starting at the governing corner and worked out towards the opposite corner. The first part was 18x9x3 to interface with the nine element wide partition in the governing corner part, and the entire strip of parts running along the lower face edge were this size. Otherwise, the rest of the interior parts were each 18x18x3. Three butterfly meshes were used, as shown

in Figure 20, each having the standard 2x2 partitions used in the previous sections. The pb command was used to locate nodes as needed to achieve a mesh with consistent and even elements.

Figure 20. The sole topology showing the three locations of the partitioned butterfly mesh parts

4.2.7 Hosel

The hosel portions of each section were all meshed last. The hosel contained two parts around its diameter: one 24 elements long that ran the length of the hosel and another nine elements long that served as the interface between the hosel and each section or band. The width of each of these parts depended on the interface. If the parts connected a

band section, the width was the same number of elements as the band. If the parts connected interior parts of a section, their width was six elements and a transitional block boundary (trbb) was used at the interface. For every case, the nine element long parts contained the master block boundaries governing the 24 element long parts. Each hosel section is shown below in Figure 21.

Figure 21. Hosel portions of the crown (left), face (middle) and sole (right) sections

The face contains one extra part which the other two sections do not have as seen in the middle picture of Figure 21. This is a $9 \times 6 \times 3$ part that has two $6 \times 3 \times 3$ and two $3 \times 3 \times 3$ partitions used for a butterfly mesh. One of the $3 \times 3 \times 3$ partitions was deleted to keep a decent mesh density at the corner. The six element long interface required a transitional block boundary (trbb) to interface to the 18 element long part beside it. A smaller number of elements were chosen for this part to prevent the elements close to the corner from becoming much smaller in comparison to the rest of the mesh. The final overall mesh of this model is shown in Figure 22.

Figure 22. Final mesh of the generic driver

4.3 Hybrid Mesh

The final club meshed was a fully-featured hybrid. The techniques learned from the previous two models were applied in order to verify that the meshing process will work when applied to a production club. This club contains various intricacies such as major variations in thickness, a protruding feature on the sole, and artwork that were not found on the smooth driver head used for the previous two models. These features provided a much larger challenge to create a quality mesh of the club head. This mesh was also used for production runs of acoustic simulations. Just as in the previous two models, the file was imported using a binary file, and inner and outer surface definitions were created using the process described in section 4.1.1.

4.3.1 Reference Curves

Just as in previous models, reference curves were created to act as boundaries for the face, sole, and crown. These curves outlined the narrow band used to interface the three sections together. The curvature of the club head at the edges requires the curves on the outer surface to create a slightly wider band than the curves on the inner surface. This will prevent the interfacing parts from wrapping around an extreme curvature, which can lead to large and improper interior angles of the elements. Ten curves were defined, as shown in Figure 23. Unlike the previous models, these curves do not extend up the hosel. The reason for this will be explained in the next section.

Figure 23. Reference curves used to govern the band of parts wrapping around the hybrid club head

4.3.2 Basic Part Topology

Some of the basic methodology used to mesh the generic driver was applied for this model, but due to the intricacies of the club surface, a more improvisational and fluid meshing process was used. In general, it is advantageous to mesh the most complicated sections of a part first, as there may only be one or two ways to mesh these sections. As a result, these sections will determine the size and locations of the surrounding elements which will propagate over the entire part as well. An extrusion containing artwork was the most difficult section to mesh and was meshed first. Towards the rear of the club head there is also a significantly thicker section than the rest of the club, shown in Figure 24. More elements through the thickness were required in this section, and meshing the transitioning parts around its edges was also a challenge.

Figure 24. The inner surface and sole section of the outer surface of the hybrid. The large gap represents a thick section

Therefore, the sole was meshed first, followed by the face and crown as they posed no significant meshing challenges. The hosel was not meshed in this model. The inner surface at the hosel section is complicated and leads to very poor mesh quality when projected to it.

The same method used for the generic driver of creating strips of parts over the whole club head governed the topology in this model. Once again, solid brick elements were used with three elements through the thickness, except for the thicker section mentioned earlier, where nine elements were used. The initial goal to use mostly 18x18x3 block parts did not work for this model. Instead, parts using nine, 15 and 18 elements were used when possible to make use of the transition block boundary technique with three, five and six elements, respectively. The parts in the narrow band contained the master block boundaries when possible, but this was not always the case. The pb command was again used liberally during all meshing phases to fine tune the nodal locations of the parts.

4.3.3 Sole

The mesh development began with the artwork on the sole, as it is the single most complicated section of the club. With the chosen element size, projecting a part to the surface large enough to cover the artwork created many deformed elements. Manually placing each node to properly mesh the artwork would have taken many hours, required much smaller elements, and created problems with the surrounding elements. It was decided for the purpose of this model that ignoring the artwork would not significantly affect the acoustic response. A part was created that covered the artwork and curves were defined outlining the section on the outer surface. The part was attached to these curves but not projected to the top surface to avoid the artwork. TrueGrid® will automatically

adjust the non-projected surface of a part to the average height of the curves its edges are attached to. By using the curves on the edges and another across the middle of the part, the part is essentially projected to the surface without using the sfi command, eliminating the artwork.

The artwork was part of the v-shaped extruded section on the sole, which was dealt with next. Since this section protrudes directly up from the surface, it would not interface with any other parts on its sides. Therefore, it was treated as its own section of parts with three elements through its thickness. It did have to interface with the rest of the sole beneath it, so parts that exactly mirrored the shape of the extrusion were meshed directly beneath it. There is no surface at their interface to project the parts to, so using the same method to eliminate the artwork mentioned above; curves were used to define the edges of the section. The parts were attached to these curves, creating an average surface height at their interface. The curves used to define this section are shown in Figure 25.

Figure 25. Sole inner surface and the curves used to outline extruded section on sole.

The parts beneath the extruded section needed to be meshed first, since they would interface with other parts surrounding them and thus, contain the master block boundaries. These parts posed a significant meshing challenge because they coincided with the section where the thickness transitions from three to nine elements. Figure 25 shows the inner sole surface and the large u-shaped section where this thickness increases. The highlighted curve 18 indicates where that transition begins, and it takes up the majority of the sole towards the rear as shown. Projecting parts to the inner surface around curve 18 resulted in highly deformed elements because the transition occurs quickly at an angle of almost 90 degrees. Small sections of the parts around curve 18 were partitioned and not projected to the inner surface to smooth the transition and improve the element quality under the assumption that these small partitions would not

greatly affect the overall acoustic response of the model. Once again the “surface averaging” technique mentioned earlier in the sections was used instead of projection.

A block 1 3 10 17 24 26; 1 3 19 21; 1 10; command was used to create the large part beneath the artwork to govern the extruded section. It was partitioned so the “tongs” of the extruded section were nine elements wide and the section in between was seven elements wide. Two butterfly meshes were used at the round edges of this part to improve mesh quality. This resulted in an overall width of 21 elements at the back edge. A trbb command was used at the seven elements wide center section to transition the parts in the middle of the tongs to 21 elements. This kept that strip of parts at the same width to go around the rest of the head. The two tong sections needed parts with matching widths at the rear of the club to create a strip to go around the club head. Nine element wide parts were created alongside the first part and a trbb command was used to reduce them to three elements wide as they approached the ends of the tongs. A butterfly mesh was performed on the last parts to reduce them to one element wide and essentially end that strip of parts at one node. Now these nine wide strips would match up at either end of the sole to provide continuity around the entire head. The extruded section of parts was then meshed above their mirrored parts below with three elements through the thickness. The parts at the ends of the tongs contained a butterfly mesh, reducing the thickness to one element to smooth out the transition to the lower surface. This section of parts on the sole is shown in Figure 26.

Figure 26. Topology of extruded section on sole of hybrid

After this complicated section was completed, the rest of the sole was much more straightforward. With most of the parts on the sole governed by this section, the narrow band of parts around the edge was meshed to create the master block boundaries. The rest of the parts were filled in, mostly using widths of nine or 18 to attempt to keep the element size constant. Four butterfly meshes were used towards the rear of the club at sections of extreme curvature as done in previous models, shown in Figure 27. A part of the sole near the hosel contains a transition from nine to three elements to transition to a three wide part in the crown band. The most difficult part was meshing the parts with nine thickness elements. The boundaries of these parts needed to be located in a spot that still had decent thickness to avoid small and poor element quality when using the transition block boundaries from nine back to three elements. The locations of these

transition block boundaries are shown in Figure 27 and are represented by the black outline. All the parts inside this outline have nine thickness elements.

Figure 27. Final hybrid sole mesh indicating nine element thick parts and butterfly meshes

4.3.4 The Governing Corner

Unlike the previous model the governing corner part was not meshed first. The complicated sections of the sole dictated how the part of the governing corner was meshed, but it turned out almost identical to the part in the generic driver. It was meshed after finishing the sole because it still would govern some of the parts in the face and crown sections. A block 1 10 13 19; 1 4; 1 6; command was used to create this part, as outlined in Figure 28. The part is five elements wide to define the crown band width and

the middle partition is three wide to define the face band width. The part also dictates the part sizes for the face and crown sections.

Figure 28. The governing corner part showing its three partitions

4.3.5 Face

The face was meshed upon completion of the sole. Each of the parts in the bands outlining the section was meshed first to create the master block boundaries. In this model the larger parts in the middle were then meshed, working towards the edges. Their lengths vary based on the parts dictated by the sole, but all their widths are 18 elements to keep a consistent element size. Two butterfly meshes identical to those used on the face in the driver model were used towards the toe here, and are indicated in Figure 29. Similar to the sole, a section of the face near the hosel contains a transition from three to nine elements to work around the complicated interior surface of the hosel. This was also

done to interface with the three element wide part in the upper band and the nine element wide part in the lower band.

Figure 29. The face topology showing the two locations of the partitioned butterfly mesh parts

4.3.6 Crown

The crown was meshed last. At this point every part in the band surrounding the crown was meshed, so the size of each part was already determined. The meshing process was just a matter of filling in the strips of parts. The part located at the governing corner was a butterfly mesh similar to the driver, except it was a 12x12 part with a 3x3 partitioned removed to avoid very small element sizes. Two more butterfly meshes were used in the corners similar to the previous two models, as shown in Figure 30. A small band of three element wide parts were added near the hosel to finish the mesh and avoid the complicated interior surface of the hosel. Figure 31 shows the final overall mesh of the hybrid.

Figure 30. The hybrid crown topology showing the three locations of the partitioned butterfly mesh parts

Figure 31. Final mesh of the hybrid

4.4 Ball Mesh

The ball used for these simulations was modeled after a standard three-piece ball with an outer cover, mantle and core. A fully featured ball with dimples is extremely difficult to mesh. Rather than using a fully featured CAD model of a ball to project the block parts to, a simplified process was used instead.

4.4.1 Cover

The cover began with a 16 element wide partitioned block in all directions. Since the ball is spherical, a butterfly mesh was required in all directions by deleting the outer corner partitions. The center partition of the block was then deleted leaving six, two element wide cover pieces as shown in Figure 32.

Figure 32. Partitioned block part used for the cover of the ball

The outer surfaces were projected to a spherical surface with a typical ball radius of 0.84 inches, and the inner surfaces to a spherical surface 0.084 inches smaller plus a thousandth of an inch, a standard golf ball cover thickness at maximum tolerance.

4.4.2 Core

The core began with an identical block part to that of the cover. Once again a butterfly mesh technique was required so the outer corner partitions were deleted, as shown in Figure 33.

Figure 33. Partitioned block part used for the core of the ball

The outer surfaces were projected to the same spherical surface used for the inside of the cover but two thousandths of an inch smaller, the minimum cover thickness tolerance.

The inside surfaces of the outer partitions were projected to a spherical surface with a radius of 0.656 inches, representing the inside of the 0.1 inch thick mantle.

This left a very typical spherical butterfly meshed part to decently represent a golf ball. The partitioning and butterfly method left a square block in the center of the part with good element quality in the outer projected partitions. An mseq command was finally used to double the amount of elements throughout the thickness in each direction to better match the element size of the club head meshes. The final model of the ball is shown in Figure 34.

Figure 34. Final mesh of golf ball used in impact simulations

4.5 USGA Plate Mesh

A simple structure was necessary to validate the acoustic impact simulations prior to using intricate club heads. Despite it being obsolete for conformation testing, a USGA

COR plate was used for physical and simulation testing and validation. This is a titanium plate weighing approximately 200 grams, similar in weight and material composition to a golf club head. Its geometry allowed for a simple mesh with many fewer elements than the club heads, which significantly reduced computing time when developing the simulation procedure. The mesh was developed and provided from past thesis projects, and its details can be found in Sharpe (2009) and Volkoff-Shoemaker (2010). The mesh of the USGA plate is shown in Figure 35.

Figure 35. USGA COR plate finite element mesh

5. Acoustic Simulations

Once the finite element meshes were completed the acoustic impact simulations were developed. The simulation was designed to emulate a real world test configuration where a golf ball is fired from an air cannon at a club head placed in an anechoic chamber containing a microphone to record the sound upon impact. This setup was chosen as opposed to modeling a golf swing where the club impacts the ball with an initial velocity for a number of reasons. The golf swing is a complex dynamic motion in which the club has an angular velocity before, at and after impact. Rather than modeling this club velocity profile, the golf ball was assigned a constant initial velocity in one direction towards the face of the club head. This simplified situation will still result in the correct ball speed after impact and closely model the impact from an actual swing. The boundary conditions of the club head during a golf swing are also difficult to model, as there are many dynamic factors involving the connections from the shaft to the human or robot swinging the club. Instead, the club head was given a free boundary condition, which doesn't require further finite element meshing of the shaft, robot etc. Lastly, this is a common real world test configuration in the golf industry that can easily be performed to validate the simulation results.

5.1 TrueGrid® Settings

The simulation parameters can be completely defined using LS-PrePost®, but certain simulation parameters can also be defined within TrueGrid® and output in LS-DYNA® keyword format. The simulation time parameters, ball position, ball velocity and material properties were defined within TrueGrid®.

5.1.1 Impact Settings

A total simulation time of 10 milliseconds with 50 time steps was chosen and defined using the `lsdyopts endtim` and `dtcycl` commands, respectively. This provides ample time for impact and pressure wave propagation to obtain the required acoustic data as well as enough time steps to properly define the impact. The ball was positioned approximately one tenth of an inch from the center of the club face and given a constant velocity of 100 miles per hour in the direction of the club face. This minimizes computation time before impact, which is not relevant to the simulation since the ball has a constant initial velocity.

5.1.2 Ball Material Properties

The golf ball material model was provided by Dr. Tom Mase, a mechanical engineering professor at California Polytechnic State University in San Luis Obispo (Cal Poly), who has an extensive background in the golf industry. An incompressible Mooney-Rivlin rubber material model (#27 in the LS-DYNA® material database) was applied to the golf ball using the `lsdymats` command within TrueGrid®. This material model is based on a strain energy function, as defined (LSTC):

$$W = A(I_1 - 3) + B(I_2 - 3) + C \left(\frac{1}{I_3^2} - 1 \right) + D(I_3 - 1)^2 \quad (11)$$

where A and B are user defined constants and C and D are related to A and B using poisson's ratio ν . The principal invariants $I_1 - I_3$ of the strain energy function are related to the right Cauchy-Green tensor. The following parameters in Table 3 were used to define the material model.

Table 3. Material properties defining the golf ball Mooney-Rivlin material model

Property	Value
Mass density, ρ	$1.106e^{-04} \frac{lb_f \cdot s^2}{in^4}$
Poisson's ratio, ν	0.495
A	3900
B	-390

5.1.3 Club Head & Plate Material Properties

The material model used for the club heads and USGA plate was also provided by Dr. Mase from previous work in this field. The USGA plate is machined out of titanium and typically metal woods are cast using a titanium alloy. A plastic kinematic material model (#3 in the LS-DYNA® material database) was used to define the titanium alloy properties used for these simulations, once again defined using the lsdymats command within TrueGrid®. This material allows the modeling of isotropic and kinematic hardening plasticity after yield by specifying the tangent modulus and yield stress of the material. The parameters used to define the titanium alloy material model are shown below in Table 4.

Table 4. Material properties of club heads and USGA plate

Property	Value
Mass density, ρ	$4.14e^{-04} \frac{lb_f \cdot s^2}{in^4}$
Poisson's ratio, ν	0.31
Elastic Modulus, E	$1.653e^{07} psi$
Tangent Modulus, E_t	$1.653e^{04} psi$
Yield Stress, σ_y	$1.60e^{05} psi$

5.2 LS-DYNA® Settings

Once the LS-DYNA® keyword input file has been created from TrueGrid®, it can be imported into LS-PrePost® to activate the remaining keywords needed to finish the simulation definition. While all of the kinematic impact parameters were defined in TrueGrid®, the acoustic keywords are activated using LS-PrePost®.

5.2.1 Listening Node

First, the listening node needed to be defined. The listening node is a point in the acoustic field where the pressure wave is generated, representing where a microphone or human ear would be. A massless node was created 12 inches behind the club head to represent the microphone location. Activating the *SET_NODE keyword allowed the node to be selected as the field point of interest in the BEM acoustic analysis.

5.2.2 Boundary Elements

Next, the boundary elements of the club head mesh needed to be defined as required to perform a BEM acoustic analysis. Activating the *SET_SEGMENT keyword allowed the boundary elements to be selected using the GUI. A node on the boundary of the club

head was selected and the GUI would select the entire set of elements on that surface by projection. This defined the set of boundary elements of the excited structure to obtain acoustic data from in the BEM acoustic analysis.

5.3.3 BEM Keyword

Finally, the *BOUNDARY_ELEMENT_METHOD_ACOUSTIC card parameters were needed. The acoustic environment in which the analysis takes place was first defined. The frequency range to be analyzed was between 20 and 20,000 Hz, the audible range of the human ear. The acoustic medium used for this simulation was air at standard temperature and pressure, and the material properties used are found below in Table 5.

Table 5. Air properties at standard temperature and pressure

Property	Value
Mass density, ρ	$1.189e^{-07} \frac{lb_f \cdot s^2}{in^4}$
Wave speed, c	$13,169 \frac{in}{s}$
Reference Pressure, P_{ref}	$2.9e^{-09} psi$

The field point and boundary element sets created earlier are designated in this card as well. It is important that the surface normals on the boundary are pointing away from the acoustic medium. Eight domain decompositions were also used to attempt to reduce the computing requirements. The full *BOUNDARY_ELEMENT_METHOD_ACOUSTIC card used is shown below in Table 6.

Table 6. *BOUNDARY_ELEMENT_METHOD_ACOUSTIC card variables used

Card 1	1	2	3	4	5	6	7	8
Variable	Dens	Sp-sound	Min_Freq	Max_Freq	NFreq	Dt-Output	Start_Time	Ref_Pres
Value	1.1889E-7	13,169.265	20	20,000	1001	2.5E-5	0	2.9E-9

Card 2	1	2	3	4	5	6	7	8
Variable	Ext_Pres	Type_Ext	Int_Pres	Type_Int	FFT_Wind			
Value	1	1	0	0	1			

Card 3	1	2	3	4	5	6	7	8
Variable	IBEM_Met	Max_Iter	Res	NDD				
Value	2	1000	1.00E-7	8				

Card 4	1	2	3	4	5	6	7	8
Variable	SSID	SSTYPE	Norm	BEM_Type	Restart			
Value	1	2	1	1	0			

6. Results

All the simulations were performed using both the Boundary Element Method and Rayleigh Method for comparison. Each of the plate simulations and Rayleigh club head simulations were performed using LS-DYNA® version ls971d.59419 R5.0 on a PC workstation running Windows 7 in the QL+ Lab on the Cal Poly campus. The BEM club head simulations were significantly more computational intensive and required computing power unavailable at the Cal Poly campus. These two simulations were performed by Yun Huang at the Livermore Software Technology Company using LS-DYNA® version mpp971d.68394 Dev on a PC workstation running LINUX. This version was used in order to utilize eight parallel processors instead of just one.

6.1 USGA Plate Results

As mentioned in section 4.5, a USGA COR plate was used to validate the acoustic simulation process being developed. It is similar in material and weight to a club head and its simple geometry was advantageous to use as a validation tool for simulation and experimental testing purposes. Roger Sharpe has previously performed a modal analysis on the USGA plate with the same free boundary conditions used in this project. The plate was excited at its center using a modal hammer in an anechoic chamber and the response was recorded using an accelerometer located on the back of the plate. The data was processed using a LDS signal analyzer. Figure 36 below shows the first two main

excitation modes at 3,940 and 11,060 Hz. This physical test was used to validate a modal analysis of the plate performed in LS-DYNA® that returned 3,917 and 11,178 as the first two excitation frequencies, closely matching the experimental test results (Sharpe 2009).

Figure 36. Experimental modal analysis results of the USGA plate

It is expected that the impact of the USGA plate and golf ball should produce an acoustic response with similar data to that of the modal tests. During the development of the simulation process in LS-DYNA®, the Rayleigh method was used to perform the simulation because of its very short run time, as seen in Table 7. The final analysis was performed using both the Rayleigh and BEM for comparison.

Table 7. LS-DYNA® simulation statistics for the USGA COR plate

	BEM	Rayleigh
Number of Nodes	8298	8298
Number of Elements	6656	6656
Operating System	Windows 7	Windows 7
Number of Processors	1	1
Memory Required	13.91 GB	0.78 GB
Simulation Time	2 hours, 49 minutes	5 minutes, 10 seconds

Figure 37 shows the acoustic pressure levels of the USGA plate acoustic simulation using the BEM in decibels and psi. The first two excitation frequencies can be seen at 2,410 and 9,940 Hz. These frequencies are roughly 1,000 Hz lower than the experimental tests, yet still show good correlation.

Figure 37. Acoustic response of USGA COR plate impact simulation using the Boundary Element Method

Figure 38 shows the acoustic pressure levels of the USGA plate acoustic simulation using Rayleigh's method in decibels and psi. This method shows four main excitation peaks at 2,420, 4,440, 5,930 and 9,720 Hz instead of only the two present in the BEM simulation. The two peaks seen in the previous simulation correlate extremely well with these results, while the two other excitation frequencies in between are prominent in the Rayleigh analysis and can only be faintly seen in the BEM analysis.

Figure 38. Acoustic response of USGA COR plate impact simulation using Rayleigh's Method

6.2 Generic Driver Results

The first club head model analyzed was the generic driver head. Again, both the BEM and Rayleigh simulations were performed. Table 8 shows a significant increase in simulation time for the driver head over the USGA plate. The BEM simulation performed at LSTC failed midway through the iterative process and was continued from the failure point and completed using another session, so the memory requirement and simulation time data is not available (but should be similar to the hybrid BEM data shown later). A glue command in LS-DYNA® was utilized in order to combine the data compiled from the failed analysis with the second analysis.

Table 8. LS-DYNA® simulation statistics for the generic driver head

	BEM	Rayleigh
Number of Nodes	127173	127173
Number of Elements	100207	100207
Operating System	LINUX	Windows 7
Number of Processors	8	1
Memory Required	N/A	9.43 GB
Simulation Time	N/A	8 hours, 1 minute

Figure 39 shows the acoustic pressure levels of the driver acoustic simulation using the BEM in decibels and psi. The acoustic response appears to be driven by four main excitation frequencies in the frequency range of 4,000 to 10,500 Hz. This acoustic response correlates to some degree with that of the Rayleigh analysis of the USGA plate.

Figure 39. Acoustic response of the driver impact simulation using the Boundary Element Method

Figure 40 shows the acoustic pressure levels of the driver acoustic simulation using Rayleigh's method in decibels and psi with similar results.

Figure 40. Acoustic response of the driver impact simulation using the Rayleigh Method

6.3 Hybrid Results

The fully-featured hybrid head was the final club simulation performed. Table 9 shows the advantage in simulation time of running a machine with parallel processing power. Most of the simulation time is taken up by the explicit impact problem. The non-iterative Rayleigh process typically takes under a minute to complete while the BEM process will take significantly more time. Despite the iteration required by the BEM analysis, the overall simulation time is much lower with parallel computing than the Rayleigh analysis

with one processor. The memory requirement of 398 GB is significant to complete the BEM iterations.

Table 9. LS-DYNA® simulations statistics for the fully featured hybrid head

	BEM	Rayleigh
Number of Nodes	101577	101577
Number of Elements	84856	84856
Operating System	LINUX	Windows 7
Number of Processors	8	1
Memory Required	398 GB	10.2 GB
Simulation Time	6 hours, 59 minutes	18 hours, 49 minutes

Figure 41 shows the acoustic pressure levels of the hybrid acoustic simulation using the BEM in decibels and psi. The acoustic response again appears to be driven by four main excitation frequencies but in a tighter frequency range of 5,500 to 8,000 Hz. This difference is reasonable as hybrid clubs typically make a much different sound on impact as a driver head.

Figure 41. Acoustic response of the hybrid impact simulation using the Boundary Element Method

Figure 42 shows the acoustic pressure levels of the hybrid acoustic simulation using Rayleigh's method in decibels and psi with similar results.

Figure 42. Acoustic response of the hybrid impact simulation using the Rayleigh Method

6.4 BEM & Rayleigh Comparison

Due to its significant time and computational savings, the Rayleigh method was evaluated in addition to the BEM. As discussed in section 2.2, the Rayleigh method is widely considered an unreliable method for accurately predicting sound pressure, although recent research has shown otherwise. Both methods were performed for each impact scenario to determine whether the Rayleigh method could be a valid time-saving alternative to the BEM.

Figure 43 shows both the Rayleigh and BEM frequency responses for the USGA COR plate impact. The Rayleigh method shows correlation in peak frequency location, but differs in magnitude at certain peaks.

Figure 43. USGA plate impact simulation using the BEM and Rayleigh methods

Figure 44 shows both the Rayleigh and BEM frequency responses for the generic driver impact. The Rayleigh method correlates well at each major and minor peak frequency in both location and amplitude.

Figure 44. Driver impact simulation using the BEM and Rayleigh methods

Figure 45 shows both the Rayleigh and BEM frequency responses for the fully featured hybrid impact. The Rayleigh method correlates well at each major and minor peak frequency in both location and amplitude, as seen with the driver impact.

Figure 45. Hybrid impact simulation using the BEM and Rayleigh methods

These results indicate that the Rayleigh method could be a viable time-saving alternative to the BEM for acoustic response simulations of metal wood and golf ball impacts. While the Rayleigh USGA plate impact simulations did not show perfect correlation in amplitude, they did still correlate in location. It is possible that the metal woods are better represented under the assumptions made by the Rayleigh method than the plate, and therefore return better results. Further testing of a wider range of club heads is necessary to verify that the Rayleigh method is indeed accurate for these simulations.

7. Validation

It is essential to validate any software based simulations with experimental testing. As mentioned in section 4.5, the USGA COR plate was used for all experimental tests. The goal in designing the test configurations was to develop a procedure and environment that can be modeled in the simulations, and vice-versa. While there are some inherent differences between a real world test environment and a simulation, the experiments should still provide a solid baseline for validating the computer analyses. Two validation procedures were developed: a tap test and an air cannon impact test.

7.1 Tap Testing

An impact hammer tap test was first performed on the USGA plate mainly to verify that the acoustic acquisition equipment was performing properly and to develop a procedure for appropriately acquiring acoustic data in a controlled setup. In this procedure, the USGA plate was struck in its center with an impact hammer. In order to achieve the free boundary conditions specified in the simulation, the USGA plate was hung from a support by a small string. This allowed the plate to move freely upon impact to closely resemble a true free boundary condition. The acoustic response of this impact was acquired using a G.R.A.S. 40AE prepolarized microphone and G.R.A.S. 26CA preamplifier positioned roughly 12 inches behind the USGA plate. The frequency range

of the microphone is 3.15 Hz to 20 kHz, which covers the audible range of the human ear and closely matches the range analyzed in the simulations. The microphone setup was powered by a PCB Piezotronics 484B constant current power supply and connected to an LDS data acquisition system. PRO FOCUS II software contains a real time FFT analyzer that processed the acoustic data from the LDS for analysis. The software was set up to begin recording data when triggered by the sound of the impact and stop recording after a very short time span to minimize recording any ambient noise. Figure 46 shows three main modes attributing to the acoustic response of this impact.

Figure 46. Frequency response of USGA plate impact hammer tap test

7.2 Air Cannon Testing

After developing the acoustic data acquisition procedure, a more representative test of the simulation impact setup was performed. A golf ball was fired out of an air cannon at 100 mph towards the center of the USGA plate placed in an area enclosed with safety nets. The test setup is shown below in Figure 47.

Figure 47. Air cannon impact test setup

This air cannon was designed and built by Nickolai Volkoff-Shoemaker and Roger Sharpe and is capable of operating at pressures of up to 130 psi (Volkoff-Shoemaker 2010). Full design specifications can be found in the publications of Volkoff-Shoemaker (2010) and Sharpe (2009). The air cannon required calibration to relate input air pressure to output ball velocity. Using the calibration data found in Volkoff-Shoemaker (2010), it was determined that an input air pressure of roughly 65 psi was required to obtain a ball

velocity of 100 mph to match the simulation. A free boundary condition was again required for the USGA plate. Rather than suspending the USGA plate, it was instead stood upright on a platform as shown in Figure 48. When impacted, the USGA plate is allowed to fly freely backwards, better representing the response seen in the simulations.

Figure 48. USGA plate free boundary condition for air cannon impact testing

The same microphone and data acquisition setup used for the tap testing was used for this experiment. The microphone was placed behind the nets close to 12 inches behind the plate so as not to damage the microphone. The data acquisition was triggered by the sound of the air cannon firing. Ideally, the air cannon noise data should not be acquired to achieve just the ball and plate impact response. Due to the extremely high ball velocity and short travel distance of roughly ten feet before impact, properly delaying the acquisition until just before impact would be difficult and would still include some response from the air cannon. A dry fire was performed with no plate impact and the

main modes governing the acoustic response found were at lower frequencies than that of the expected plate impact. Therefore, it was concluded that the data acquired from the air cannon noise would not significantly alter the impact acoustic response data. The results from the air cannon impact testing are shown in Figure 49. This data contains significantly more noise than the tap testing, but four main modes governing the acoustic response are shown at roughly 4,250, 7,500, 8,000 and 12,500 Hz.

Figure 49. Frequency response of USGA plate air cannon impact test

7.3 Results

The USGA plate air cannon impact and tap test results correlate well, as seen in Figure 50. The frequencies of the two main excitation modes at ~4,000 and ~12,000 Hz appear

in both test results. The tap test shows another mode at 8,000 Hz, while the air cannon shows two at roughly the same location. This could be attributed to the large difference in impact force and area between the impact hammer and golf ball.

Figure 50. Combined results of the USGA plate tap and air cannon impact tests

The air cannon results containing the two middle modes correlate best with the Rayleigh simulation USGA plate results. Their frequency responses both contain four main excitation modes distributed similarly across the frequency spectrum. The simulation results show each of the main frequencies at roughly 2,000 Hz lower than the experimental results as shown in Figure 51.

Figure 51. Combined results of the USGA plate LS-DYNA® and air cannon impact tests

There are many factors that could contribute to this difference. The simulations assume that the acoustic medium is infinite while the room in which the tests were performed is far from that. The walls and other objects inside the room can have an effect on the acoustic response. The temperature and pressure inside the room effects the properties of air which could differ from those specified in the simulations. Also, only the club or plate's acoustic response was obtained during the simulations because only its boundary elements were defined and specified for analysis. Both the plate and ball will affect the acoustic response during the experiment.

8. Conclusions

An improved and repeatable method for meshing golf club heads with finite elements in TrueGrid® was developed. Using solid brick elements through the thickness of the club head instead of shell elements better represents the many thickness variations throughout each section of a club head. Building up the mesh by wrapping strips of small block parts around the club head provides a methodical way to efficiently mesh the head. Intricate features of the part can still be meshed reasonably well to improve the accuracy of the mesh. This method also results in a high quality mesh at the center of the club head sections while still maintaining decent quality at the edges, which serves well for acoustic simulations.

A simulation procedure was also developed to predict the acoustic pressure at a designated point in an acoustic medium of a golf club and ball impact using the BEM and Rayleigh methods in LS-DYNA®. This simulation can be repeated for any club head mesh with minimal work. A mesh can be inserted into the TrueGrid® file developed that outputs the LS-DYNA® keyword file. The final simulation parameters are set in LS-PrePost® prior to creating the final file used to run the analysis. The simulation time and computing power required for the impact is modest, while the acoustic simulation time and computing power is much greater. The Rayleigh method provides an alternative which can greatly reduce these requirements. The three simulations performed in this project suggest that the Rayleigh method can be a viable substitute for the intensive BEM

analysis. More simulations are necessary to further prove the accuracy of this method but it is practical to use as a quick preliminary run.

The simulation of sound produced from the ball and a USGA COR plate, generic driver, and hybrid impact was accomplished with reasonable results. Experimental testing was performed using a USGA plate to validate these results. A simple tap test was performed impacting the plate with a modal hammer and recording the acoustic response with a microphone. In addition, an air cannon test was performed impacting the plate with a ball fired at 100 mph, again recording the acoustic response. These two tests correlated with each other, indicating that air cannon procedures could be negated in favor of a much simpler tap test during prototype testing for acoustics. Further testing would be necessary to prove this to be true. The simulation frequency responses showed similar results to the experimental tests except the general location of the excitation modes were roughly 2,000 Hz lower in the simulations. This could be attributed to a number of discrepancies between the simulation and testing environment, as explained in section 7.3. Despite the differences in results, the general correlation indicates the simulation procedure works reasonably well in predicting the acoustic response and could be implemented into the design phase of golf clubs. More tap and air cannon impact testing is necessary using driver and hybrid club heads to validate these simulations, and other clubs should be meshed and tested as well. A mesh convergence study should also be performed to attempt to better match the experimental and analytical results.

At this point, the frequency response obtained from the club head simulations shows the main excitation frequencies that determine the sound. It remains to be determined which frequencies attribute to a “good” and “bad” sound at impact and could require extensive

impact testing of various club heads to deduce the desired acoustic frequency range. The end game for this project would be to take the frequency responses obtained from the simulations and convert them to a sound file for playback. This would require further development of the simulation procedure and may require the addition of the hosel and shaft to the mesh, assigning the club an initial velocity with the proper swing path, and including environmental effects into the simulation. These factors could increase the required computing power to unrealistic levels and may be unreasonable to achieve. The procedure developed in this project, though, can still be a viable and effective method for determining the acoustic response of the golf club and ball impact.

Bibliography

"A History of the Golf Ball" Golf Europe: Golf Courses in Ireland, Scotland, England and more. Web. 2009. <http://www.golfeurope.com/almanac/history/golf_ball.htm>.

"A History of the Golf Club" Golf Europe: Golf Courses in Ireland, Scotland, England and more. Web. 2009. <http://www.golfeurope.com/almanac/history/golf_club1.htm>.

Alia, A. and M. Souli. Acoustic and Vibroacoustic Modeling in LS-DYNA® Based on Variational BEM. 5th European LS-DYNA® Users Conference. 2005.

Grimsley, W. Golf: Its History, People & Events. Prentice-Hall Inc., Englewood Cliffs, New Jersey. 1966.

Herrin, D. W., F. Matinus, T. W. Wu, and A. F. Seybert. A New Look at the High Frequency Boundary Element and Rayleigh Integral Approximations. Society of Automotive Engineers. 2003.

Huang, Y. A New Acoustic Solver Based on Boundary Element Method in LS-DYNA®. Livermore Software Technology Corp., CA.

Huang, Y., M. Souli. Simulation of Acoustic and Vibro-Acoustic Problems in LS-DYNA® using Boundary Element Method. Livermore Software Technology Corp., CA. 2008.

LSTC. LS-DYNA® Keyword User's Manual. Livermore Software Technology Corporation, Livermore, California. 2007.

- Nesbitt, R.D., Sullivan, M.J., T. Melvin. "History and Construction of Non-wound Golf Balls" Science and Golf III: Proceedings of the 1998 World Scientific Congress of Golf. Farrally, M.R., Cochran, A.J. The World Scientific Congress of Golf Trust, UK. 1999.
- R&A & USGA. "Rules of Golf" Guide. 2007.
- Rossing, T. Springer Handbook of Acoustics. Springer Science+Business Media, New York 2007.
- Sharpe, R. A Prediction of the Acoustical Output of a Golf Driver Head Using Finite Elements. Master's Thesis. California Polytechnic State University, San Luis Obispo, California. 2009.
- Tai, CH., Chao CY., Leong, JK., Q.S. Hong. "Effects of golf ball dimple configuration on aerodynamics, trajectory, and acoustics" Journal of Flow Visualization and Image Processing. Begell House Inc., Redding, Connecticut. 2007.
- Thomas, F. "Everything You Need to Know About COR" Golf Digest. 2002.
- Volkoff-Shoemaker, N. Finite Element Analysis of a Golf Ball and Driver Head Impact: Understanding the Feasibility of an Acoustical Optimization. Master's Thesis. California Polytechnic State University, San Luis Obispo, California. 2010.