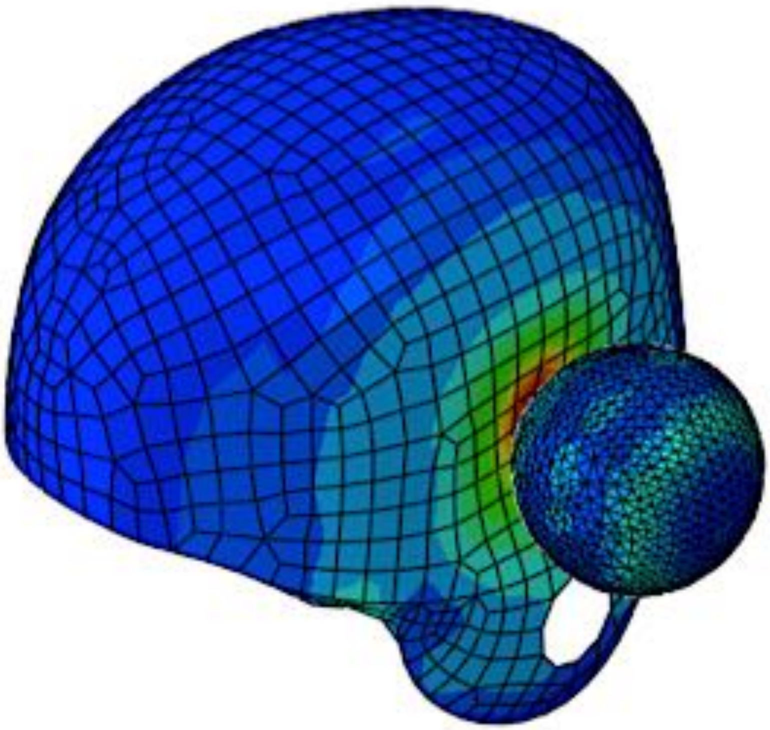

ME 461:
Finite Element
Analysis

Fall 2015

A Semester Report on:

The Development and Analysis of the Impact of a Baseball on a Baseball Helmet

Group Members:

Alyssa Rubinstein, John “Grant” Caspero, Emily Karas, Ryan Wise

PennState
College of Engineering

Department of Mechanical and Nuclear Engineering, University Park, PA

Table of Contents

Executive Summary

The following report summarizes the development and execution of a contact simulation between a baseball and baseball helmet. This simulation was aimed to mimic a pitch hitting a player in the side of the head. We have analyzed the resulting stresses and strains applied to the helmet by the baseball.

Before starting the construction of the model, research was required to obtain background information on the topic. Statistics and parameters were gathered from Major League Baseball records. These parameters included knowledge such as the distance a batter stands from the pitcher, the average speed of a professional pitch, the materials used in both helmets and baseballs, and how frequently these incidents typically occur.

Further research was conducted to define the standard material properties of each component involved in this simulation and the proper dimesnions for each part as well. These factors allowed accurate representations of both a baseball helmet and ball to be created in SolidWorks. Due to the complex geometry of the helmet, a previously created model was obtained from the Internet and modified in order to fit the needs of this project. The baseball has a much simpler geometry, and was created in two parts to be assembled in Abaqus – the core and the outer shell.

The meshes for both components of the simulation were refined numerous times as minor problems surfaced throughout the development stages. The helmet was inputed into Abaqus as a pre-assembled part, where it was eventually converted into ANSYS to complete the final meshing. During this stage, the bill was removed due to meshing errors. The baseball was inputed into Abaqus as two separate parts, where it was then assembled and merged together in order to be properly meshed.

Two crucial boundary conditions were assigned to this simulation. A patch of nodes on the helmet was assigned a constraint of zero displacement, thus fixing the helmet in space. The entire ball was assigned a linear velocity of 40.2 m/s in the x-direction, with zero rotational conditions. This velocity caused the baseball to impact the helmet, similar to a real world occurance.

A surface-to-surface contact was created with a dynamic, explicit interaction in Step 1. Two behaviors were assigned to the contact interaction: tangential behavior and normal behavior.

After the above steps were completed, the job was run. The simulation took 12.52 seconds of computing time for one second of real-time. The ball was moved closer to the helmet than initially planned to decrease the amount of time and money needed to run test trials. In the final model, the ball struck the helmet at 40.2 m/s, which resulted in a Von Mises stress of 9.792 MPa. The impact caused a ripple effect that propagated outward in waves from the contact point. In conclusion, the impact of a professional baseball pitch does not exceed the yield strength of an ABS polymer helmet; therefore, today's standard helmets properly protect the players.

Acknowledgements

We would like to thank the following people for their guidance and assistance in the completion of this project:

- Dr. Reuben Kraft for sharing his knowledge of Abaqus and the Finite Element Method.
- Dooman Akbarian for aiding in the meshing and simulation processes.
- Major League Baseball for allowing us to analyze the structural safety and effectiveness of its protective headgear.

List of Figures

Section 1: Background and Project Plan

Objective:
The objective of our team's efforts is to accurately analyze the stresses and strains of a baseball hitting a stationary helmet at 40.2 m/s, and to ensure that the impact does not exceed the yield strength of the helmet material. This model resembles the effects of a baseball player being hit by a pitch and will be developed through Finite Element Analysis.

Figure 1: Ideal Situation for Analysis

Background Information:
Baseball batters are required to stand 18.4 meters from a pitcher throwing fastballs weighing 0.145 kg at 40.2-44.7 m/s. On occasion, the pitcher will miss his target and hit the batter. To prevent serious head injuries, batters are required to wear helmets. These helmets must endure a great deal of stress to keep the batter's head safe. Fortunately, with modern day helmet technology, head injuries are rare – about 10 major league players per season obtain concussions. We want to analyze the magnitude of stress the helmet will endure, and model the strains and deformations that typically occur during a ball-to-helmet collision.

General Approach:
A model will need to be created for both the baseball and helmet. The ball will be created under the assumption that it is a perfect sphere and has mechanical properties similar to the material of a standard baseball. Due to its tricky design, the helmet model was found online, but was made with a simplistic, round shape. It will have mechanical properties of ABS polymer – the current material that helmets are constructed of. In modeling the collision, the helmet will remain fixed in space and the baseball will come in contact at approximately 40.2 m/s. The resulting collision will be analyzed numerically and visually as a result of finite element analysis.

External Loading Conditions:

Average pitch speed: 40.2 m/s (90.0 mph)
Mass of baseball: 0.145 kg

$F = m\Delta V/\Delta t$ (Force = mass * change in velocity / change in time)
$F = [40.2 \text{ m/s} * (0.145 \text{ kg})] / (0.007 \text{ s})$
**the impact time of a baseball striking a helmet is estimated to be 7 milliseconds*
$F = 833$ N

Section 2: Development and Description of the CAD Geometry

Material Properties (HELMET):
Outer Shell: ABS Polymer (Acrylonitrile Butadiene Styrene)

Density = 1.07 g/cm^3
E = 2.25 GPa
v = 0.29 (Poisson's Ratio)
Mass = 0.9072 kg
Yield Strength = 48.3 MPa

*This makes sense because hard plastic should have a large density and elastic modulus but a very small mass.

CAD Geometry Description:
Below are various views of the SolidWorks file used for the helmet. Our group experienced difficulties in modeling the helmet; therefore a model found on the Internet was used instead. This part has very complicated geometry, which was best illustrated through multiple figures. The helmet is symmetric about the centerline, which is highlighted in light blue. Each dimension can be seen below. **All dimensions are in mm.**

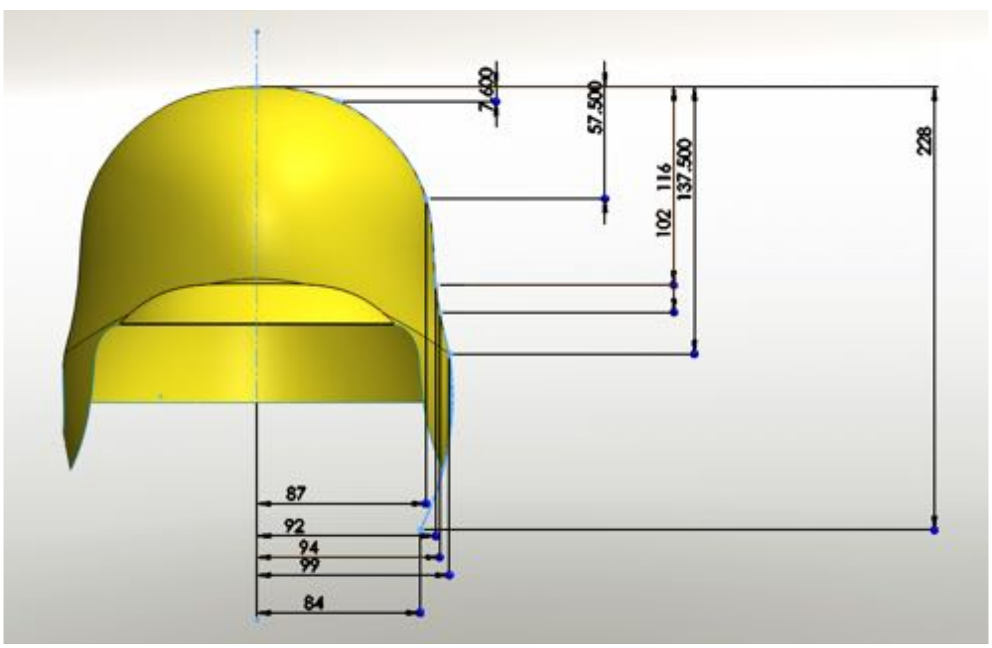

Figure 2: Front View of Helmet Geometry

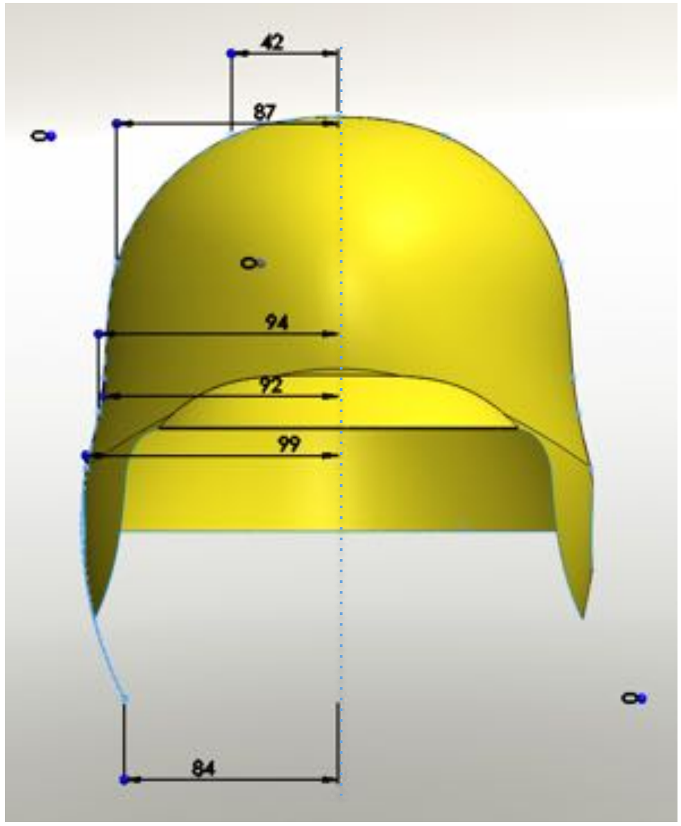

Figure 3: Additional Front View of Helmet Geometry

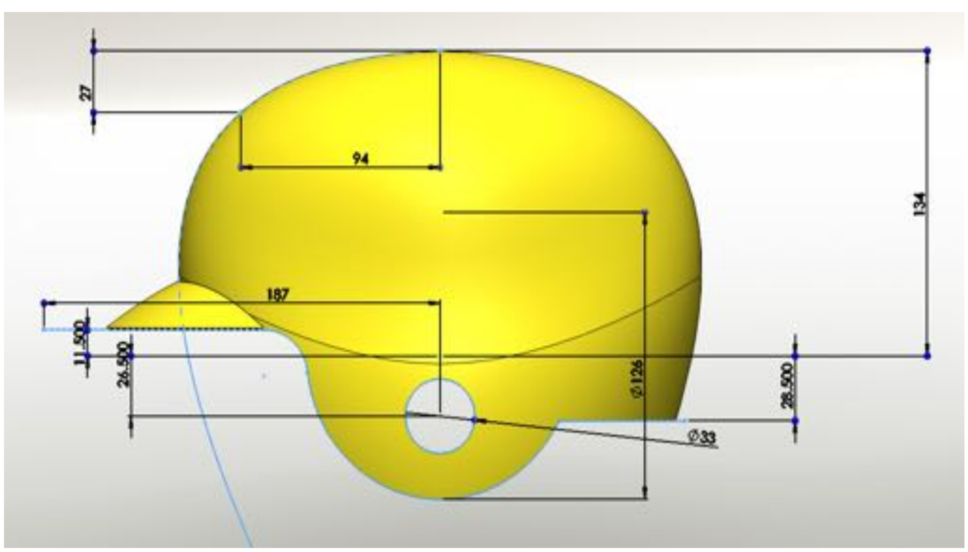

Figure 4: Side View of Helmet Geometry (Right)

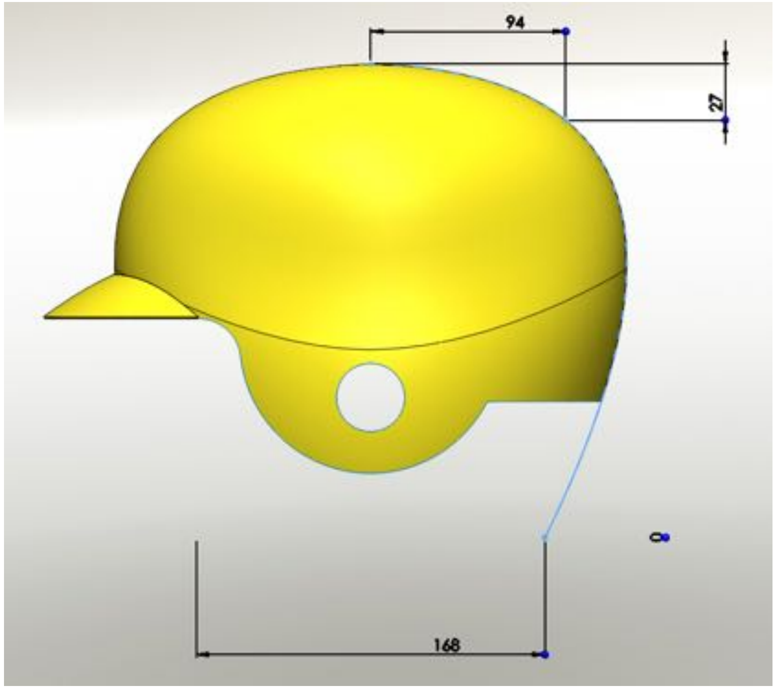

Figure 5: Additional Side View of Helmet Geometry (Right)

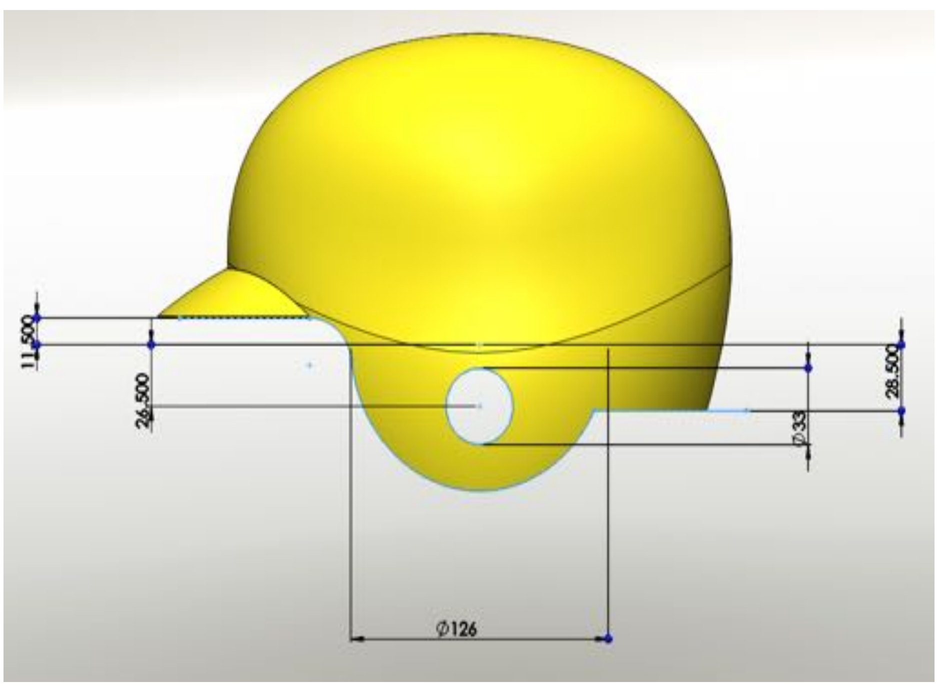

Figure 6: Third Side View of Helmet Geometry (Right)

Material Properties (BASEBALL):

Core: Rubber-Coated Cork

Density = 200 kg/m^3

E = 6 MPa

v = 0.1

Outer Shell: Woven Yarn

Density = 1000 kg/m^3

E = 70 MPa

v = 0.44

CAD Geometry Description:

Below, the dimensions of the baseball can be seen. The core of the baseball, shown in Figure 7, is a simple sphere with a diameter of 20 mm. Figure 8 illustrates the outer shell of the baseball. This part is a sphere that has a total diameter of 74 mm, with a hollowed out region to fit the core inside. These two parts were assembled Abaqus so that each component could be independently meshed. **All dimensions are in mm.**

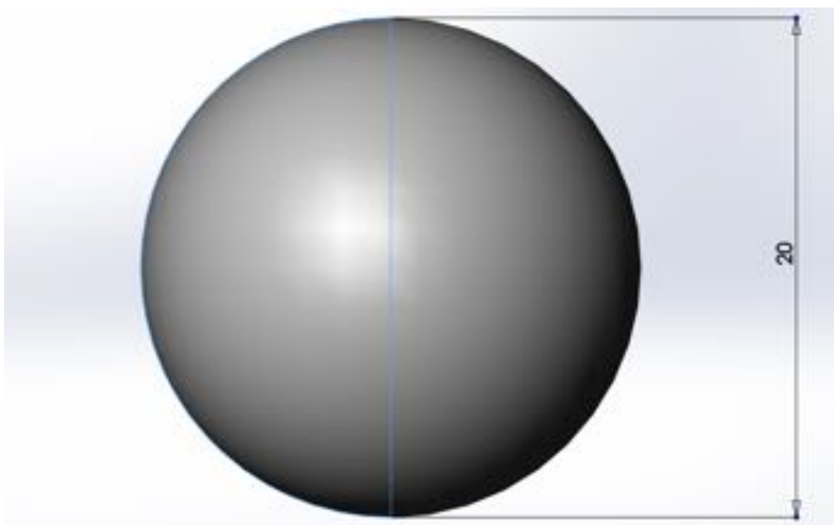

Figure 7: Core of Baseball Geometry

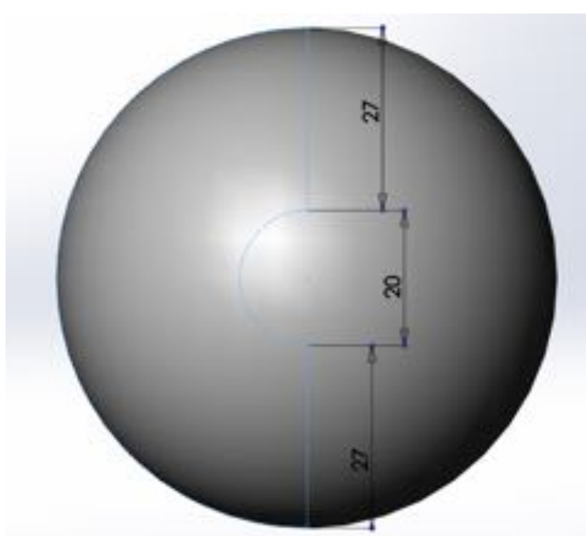

Figure 8: Outer Shell of Baseball Geometry

Section 3: Development of Finite Element Meshes

Helmet Mesh Steps:

1. Import: The baseball helmet is made up of 14 different parts. The SolidWorks file was imported as a single .step file to avoid tedious assembly. Each individual part of the helmet was therefore pre-assembled in Abaqus. The material properties, as stated in Section 2, were then assigned accordingly.
2. Assign Mesh Controls: Each individual component of the helmet was assigned a specific type of element mesh control. The earpieces and back of the helmet were designated as free surfaces with quadrilateral elements. The headpiece of the helmet was designated as a structured surface with quadrilateral elements.
3. Abaqus Mesh: The mesh needed to be coarser than what was previously chosen in order to reduce both the time and monetary constraints of running trials on the helmet. In order to mesh the entire baseball helmet, a mesh was applied to the entire part at one time. 1675 elements were generated.
4. ANSYS Mesh: In order to refine the mesh, the helmet model was exported to ANSYS. At this point, the bill of the helmet was removed to alleviate the number of error meshes created. After these adjustments, the final mesh consisted of 1205 elements. The global seed size was lost in converting between programs, resulting in an "Orphan Mesh Part" with no size assignments.

Element Type/Quality

Type: Quadrilateral (free and structured surfaces)
Quality: Number of elements: 1205
Analysis errors: 0 (0%)
Analysis warnings: 10 (0.8299%)

The majority of the helmet was meshed using quadrilateral structured elements. However, due to the curvature of the earpieces and back of the helmet, quadrilateral free elements were needed to form more freely and properly around the piece.

Figure 9: Meshed Baseball Helmet

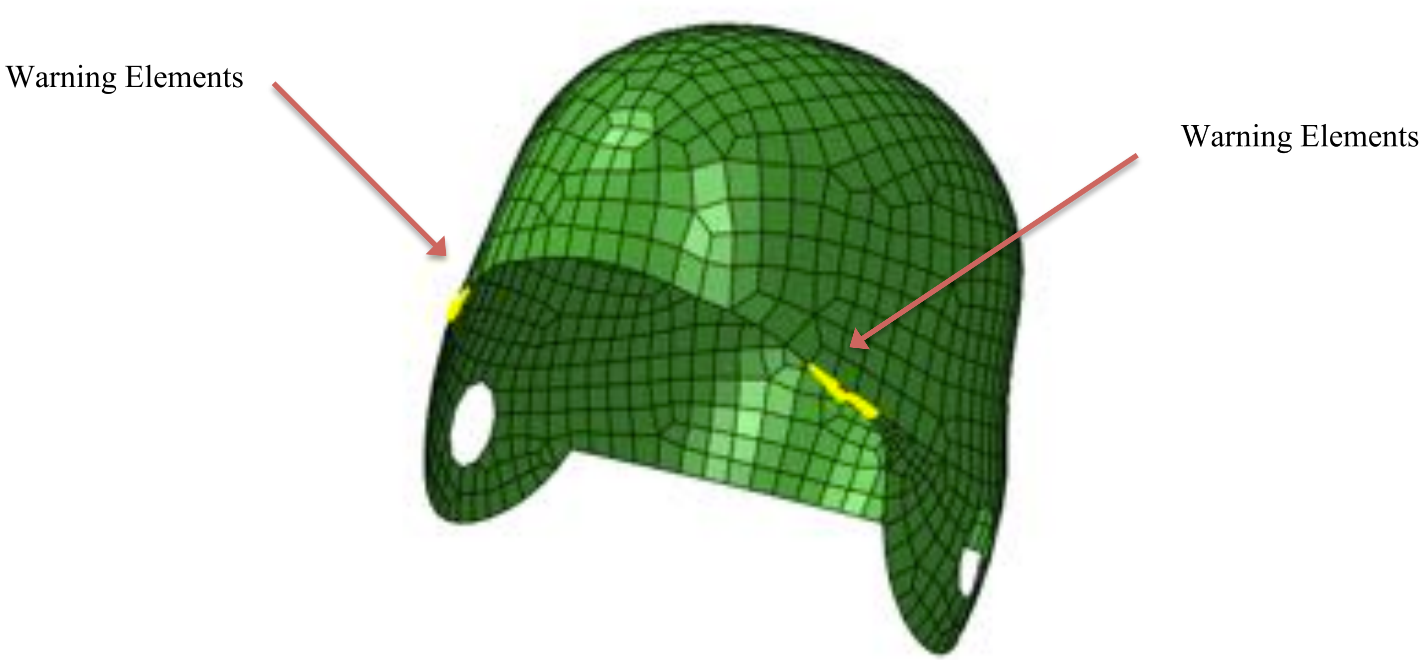

Figure 10: Highlighted Error Meshes

Baseball Mesh Steps:

1. Import**:** Part 1 was imported into Abaqus, which consisted of the core that was .02 m in diameter. Afterwards, Part 2 was imported, which consisted of the hollow sphere with an outer diameter of .074 m and hollow center with a diameter of .02 m. Each of the parts were individually saved as a .step file and imported into Abaqus. In Abaqus, the material properties were assigned accordingly, as stated in Section 2. In order to do this, a section for each part was created and then the respective sections were assigned their material properties.
2. Assembly: In the assembly module, an independent assembly was created to incorporate the core into the hollow center of the outer shell.
3. Assign Mesh Controls: In the mesh module, mesh controls were set for the entire assembly. Tetrahedral elements were used, along with the default settings. When creating seed edges, the default settings were chosen with a global seed size of 4.
4. Mesh Part: The mesh was created, which resulted in the images shown below. 1887 elements were created in total.

Element Type/Quality

Type: Tetrahedral (structured surfaces)

Core Quality: Number of elements: 397

Analysis errors: 0 (0%)

Analysis warnings: 0 (0%)

Outer Shell Quality: Number of elements: 1490

Analysis errors: 0 (0%)

Analysis warnings: 0 (0%)

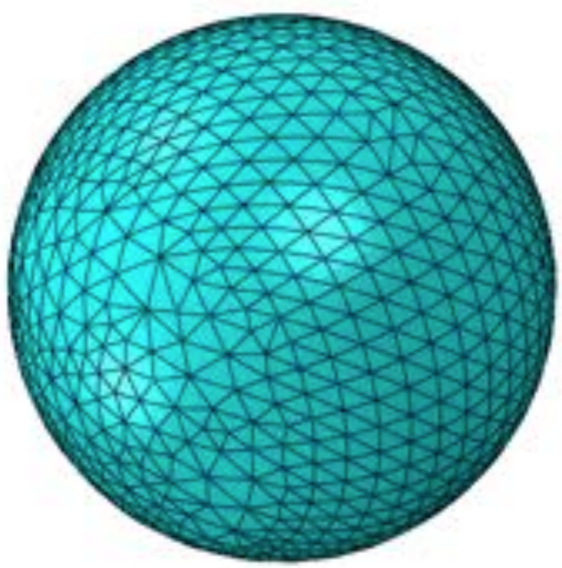

Figure 11: Meshed Baseball Assembly

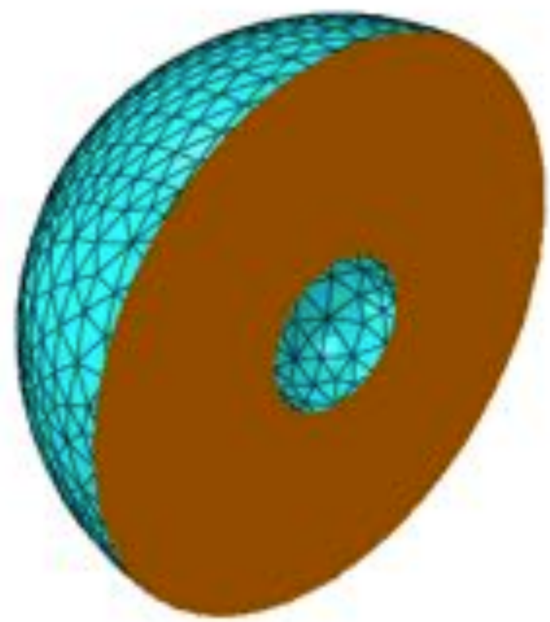

Figure 12: Cross-Section of Meshed Baseball Assembly

** There are no mesh error figures due to 0 errors existing in the ball mesh.*

Section 4: Development and Description of the Model Assembly and Boundary Conditions

The helmet did not require any assembly because it was imported into Abaqus as a single part.

Helmet Boundary Conditions:

1. Surfaces: A surface was created by selecting several elements on the left side of the helmet (highlighted in orange below). This elemental region serves as an invisible wall that holds the helmet stationary during the simulation.
2. Load/Condition Type: At the newly created surface, a Displacement/Rotation Constraint was created to pin the helmet in place. A zero displacement condition was applied to the U1, U2, and U3 vectors. In addition, a zero rotation condition was applied to the UR1, UR2, and UR3 angular vectors. This effectively restrained the helmet from moving translationally and rotationally in all three planes.

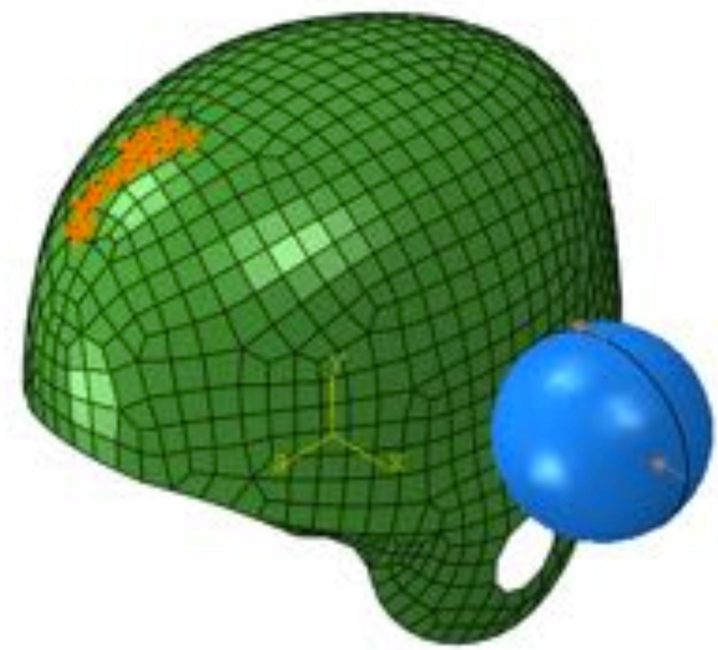

Figure 13: Pinned in Place Baseball Helmet

The baseball required some assembly as it consists of a center sphere as the core and a hollow outer sphere, which encases the core. The core was merged with the outer shell to complete the baseball model.

Baseball Boundary Conditions:

1. Surfaces: The entire ball assembly was selected as a whole because the ball moves as a single unit.
2. Load/Condition Type: When creating the boundary condition for the ball, a linear velocity step type was used. The V1 vector was selected and inputed as 40.2 m/s to assign the ball movement in only the x-direction. We did not assign an angular velocity; therefore the ball has no rotational movement.

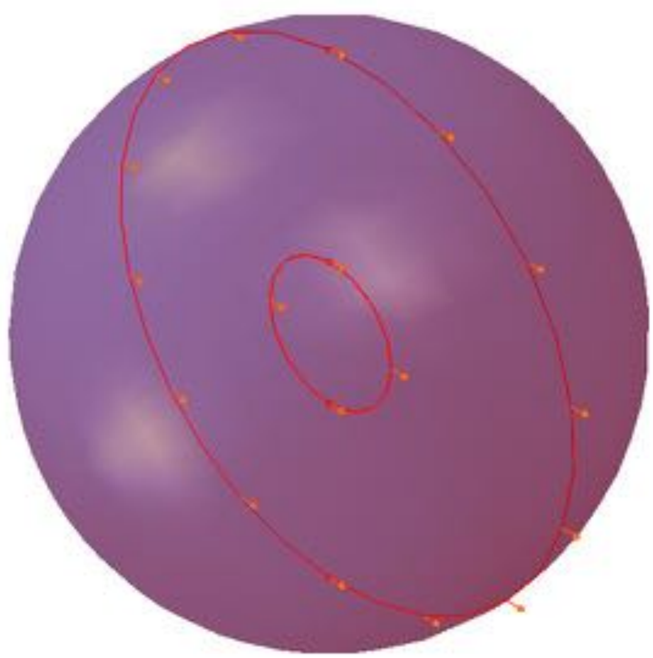

Figure 14: Velocity/Angular Velocity Conditions on Ball Assembly

Section 5: Development and Description of Model Interactions

A surface-to-surface contact was created with an explicit interaction in Step 1 (Dynamic, Explicit). The two surfaces involved in the kinematic contact constraint are the right side of the helmet and the ball. All other contact properties were selected as the default options. Under contact property options, two behaviors were selected. The first contact was a tangential behavior with frictionless characteristics. The second contact was a normal behavior with "hard" contact selected under the Pressure-Overclosure category. All other constraints were set as default.

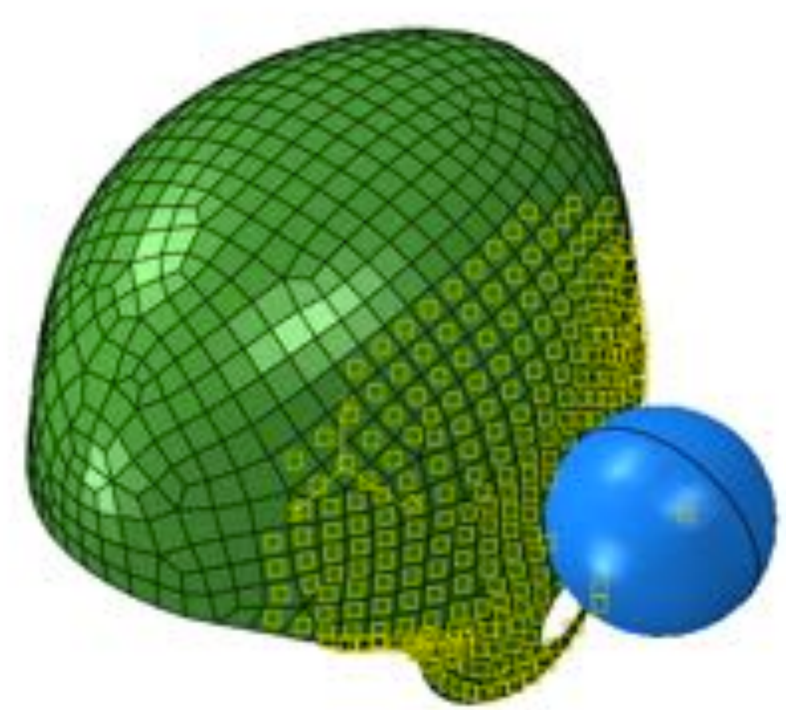

Figure 15: Side of Helmet and Ball Interaction

Section 6: Analysis of Finite Element Model

Intially, the ball was oriented the appropriate 18.4 m away from the helmet, but the simulation was too expensive and time consuming to run repeatedly. Instead, the simulation was run by orienting the ball directly next to the point of contact on the helmet, only about 5 mm away. A dynamic simulation was run, in which the ball was given an initial velocity of 40.2 m/s. The simulation took 12.52 seconds to computue 1 second of real time. This final simulation took longer to complete than the first attempt before any mesh refinements were made. This is because the ball mesh was refined fom a global seed size of 8 to 4, creating more elements to include in the calculations.

The intial job caused the ball to split in half due to an incorrect part selection when setting the boundary conditions. After adjusting the mesh and boundary conditions of the ball, the new simulation ran successfully. Figure 17 below illustrates the colored contour model of the Von Mises stresses upon impact. The maximum stress occurs in the red region with a magnitude of of 9.792 MPa.

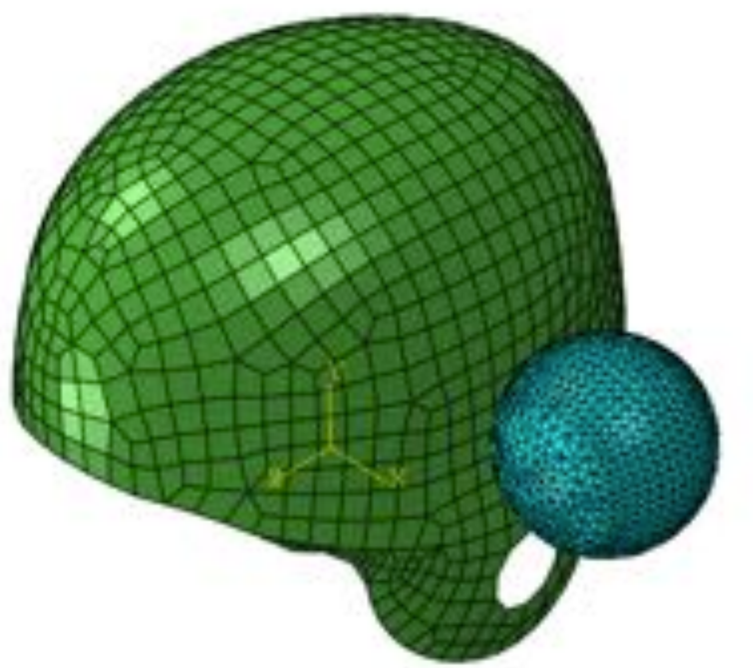

Figure 16: Mesh and Assembly of Job Prior to Simulation

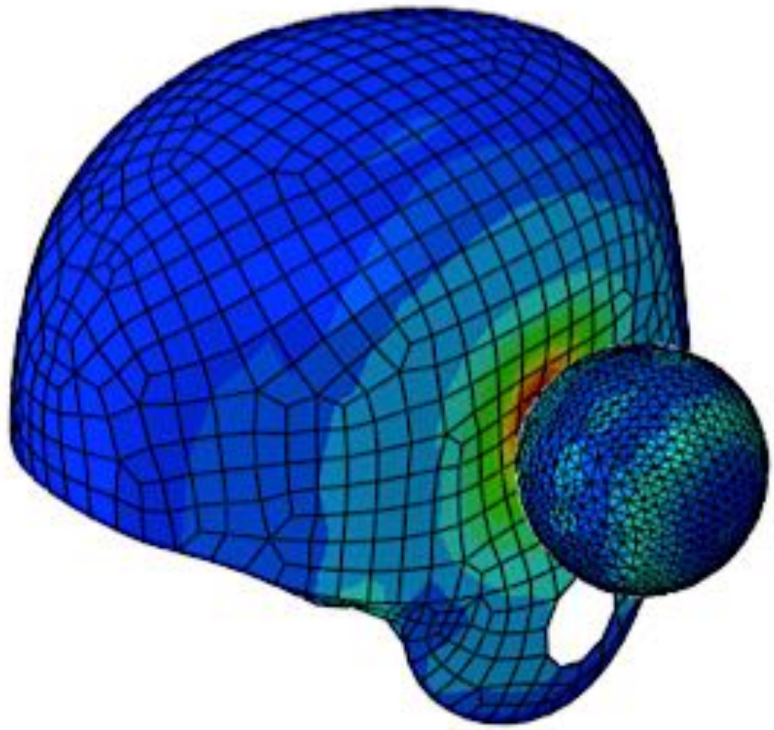

Figure 17: Colored Contour Model After Impact Simulation

The node selected for analysis was in the red region of stress, directly adjacent to the point where the baseball struck the helmet. The stresses and displacements at this node on the helmet were analyzed across time. According to the stresses calculated in Abaqus, the level of stress experienced by the helmet in our simulation did not exceed the yield strength of ABS polymer. Therefore, the helmet is capable of withstanding this magnitude of impact. Although the magnitude of helmet displacement increases over the entirety of the impact, it remains small enough to prevent skull damage at a maximum 6 mm. It is interesting to note that the displacement oscillates periodically.

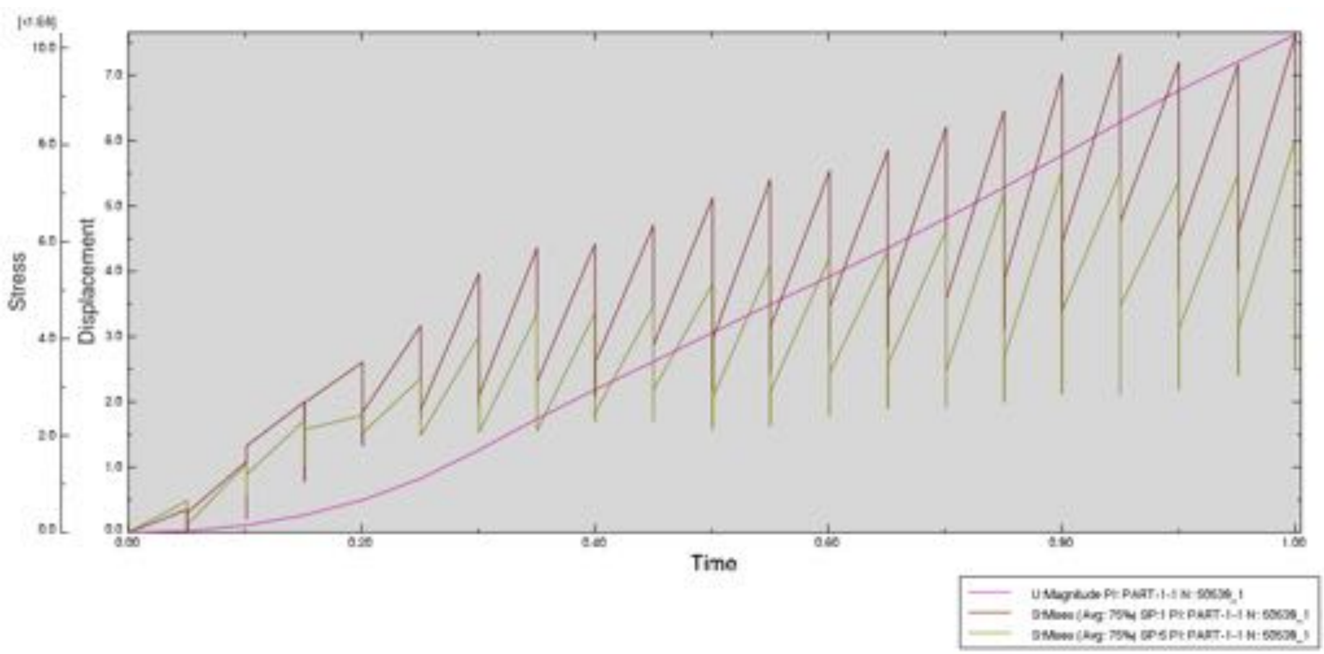

Figure 18: Stress and Displacement vs. Time

Section 7: Summary of Major Findings

By running a dynamic simulation, the stresses were observed to translate in a wave-like pattern. The stresses on the helmet shell dissipated radially outward from the central point of contact where the baseball struck. It was expected to see spikes in the magnitudes of stress around areas of curved geometry or sharp edges. However, it was witnessed that the element stresses lessened the farther the elements were from the contact point – they did not experience a jump in stress at areas of sharper geometry.

It was hypothesized that the main concentration of stress would be confined to the contact area on the baseball where it struck the helmet. However, the stress distribution on the baseball moved unexpectedly in a periodic wave-like fashion. Stresses would oscillate from the contact area to the opposite side, and back again. This was most likely due to the slight elasticity of the woven yarn and rubber-coated cork that made up the baseball. These characteristics allowed it to compress and expand repeatedly when hitting the helmet.

In order to improve the simulation, a number of factors can be changed for the future. First, the meshes could be refined even further to better accommodate the contours and curves of the helmet and baseball. Hexahedral meshes and block meshing would improve the element quality and allow for more accurate calculations to be taken as well.

Additionally, it would be beneficial to account for cyclic loading on the helmet, and in turn, better monitor the player’s health and safety. Players often wear the same helmet throughout the course of a season or even multiple years. Constant exposure to multiple stresses from getting hit by a pitch can plastically deform and weaken the helmet. Analyzing theses stresses over a repeated period of time would help illustrate the lifespan of an MLB batter’s helmet.

Furthermore, with less strict time and monetary constraints, the simulation could be run for a longer period of time. With a longer run-time, the wave-like pattern of dissipating stresses would be allowed to spread across the entire helmet shell surface. Therefore, stresses and strains over the entire helmet can be analyzed, not just those concentrated near the contact region.

Finally, the simulation included a few assumptions and approximations that were not entirely plausible. Not all baseballs will come in perfect perpendicular contact with the surface of a helmet. It is just as likelt for players to get hit in the earpiece, back of the head, or even the front. Pitches often skim the edge of a helmet rather than hitting it head on. They also approach the batter at varying speeds and with rotation – conditions not accounted for in the experiment.

However, the simulation modeled in this report is sufficient because it displays the theoretical situation that results in the helmet experiencing maximum stress. The stress the helmet experienced did not exceed the yield stress of ABS Polymer. Therefore, the helmet is an adequate piece of equipment to protect the player’s from serious head injury.

Section 8: Works Cited

"ABS (Tecaran®)." *ABS*. N.p., n.d. Web. 09 Dec. 2015.

Broderick, Evelyn. "What Materials Are Baseballs Made Of?" LIVESTRONG.COM. LIVESTRONG.COM, 21 Oct. 2013. Web. 09 Dec. 2015.

"Helmet CAD." GrabCAD - CAD Library. GrabCAD, n.d. Web. 3 Oct. 2015. <https://grabcad.com/library/helmet-28>.

"How to Choose a Batting Helmet." *How to Choose a Batting Helmet*. N.p., n.d. Web. 09 Dec. 2015.

Morosi, John P. "MLB Taking Measures to Protect Players with Concussion Symptoms." *FOX Sports*. FoxSports, 14 June 2012. Web. 09 Dec. 2015.

"Thermoplastics - Physical Properties." *Thermoplastics - Physical Properties*. N.p., n.d. Web. 09 Dec. 2015.

Appendix 1

Appendix 2