Modeling and Stress Analysis of Chicken Hip Implants using Mechanical Testing and SolidWorks

Abstract

The objective of this study was to conduct stress-analysis on pre-existing hip implant models and then improve upon the design. Hip implants models are extremely important in the medical field as hip replacement surgeries are conducted frequently. A hip implant consists of three main parts: femoral head, femoral neck, and femoral stem. In this experiment, chicken-scaled 3D-printed hip implants underwent stress testing where the maximum load at failure was determined. Using this load, the stress at failure was determined and statistical testing was performed to evaluate if the samples could withstand 5.09×10^5 Pa of stress on the femoral head. Using a t-test, a p-value of 8.0511e-07 was found, which meant the null hypothesis was rejected, indicating that the samples were able to significantly withstand 5.09×10^5 Pa before failure. Then, a 3D design of a hip implant was created in SolidWorks to improve upon the design of the previously tested implants. It was found that the redesign had a stress at failure of 2.016×10^7 Pa, which was almost ten-fold of the average maximum stress of the 3D-printed models, 2.348×10^7 Pa. This was due to the increased distribution of stress in the redesign. Several features of the redesign allowed for this, including increased thickness of the femoral neck and reduced length of the femoral neck. Ultimately, it was determined that the 3D design created in SolidWorks did improve upon the design of the previous 3D-printed hip implant models.

Introduction

In this experiment, the goal was to act in place of a bioengineer at a medical device company that is responsible for evaluating a hip implant design. In the medical field, total hip replacement is conducted on an extremely frequent basis, specifically over 500,000 times per year in the United States.[1] Modern-day hip replacement relieves pain and improves function in individuals, improving quality of life and allowing patients to remain active for longer. While hip implant technology has steadily improved over the past few decades, advancements have plateaued, and few breakthroughs have been made in the field.

To further elaborate, during hip replacement surgery, the ball and socket joint of the hip are removed along with the femoral head. To replace the femoral head, hip implants are placed in the femur and consist of three different portions: a femoral head, femoral neck, and femoral stem. Each of these portions have a specific purpose and is responsible for a different action during the procedure. Specifically, the femoral head replaces the head of the femur and fits into the pelvic bone, while the femoral stem is inserted into the femur. Once the implant is placed, the femoral head on the implant is able to rotate, replicating the motion of the ball part of the hip joint.

The specific task in this experiment was to evaluate a specific hip implant design and determine whether or not it can withstand 5.09×10^5 Pa of stress on the femoral head. To evaluate this, stress testing was conducted on 3D-printed hip implants that were chicken-scaled and made of ABS plastic. Each engineer in this study tested three samples, and the data in the lab section was pooled to conduct calculations in MATLAB. In the medical field, mechanical testing is often completed to evaluate the strength of implants and to improve upon the design. For example, dental implants undergo stress testing to replicate the bite force of a human.[2] When conducting these tests, the samples are tested until failure, and the forces are repeated to predict survival times of the implants.

Additionally, in this experiment, once the 3D-printed hip implants were tested to failure, SolidWorks was used to create an implant with an improved design. Once the design was complete, simulations were run in SolidWorks to determine the stress at failure of the implant. In engineering research, SolidWorks is a common tool that allows for engineers to test different designs and materials at a low cost. For example, in a study conducted by Abbass et al., SolidWorks was used to conduct biomechanical analysis on hip and knee implants.[3] Specifically, the use of different metallic materials were tested to investigate the differences in stress and deformation distribution depending on material.

In this experiment, the hypothesis is that the implants tested in the lab section will be able to withstand the minimum stress level of 5.09×10^5 Pa. While the hip implants may be 3D-printed models made of different materials, they are designed to be representative of models used for commercial use, and therefore should be able to handle the compressive load. In terms of the model designed in this experiment, the hypothesis is that the design will perform better than the implants tested by the class. This is based on the idea that the design will improve upon the areas where the existing model failed as the engineer will have hands-on experience testing the existing models. Additionally, the use of SolidWorks allows for an extremely precise design to be created and tested, leading to a more polished design.

Materials and Methods

Stress Testing of Chicken Hip Implants

To test previously designed hip implants, three copies of 3D-printed hip implants were used. To conduct these tests, a Thwing-Albert Universal Mechanical Testing Instrument was utilized to test the hip implant to failure. When testing the samples, the hip implants were placed in a clamp, which allowed for the force to be applied to the neck of the implant. To test the sample, the machine was lowered until it nearly touched the hip implant and was then turned on. The samples were tested until failure and the maximum compression force for each sample was recorded.

Stress Analysis in MATLAB

Once each student conducted compression testing on the hip implants, a dataset was created containing the maximum force for each sample in the lab section. Then, using MATLAB R2023a, the stress at failure was calculated for individual samples and the class dataset. To do so, the class dataset was copied into MATLAB as a vector. Then, the force parallel to the cross-sectional area of the neck was calculated by finding the vertical component of the compression force. This involved multiplying the compression force by cos (a), where a is the angle between the neck and femoral stem, which was 43.5°. Afterwards, the cross-sectional area of the neck was found by using the area of a circle, $A = \pi r^2$, where r was 2.5 mm. Then, the stress at failure was able to be calculated using the equation for stress, $\tau = \frac{F}{A}$. The descriptive statistics were then calculated for each dataset, using the functions: mean(), median(), and std(). Additionally, a normal probability plot was created for the stress values for the class dataset. Next, a

1-sample t-test was conducted to determine whether the samples withstand 5.09 x 10^5 Pa using the ttest() function, where only the upper tail was tested and a confidence interval was found.

Moreover, a file of tensile test results on a 3D-printed implant was provided. Using this dataset, the stress and strain on the implant were determined and a stress-strain curve was created. To find the stress, the force values were divided by the cross-sectional area, which was the same as the earlier part of this experiment. Then, to find the strain values, the change in length from the initial length was calculated for each data point. Finally, the stress and strain vectors were plotted to create a stress-strain curve.

3D Modeling of Chicken Hip Implant in SolidWorks

The next part of this lab involved creating a 3D design of a hip implant to improve on the design of the previously tested implants. To create this design, SolidWorks 2023 was utilized. First, a sketch was made on the front plane and revolved around the y-axis using the revolved boss base tool. This created the femoral stem, which was 2.5 cm long, and the radius varied from 1.5mm at the bottom to 2.5mm at the top. Next, another line was sketched on the front plane which extended from the center of the top of the femoral stem to the presumed location of the femoral head. Using the swept boss base tool, this line created the femoral neck, which had a length of 1.2cm and a radius of 5mm. While creating the femoral neck, a gap was still present between the femoral neck and the stem, so a semi-circled was sketched to fill the gap on the femoral stem. This semi-circle was then revolved to meet the bottom face of the femoral neck. Next, a plane was created that offset 3mm from the top face of the femoral neck. On this plane, a semi-circle was sketched with a diameter of 9mm. Finally, the semi-circle was revolved using the revolve boss base tool and the femoral head of the hip implant was created. Once the design was finalized, it was inspected to ensure all dimensions fit the required parameters and was saved as a SolidWorks part file.

Simulation of Chicken Hip Implant in SolidWorks

Once the hip implant was created in SolidWorks, a simulation was run to determine if the new design improved upon the previous one. To run this simulation, a coordinate system had to be created using the front, top, and right planes as axis. Then, a new study was created, which was nonlinear and dynamic, and was then named. Under the simulation, the material of the part was changed to ABS plastic to match the previous hip implants. Afterwards, fixed geometry was selected under fixtures under the simulation name. For this part, the edges where the femoral neck met the femoral stem were selected as fixed. Next, force was selected under external loads. The force was applied normal to the femoral head, and the average maximum force from the lab section was used. Under the force settings, per item was selected, linear was selected under variation with time, and nonuniform distribution was selected. Additionally, the coordinate system created was selected and the equation was edited to where the direction of the applied force to the femoral was multiplied by 1, which was the y-axis. The other two directions were multiplied by 0. Next, the result options were defined, where specified solution steps was selected. For solutions steps, the start was 1, the end was 250, and the increment was 4. Finally, a mesh was created using standard mesh and the software-generated numerical values.

The simulation was then able to be run. Under the simulation tab, run this study was selected. Once the simulation was complete, regions of stress were indicated by a color legend. Once the simulation was complete, the stress entry under results was selected and the animate button was pressed. To create an animation video, the stop button was pressed and reciprocate was selected. The speed of the animation was then slowed down to three frames. The animation was saved as an AVI file, which was created once the play button was selected. Once the animation was exported, screenshots were taken of the simulation results to view regions of the stress.

Results

Dataset	Individual Samples	Lab Section
Mean (Pa)	2.4592e+06	2.3481e+06
Median (Pa)	2.2757e+06	2.2591e+06
Standard Deviation (Pa)	4.0046e+05	5.2871e+05

Table 1: Descriptive statistics for stress at failure for 3 implant samples and the lab section.

The descriptive statistics were found for the stress at failure for the three individual implant copies and the dataset for the whole lab section (Table 1). From these statistics, it can be observed that the mean and median is slightly higher for the individual samples when compared to the lab section's data. In terms of the standard deviation, a larger standard deviation was present for the lab section's data, indicating that larger variation was present. Although, as the individual samples were essentially a subset of the lab section's data, only slight differences can be observed.

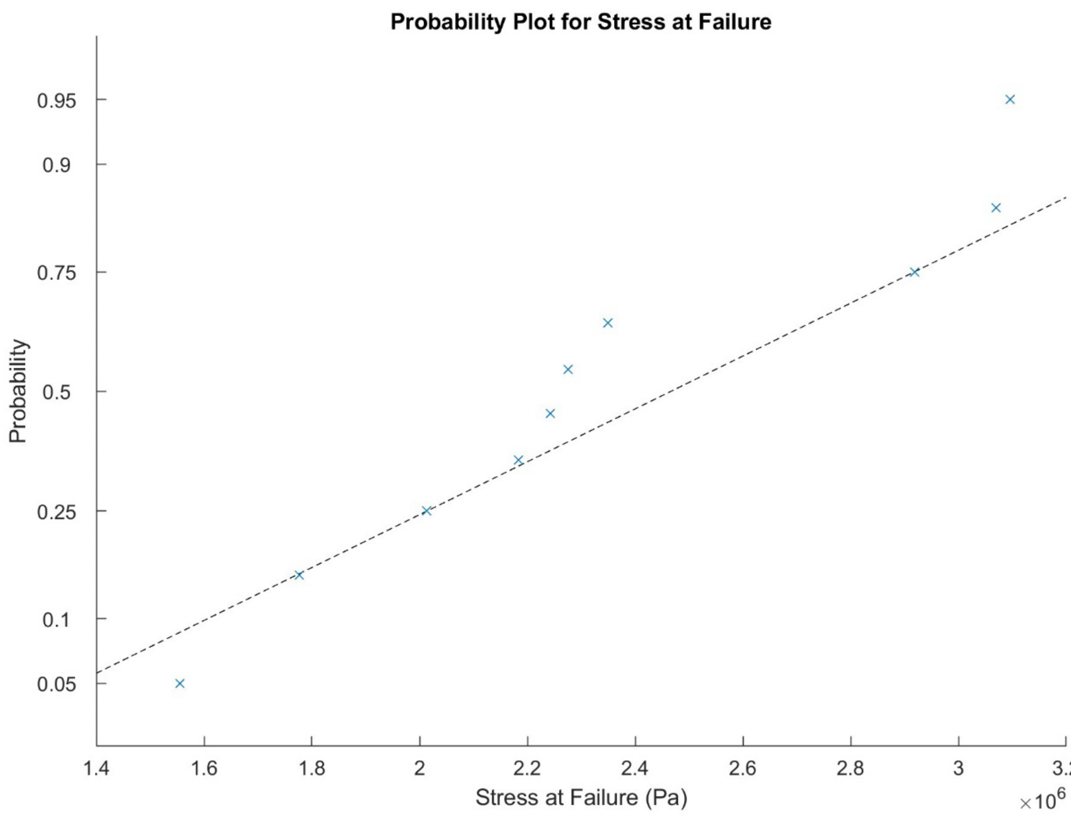

Figure 1. Normal probability plot for stress at failure for lab section data.

A normal probability plot was created for the stress at failure for the lab section's data to observe if a normal distribution was present (Figure 1). While the points seem to roughly form a linear pattern, there are several points that appear to deviate from the line. These deviations seems to occur towards the tails of the central line of distribution. To further understand if the data was normally distributed, a chi-squared goodness of fit test was completed, which the data failed to reject the null hypothesis. These results indicate that the data is normally distributed.

In this lab, a 1-sample t-test was also conducted to determine whether the implant design was capable of withstanding 5.09 x 10^5 Pa before failure. This t-test was completed for the both the individual samples and the lab section's data. For the three individual samples, a p-value of 0.0069 was found and a confidence interval greater than 1.7841 x 10^6 Pa. With an alpha value of 0.05, the results indicate that the null hypothesis was rejected, and that the individual samples were able to significantly withstand 5.09 x 10^5 Pa before failure. For the lab section's data, a p-value of 8.0511e-07 was found and a confidence interval greater than 2.0416 x 10^6 Pa. Once again, the null hypothesis was rejected, indicating that the samples were able to significantly withstand 5.09 x 10^5 Pa before failure.

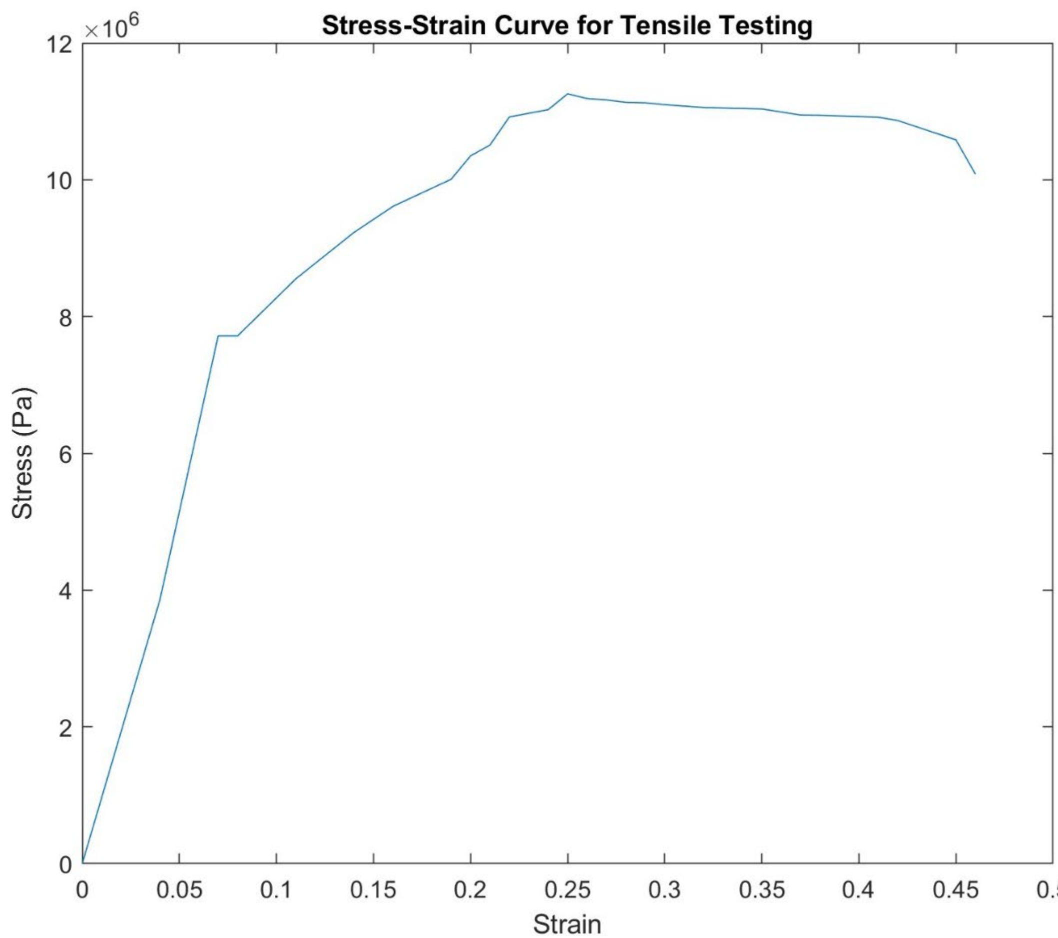

Figure 2. Stress-strain curve for tensile testing results on the neck of a 3D-printed hip implant.

A stress-strain curve was created using the tensile testing dataset provided. In this stress-strain curve, elastic deformation was present until the strain was 0.07 and the yield stress was 7.7164 x 10^6 Pa. After elastic deformation, plastic deformation was present until the point of fracture. The ultimate strength point occurred at a strain of 0.25 with a stress of 1.1257 x 10^7 Pa. Finally, the fracture point occurred at a strain of 0.46 and a stress of 1.0084 x 10^7 Pa.

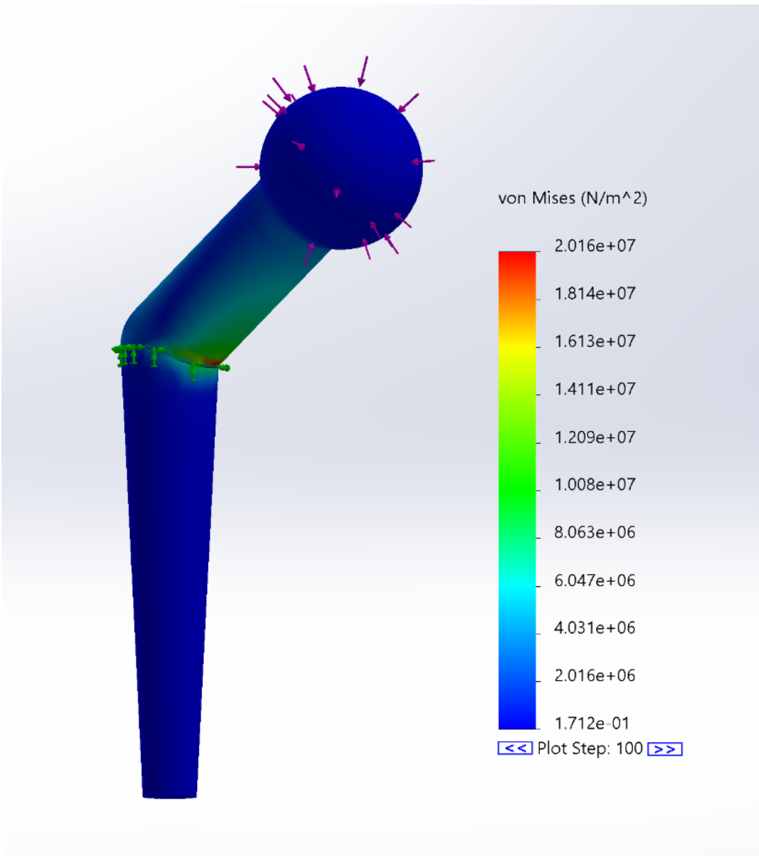

Figure 3. Simulation results for a hip implant created in SolidWorks. Von Mises are equal to Pascals.

A simulation was conducted on the 3D design of the hip implant created in SolidWorks (Figure 3). In this simulation, it can be observed that a majority of the stress was placed upon the femoral neck, specifically, where it approaches the femoral stem. At the point of intersection between the femoral stem and neck, the stress reaches 2.016 x 10^7 Pa. Along the femoral neck, the stress ranges from 1.411 x 10^7 Pa to 6.047 x 10^6. The stress becomes less concentrated as the femoral neck approaches the femoral head. In the animation for the simulation, it can be observed that the model does not compress enough to restore plastic deformation as it returns to its initial shape.

Discussion

In this study, an analysis was completed on the lab section's stress at failure values for the 3D-printed hip implants. To determine if the data was normally distributed, a normal probability plot was created (Figure 1). This plot indicated that the data is normally distributed, as the data did not significantly deviate from the central line of distribution. Additionally, a chi-squared goodness of fit test was conducted, confirming a normal distribution. As a normal distribution was present, this justified the use of a single-sample t-test to analyze the data. Moreover, the data met the rest of the requirements for a single-sample t-test including the dependent variable being continuous and the observations independent of one another.

As stated previously, a single-sample t-test was used to determine whether the implant design was capable of withstanding 5.09×10^5 Pa before failure. For the lab section's data, a p-value of 8.0511e-07 was found and a confidence interval of any stress value greater than 2.0416×10^6 Pa. As the p-value was less than alpha, 0.05, the null hypothesis was rejected, indicating the class's samples were able to significantly withstand 5.09×10^5 Pa before failure. Additionally, the mean stress at failure for the lab section's data was 2.3481×10^6, which was within the confidence interval. These results supported the initial hypothesis that the tested implants would be able withstand the minimum stress level of 5.09×10^5 Pa. While the hypothesis was supported, there are several potential sources of error that may have been present. One potential source of error may be disparities in the 3D printing of the hip implants, including variations in the mechanical properties of the models. Additionally, when mechanical testing was conducted, the hip implants may not have been perfectly aligned or placed within the clamp correctly. Although, variations within the lab section's data were minimal, indicating that these potential errors did not have a large impact on the results.

Moreover, in this study, a stress-strain curve was produced using tensile testing data of the 3D-printed hip implant models (Figure 2). In the curve, elastic deformation was present until a strain of 0.07, where it transitioned into plastic deformation before its fracture. At the upper yield point, stress was 7.7164×10^6 Pa, while at the ultimate strength point stress was 1.1257×10^7 Pa. To design an implant with a larger upper yield point, several modifications can be made to the design. Since the femoral neck is an area concentrated in stress, its thickness can be increased, and its length decreased to allow for more strength. Additionally, the angle the neck makes with the testing surface can be increased, which would result in the downward component of the force to be reduced.

Finally, the computer software SolidWorks was used to redesign the hip implant using the information gained from the stress analysis of the 3D-printed models. To determine if this design would resist breaking more than the 3D-printed implant design, a simulation was run in SolidWorks (Figure 3). In this simulation, it was determined that the maximum stress this design had was 2.016×10^7 Pa, which was almost ten-fold of the average maximum stress of the 3D-printed models, 2.3481×10^7 Pa. These results indicate that the new design offers increased resistance to breaking due to stress in comparison to the 3D-printed models. In the redesign, the goal was to reduce the stress concentration present on the intersection of the femoral neck and femoral stem. Several features of the redesign allowed for this, including increasing the thickness of the femoral neck and reducing the length of the femoral neck. These changes to the femoral neck led to a wider area to distribute the stress in the model. Additionally, the angle between the femoral neck and the top of the femoral stem was increased, reducing the stress concentration on the femoral neck. While the redesigned hip implanted offered an improved model, several sources of error may have been present in this study. First, since SolidWorks was used in this study, the simulated properties of ABS plastic may be inaccurate and not true to the 3D-printed hip implants. Additionally, SolidWorks' simulation studies are based on assumptions and are conducted in an environment with no external factors. This means that SolidWorks' measurement method for stress simulations may have small inaccuracies due to material properties and the application of the load, leading to differences in true stress values. Ultimately, these potential sources of error may lead to variability in the data, specifically inaccurate stress values, requiring for future studies to be conducted to further support the results found in this experiment.

Conclusion

The objective of this study was to conduct stress-analysis on pre-existing hip implant models and then improve upon the design. In this experiment, chicken-scaled 3D-printed hip implants underwent stress testing to evaluate if the samples could withstand 5.09×10^5 Pa of stress on the femoral head. Using a t-test, it was found that the samples were able to endure this level of stress. Afterwards, a 3D design of a hip implant was created in SolidWorks to improve upon the design of the previously tested implants. It was found that the redesign had a stress at failure ten-fold of the average maximum stress of the 3D-printed models. This was due to the increased distribution of stress in the redesign as the thickness of the femoral neck was increased and the length of the femoral neck was reduced.

In the future, this study can be expanded in several ways to further improve upon the hip implant design. To begin, more trials can be conducted to further support the results in this experiment. This may include physically printing the model designed in this experiment and conducting mechanical testing on it. Additionally, the experiment can develop to designing a human-sized chicken hip implant instead of a chicken-scaled one. This can be completed through SolidWorks, where different designs and materials of the hip implant can be tested.

References

1. Ferguson RJ, Palmer AJ, Taylor A, Porter ML, Malchau H, Glyn-Jones S. Hip replacement. *The Lancet*. 2018;392(10158):1662-1671. doi:10.1016/S0140-6736(18)31777-X

2. Bonfante EA, Coelho PG. A Critical Perspective on Mechanical Testing of Implants and Prostheses. *Adv Dent Res*. 2016;28(1):18-27. doi:10.1177/0022034515624445

3. Abbass SJ, Abed RI. Mechanical analysis of hip & knee implants regarding different activities conditions and different materials. *Materials Today: Proceedings*. 2021;42:2193-2201. doi:10.1016/j.matpr.2020.12.304