FINITE ELEMENT ANALYSIS OF HIP IMPLANT UNDER DIFFERENT LOADING CONDITIONS

By: Zajiba Sadia Islam, 501279357

Program: MEng in Biomedical Engineering, Course: Finite Element Methods in Engineering, AE8115/ME8115, Spring 2024.

Table of Contents

Abstract

Hip replacement surgeries, essential for restoring mobility and alleviating pain in patients, often involve the implantation of a hip prosthesis. This study focuses on the finite element method (FEM) analysis of hip prostheses to understand their mechanical behavior under various loading conditions. The objectives include creating a 3D model of the prosthesis, performing FEM analysis to determine stress and deformation distributions, identifying potential failure points, and validating simulated results with actual data. The prosthesis model, developed using AUTOCAD 2024 and analyzed in ANSYS 2023 R2, was subjected to walking and jogging loads representing normal and strenuous activities respectively. The material considered for the model simulation was Titanium alloy, known for its common use in hip prosthetics. In the study, during walking, a pressure of 5.26 MPa was applied, while jogging induced a pressure of 8.41 MPa. Two meshing scenarios with element sizes of 10mm and 1mm were evaluated for convergence. Results showed a significant reduction in maximum von Mises stress from 161 Pa to 109 Pa as the mesh was refined, with a relative difference of 32%. The computational cost, measured by simulation time, increased with finer meshes. Validation against experimental data demonstrated close agreement, highlighting the accuracy of the FEM analysis. The study concludes that FEM is a robust tool for analyzing the mechanical performance of hip prostheses, providing insights into stress distribution and potential failure points. Future work may include finer meshing and consideration of additional activities to enhance the model's accuracy and applicability.

Introduction

Hip replacement surgeries are standard surgical procedures that can help patients with regaining of mobility and pain relief. The surgeries can be of two types, depending on the age and condition of the patient: partial hip replacement and total hip replacement. The femoral head prosthesis is one of the most critical components in hip replacement surgeries, which are designed to replace the damaged or diseased femoral head. This study aims to gain practical experience in analyzing the mechanical behavior of hip prostheses. Understanding the stress distributions within the prosthesis can lead to improved designs, enhancing the longevity and performance of the implants. Identifying the areas where maximum deformity occurs due to loading can help identify the points of failure of the prosthesis and thus help to improvise them. The finite element method (FEM) analysis plays a crucial role in this regard. FEM can be a powerful computational tool used to simulate and study the mechanical behavior of the hip prosthesis under various loading conditions. For this study, walking has been considered a normal activity and jogging has been taken as a stressful activity. FEM method can be implemented to calculate the stress and deformity points of the prosthesis. In this regard, software such as Ansys can be very effective and efficient. Results obtained from the simulation can be utilized to optimize the design of the model, ensuring that the prosthetics can withstand the repetitive and high-stress environment of the human hip joint. This is particularly vital in preventing implant failure and recurrent revision surgeries. The modeling of the prosthesis is a key factor in determining the results. The stress and deformation points may vary depending on the design and material choice of the prosthesis. Since material of the prosthesis can have drastic influence on the longevity of the prosthesis, a standard, non-corrosive material like Titanium alloy has been chosen for the prosthesis. How the femur head is in contact with the stem of the prosthesis also affects the results. During the analysis in Ansys, the type of the mesh applied to discretize can have significant effects on the results. In this study, attempts have been made to explore how the difference in mesh density, as well as different loading conditions can affect the result outcome. Various studies have previously been conducted to measure the actual stress that occurs in the hip implants during various activities of daily life. Finally, in order to validate the results of the experiment, the obtained results have been compared with the actual prevailing data to see if there are any significant discrepancies.

Objectives

1. To create a 3D model of the femoral head prosthesis.

2. To perform finite element analysis to determine the stress and deformation distribution regions.

3. To identify potential failure points under various loading conditions.

4. To evaluate different elements for the prosthesis analysis and compare between them.

5. To validate the simulated result by comparison with actual data.

Loading and Boundary Conditions

In this study, the mechanical impact of functional loading to the hip prosthesis is to be explored.

For conducting the FEM analysis, the lower part of the implant stem has been considered as fixed support. This is the boundary condition of the study. The head of the implant is in complete contact with the stem in all simulation scenarios. The stem and head are both made of titanium, having Young's modulus of 110 GPa and Poisson's ratio of 0.33.

For this study, the impact of walking and jogging on the hip prosthesis for a person with an average weight of 75kg has been considered. Walking is considered normal activity while jogging is more physically demanding.

During walking, average pressure experienced at the femoral head has been calculated as 5.26 MPa and during jogging, this pressure can be increased significantly, to an average value of 8.41MPa.

The resultant force values for walking and jogging loading conditions for an average person of 75kg can be shown below:
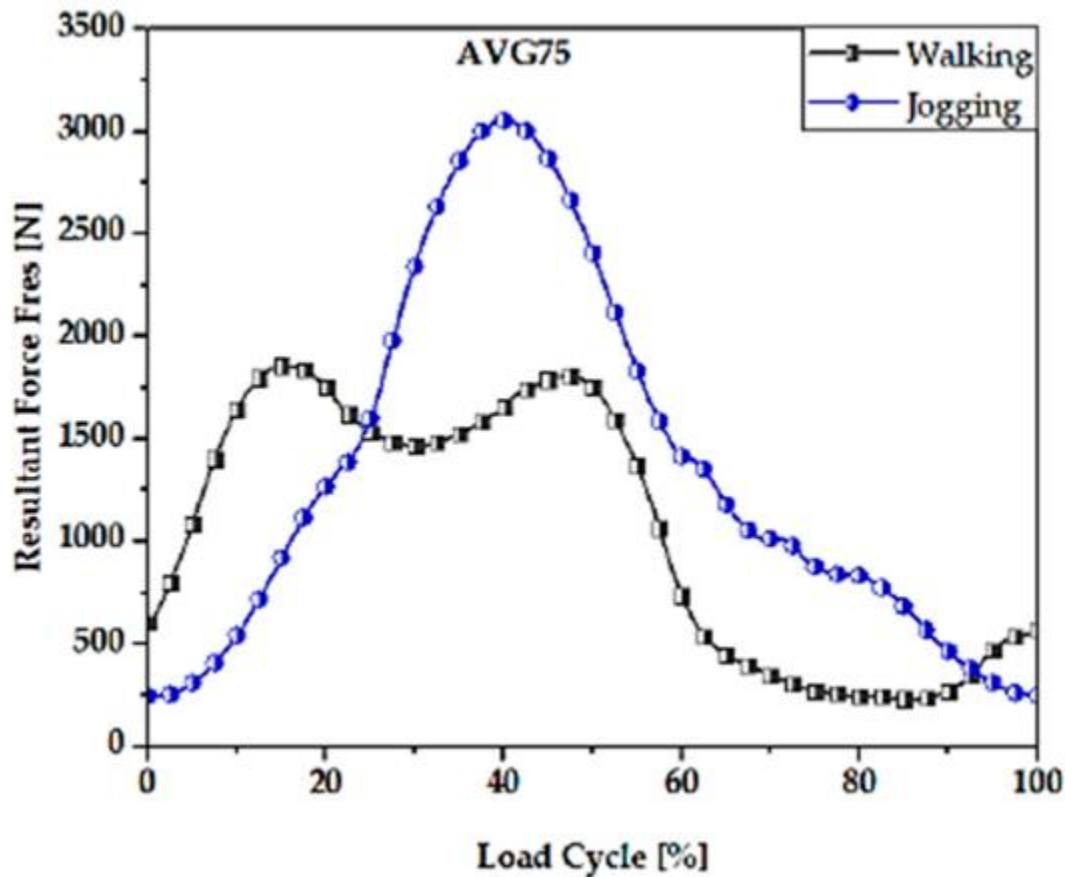

Image: Resultant force values for walking and jogging conditions [1]

Material Properties

The standard materials for the prosthesis commonly include titanium alloys for the stem and cobalt-chromium alloy for the head. These are the most used materials for hip replacement prosthetics.[2] The mechanical properties (e.g., Young's modulus, Poisson's ratio, yield strength) of these materials are obtained as such:

	Elastic Modulus (GPa)	Yield Strength (MPa)	Ultimate Strength (MPa)	Density (kg/m^3)
Ti6Al4V	110	910	1000	4400
Co-Cr	220	840	1280	8500

Table: Table showing properties of various materials.[3]

However, for simplifying, the model of the implant has been made of titanium alloy for both the head and the stem.

Model Creation

A 3D model of the femoral head and stem prosthesis is created using AUTOCAD 2024 software. The geometry of the model is based on standard prosthesis designs and anatomical considerations. The 2D geometry of the model is shown below:
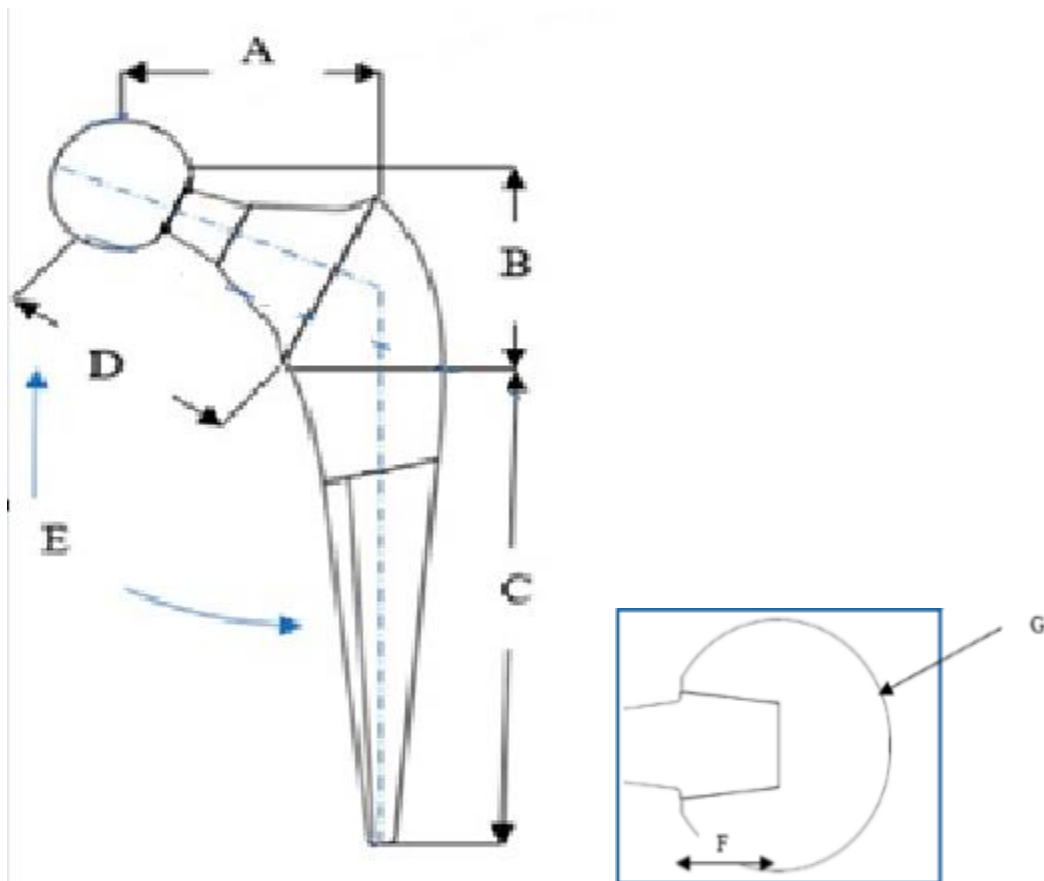

Image: 2D geometry of the model [1].

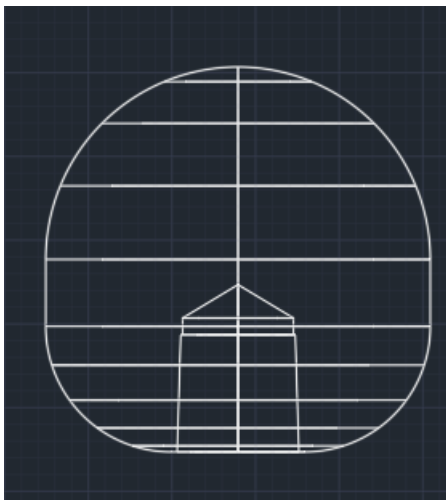The measurements of the various parts of the model are as follows:

A= 39.5mm, B=32 mm, C= 108 mm, D=42mm, E= 135-degree, F= 11, G= 30mm.

In the model, the head is in complete contact with the stem.

3D Modeling the implant stem in AUTOCAD

Modeling the prosthetic head in AUTOCAD:

Overall 3D model of the entire hip implant:

Analysis

For this study, analysis was performed with ANSYS 2023 R2. The 3D model was created using Autocad 2024 and later exported to ANSYS for the analysis. The mesh used to discretize the 3D into tiny tetrahedral elements was of 10mm and 1mm respectively. Pressures were applied on the head of the implant, perpendicular to it. The stem of the implant is considered as fixed support.

Meshing

Element size has been taken as 1mm and 10mm. The element shape is tetrahedron.

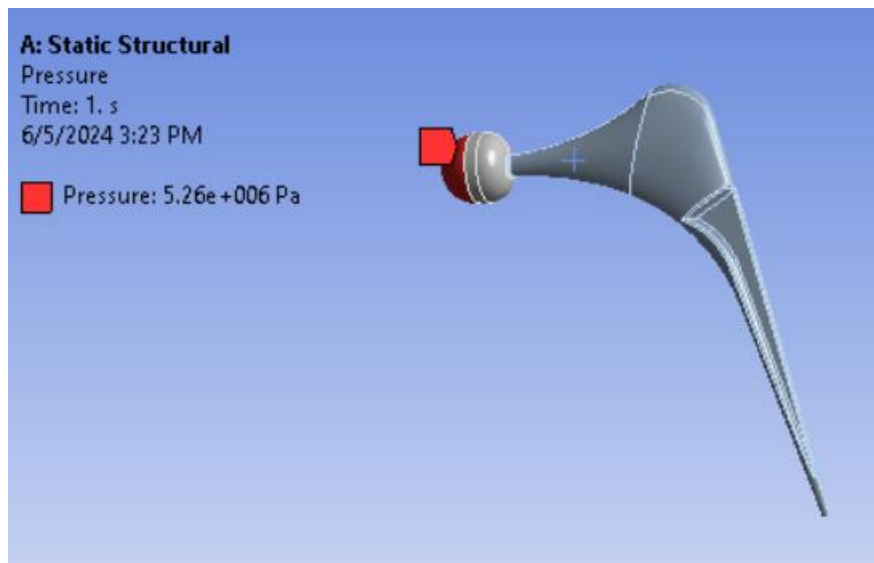Tetra 10mm mesh: Tetra 1mm mesh:
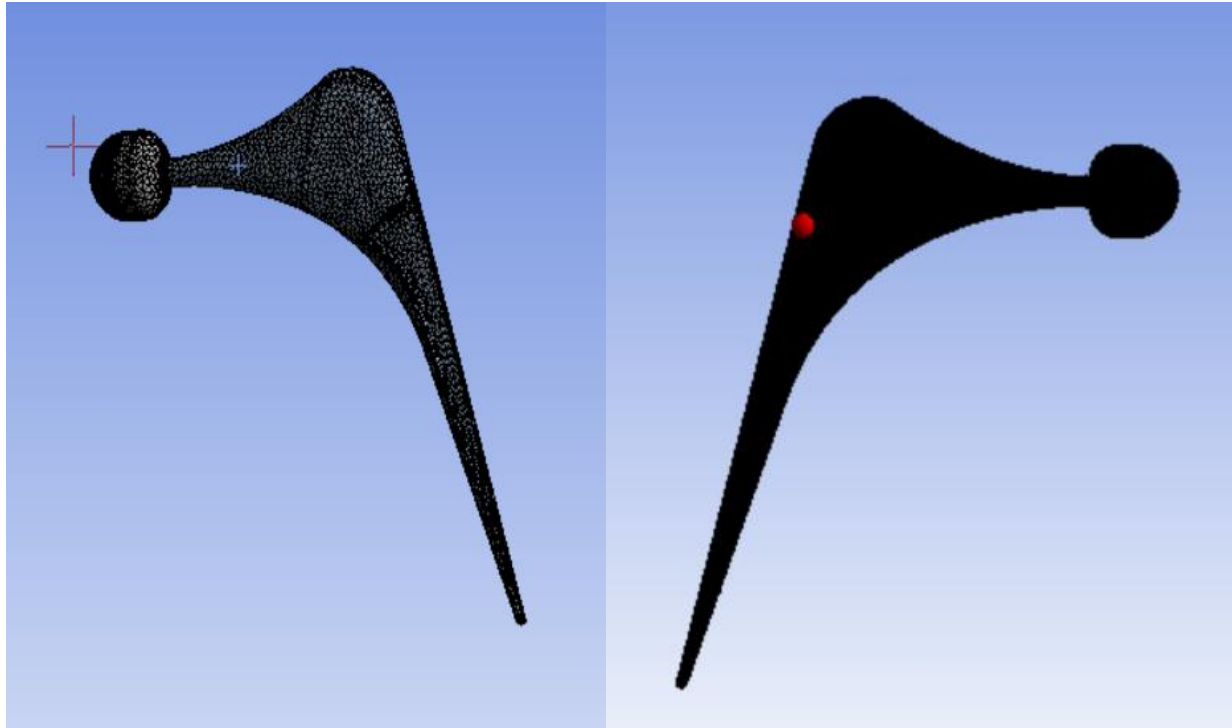
Results

Load Condition: Walking.

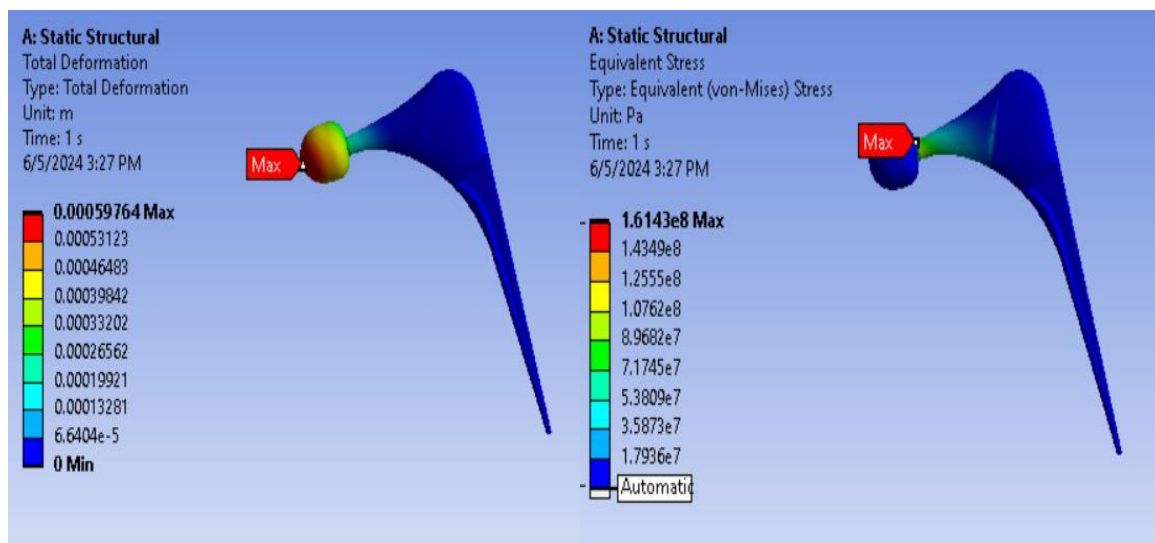Material of Implant: The entire prosthesis (head and stem) has been made of Titanium alloy.

Pressure: During walking, an average person of 75kg experiences an impact force of about 2.5 times their body weight. Hence, the impact force is about 1840N. Considering the contact area at the femoral head to be 3.5cm^2, the assumed pressure at the femoral head during walking has been calculated as 5.26 MPa.

Element size:10mm

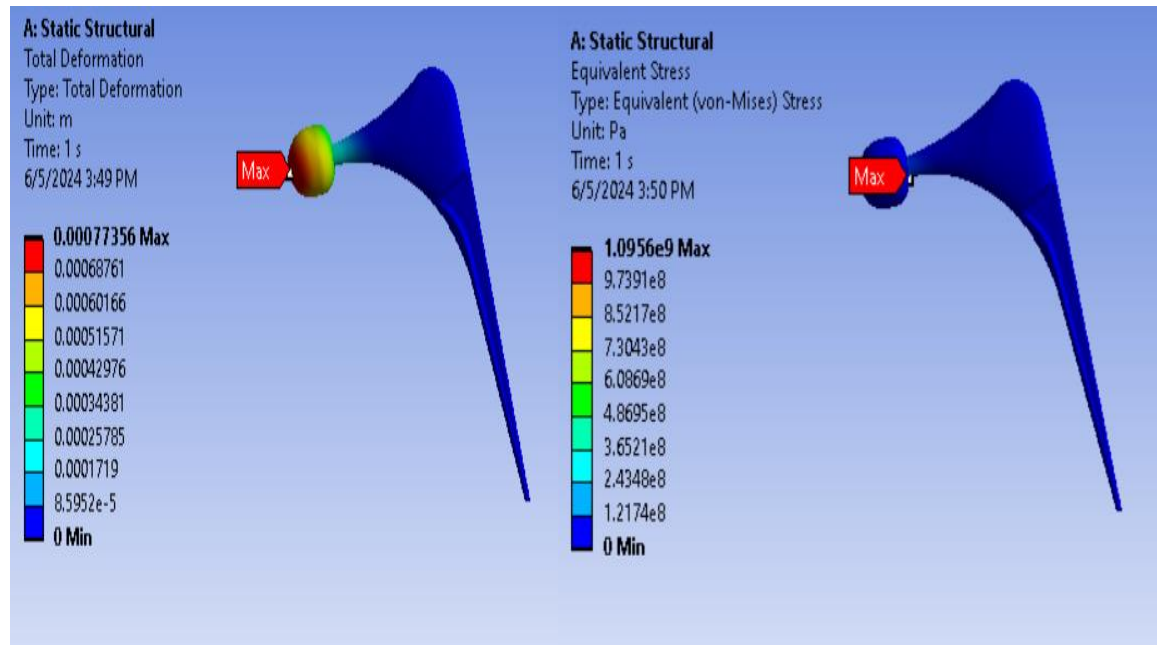Total Deformation: Equivalent Stress:

Element size: 1mm

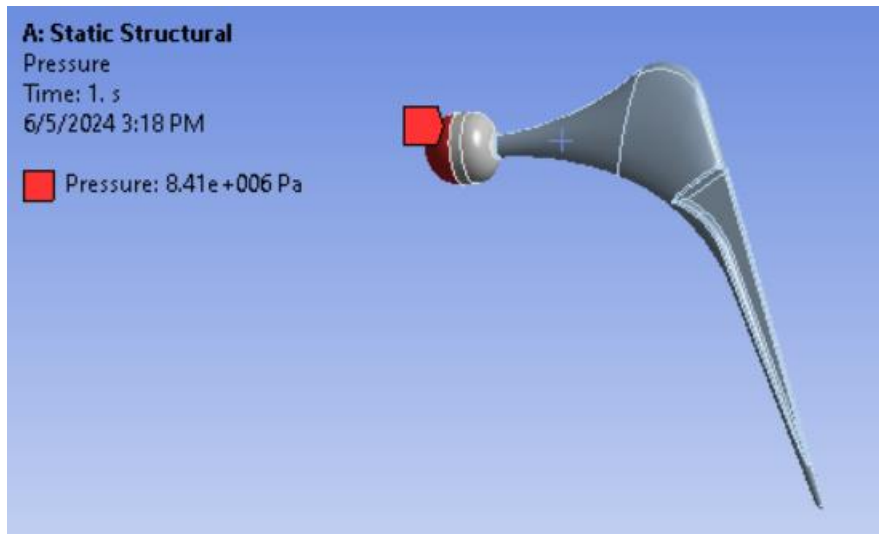Total Deformation: Equivalent Stress:

Here, maximum deformation occurs at the head, while the neck of the implant experiences maximum stress. Hence, the neck is the region for potential mechanical failure to occur. Differences can be seen in the amount of stress and deformation with the difference in meshing, but the region remains the same. Finer meshes provide more accurate results.

Load Condition: Jogging.

Material of Implant: For simplicity, the entire prosthesis (head and stem) has been made of Titanium alloy.

Pressure: During jogging, an average person of 75kg experiences an impact force of about 4 times their body weight. Hence, the impact force is about 2943N. Considering the contact area at the femoral head (3.5cm^2), the assumed pressure at the femoral head during jogging has been calculated as 8.41 MPa.
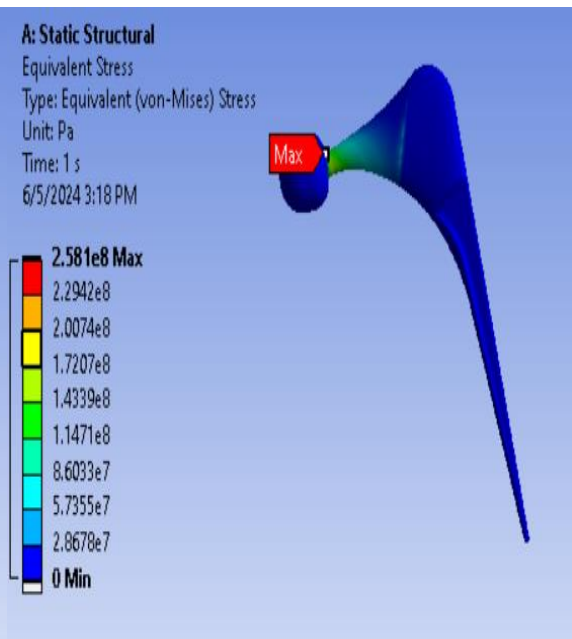
Element size: 10mm
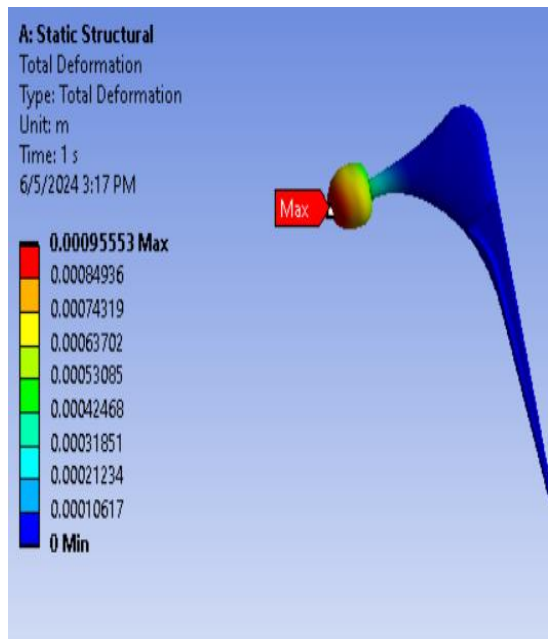
Total Deformation: Equivalent Stress:

Element size: 1mm

Total Deformation: Equivalent Stress:
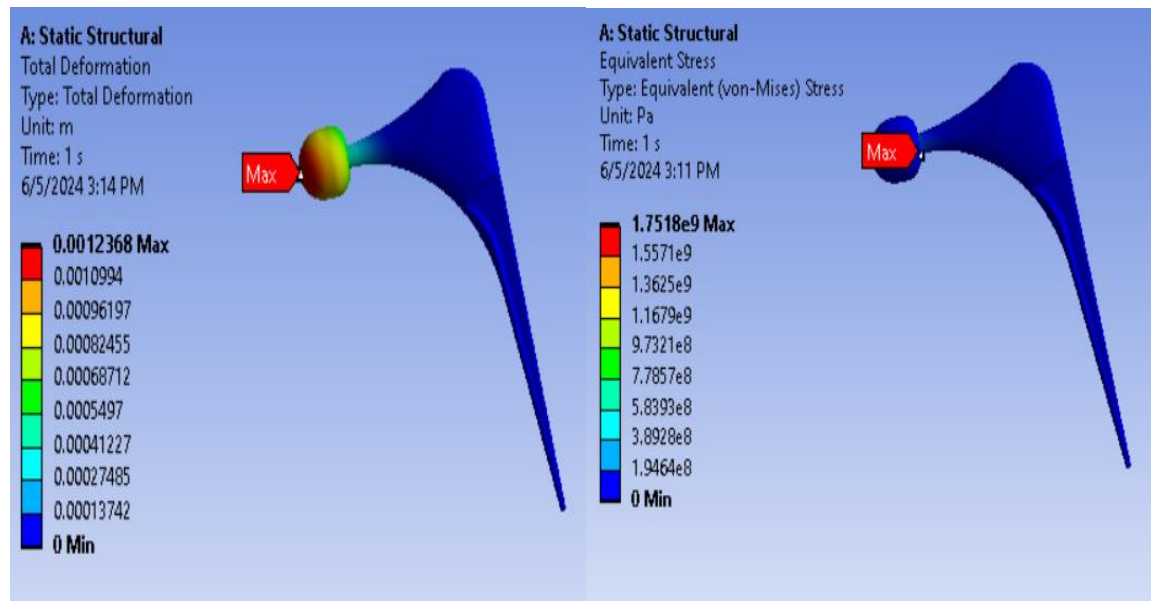
Like walking, maximum deformation occurs at the head, while the neck of the implant experiences maximum stress. Hence, the neck is the region for potential mechanical failure to occur. Differences can be seen in the amount of stress and deformation with the difference in meshing, but the region remains the same. Finer meshes provide more accurate results.

Hence, change in the amount of load has no effect on the area where maximum deformation and stress occurs.

Convergence Study

In this convergence study, evaluation has been done on the accuracy and reliability of the numerical simulation. For this, examination of how the result converge as the mesh is refined has been observed. The model of the hip prosthesis here has been subjected to a load of 5.26 MPa normally to simulate the loading condition during walking. Here, 2 cases of meshing have been done.

Mesh 1: Tetrahedral mesh with element size of 10mm.

Mesh 2: Tetrahedral mesh with element size of 1mm.

Criteria:

The primary area of interest to observe the convergence is the maximum von Mises stress [4] in the implant model. It is based on the successive difference in the von Mises stress between successive mesh refinement. The relative difference is calculated as:

$$\Delta\sigma = \frac{\sigma i + 1 - \sigma i}{\sigma i} * 100\%$$

The maximum von Mises stress values are shown below in the table:

Mesh	Element Size	Maximum von Mises Stress (Pa)	Relative Difference
Mesh 1	10mm	161	
Mesh 2	1mm	109	32%

The computational cost in this regard has been defined as the simulation time. For each mesh refinement, the simulation time has been increased significantly. The table below shows the simulation time needed for each mesh:

Mesh Type	Element Size	Simulation Time (Min)
Mesh 1	10mm	4
Mesh 2	1mm	10

The convergence study demonstrates that the finite element model of the femoral prosthesis produces increasing accurate and reliable results when an appropriate mesh density is used. With this, the computational cost should be considered while applying the mesh.

Limitations: Due to limitations of the simulation software Ansys, meshing could not be performed with element size smaller than 1mm.

Validation of the Result

For validating the result of the FEM analysis, comparison has been made with actual patient data. In this regard, the values of different regions of the implant during walking with tetrahedron element of 1 mm has been considered. The actual patient data has been obtained from a previous study [5]. The data present in Pa unit has been converted to MPa unit.
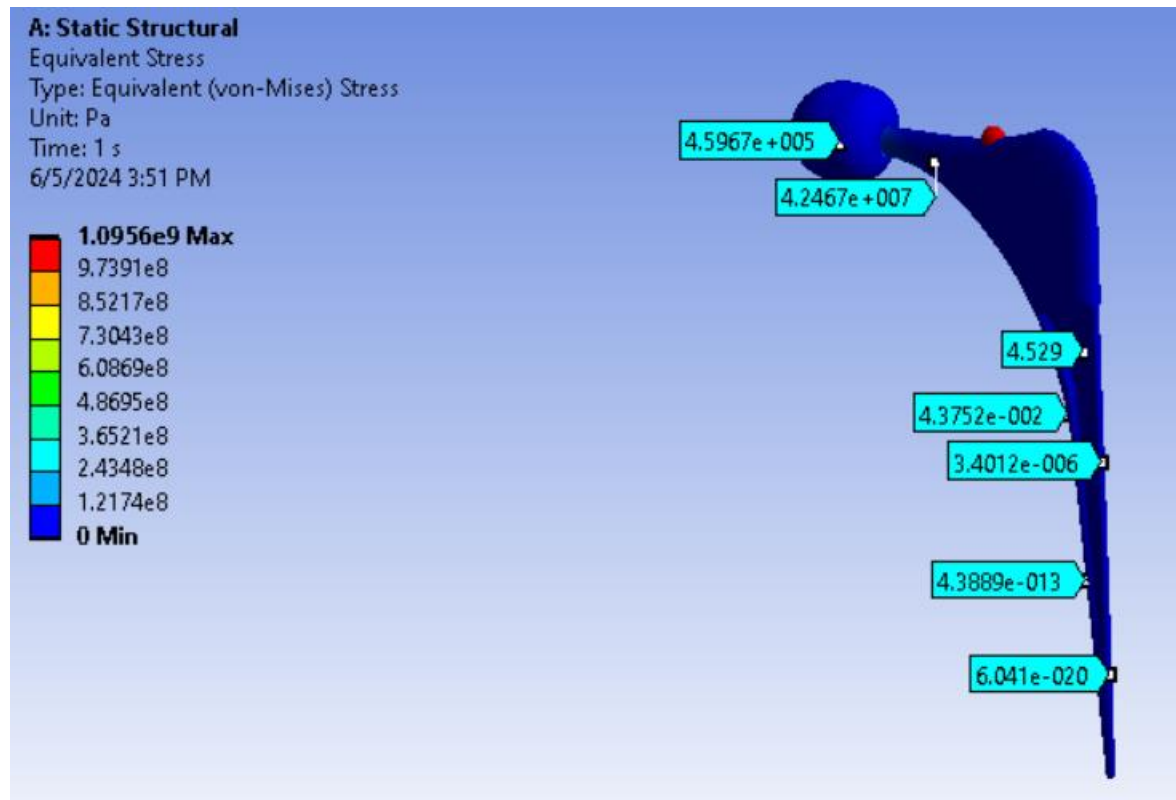
Stresses	Experimental	FEA
S1	4.913	4.59
S2	3.9142	4.24
S3	3.66	3.40
S4	5.85	6.04
S5	4.43	4.37
S6	5.18	4.53
S7	4.88	4.38

Table: Comparison between the actual and simulated data.

Interpretation: Comparing between the results of the stress analysis during walking and the actual experimental value obtained, it has been seen that the results in the simulation are in close agreement to the actual available data.

Limitations and Differences affecting the Experimental Results

- The model of the hip implant is different from the actual implant that was used to conduct the study. Thus, the values are affected by the differences in the design.
- The mesh applied in the simulation model has elements of 1mm size. The finer the mesh, the better the similarities between them.
- The exact point of stress could not be determined, and approximate positions have been selected.

Future Expansions of the Study

This study can be further expanded to include even finer meshing. Due to limitation of the current software, the element size could not be lowered than 1mm. In future work, the effects of finer meshing and element types can be observed more clearly and accurately.

The model of the prosthesis can be modified to account for the distance of contact between the stem and head of the femoral prosthesis. Since the distance and area of contact between the head and stem affects the amount of stress and deformation[6], it can be considered in the future studies to observe the effects.

 Different loading conditions involving the activities of daily living can be considered to see the effect, and potentially identify failure point of the prosthesis. In this regard, activities that involve rotation of the hips can be specially considered and studied further.

Conclusion

Overall, FEM is a powerful method of analyzing strain and stress acting on a body. In this study, the FEM analysis of the hip implant provided valuable insight into the amount of stress and deformity experienced by the implant during normal activity like walking and stressful activity like jogging. It also provided an idea of the area of the prosthesis where mechanical failure is likely to occur. The effect of the meshing density was also a prime focus of this study, and a convergence study was conducted to observe changes in the results with certain limitations. The comparison between the actual data of stress obtained from experiments in a different study and the simulation stress occurring in different regions of the implant showed there were no significant difference between the two. This provided valuable insight and established relevance of the whole study with the practical implementation. In future, this study can be expanded to avoid certain limitations faced and with expanded loading and analysis conditions to better understand the mechanical characteristics of hip implants. This will be helpful to create better implant designs in future and avoid revision surgeries.

References

1. Ceddia M, Solarino G, Cassano GD, Trentadue B. Finite Element Study on Stability in the Femoral Neck and Head Connection to Varying Geometric Parameters with the Relates Implications on the Effect of Wear. *Journal of Composites Science*. 2023;7(9):387. doi:10.3390/jcs7090387

2. Kao YYJ, Koch CN, Wright TM, Padgett DE. Flexural Rigidity, Taper Angle, and Contact Length Affect Fretting of the Femoral Stem Trunnion in Total Hip Arthroplasty. *The Journal of Arthroplasty*. 2016;31(9, Supplement):254-258. doi:10.1016/j.arth.2016.02.079

3. Ashkanfar A, Langton DJ, Joyce TJ. A large taper mismatch is one of the key factors behind high wear rates and failure at the taper junction of total hip replacements: A finite element wear analysis. *Journal of the Mechanical Behavior of Biomedical Materials*. 2017;69:257-266. doi:10.1016/j.jmbbm.2017.01.018

4. Gilbert JL, Buckley CA, Jacobs JJ. In vivo corrosion of modular hip prosthesis components in mixed and similar metal combinations. The effect of crevice, stress, motion, and alloy coupling. *Journal of Biomedical Materials Research*. 1993;27(12):1533-1544. doi:10.1002/jbm.820271210

5. Gavali SL, Gawande SH, Patil SR, Yerrawar RN. Experimental Stress Analysis of Hip Joint Implant for Fracture Analysis. 2016.

6. Deng C, Gillette JC, Derrick TR. Finite element analysis of femoral neck strains during stair ascent and descent. *Sci Rep*. 2021;11(1):9183. doi:10.1038/s41598-021-87936-y