# Optimization of acetabular-femoral joint

New York University, Department of Biomedical Engineering

Supreeth Sai Krishna

## Abstract

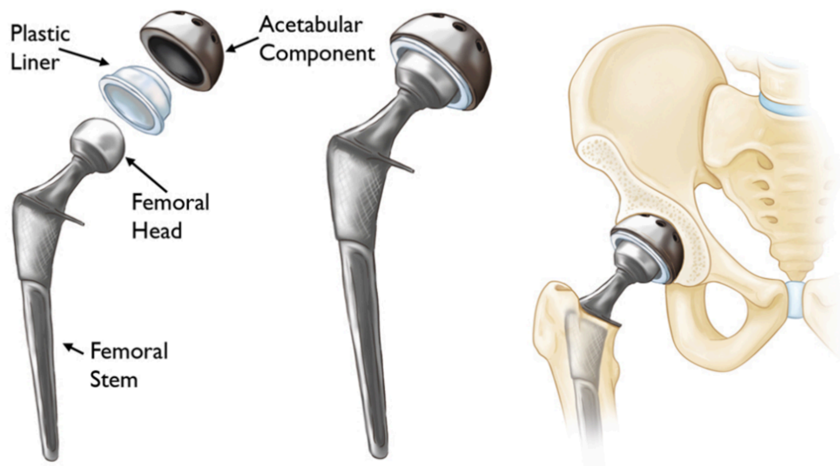
This study presents a comprehensive computational and experimental investigation of a modular femoral implant, encompassing detailed geometric modeling, finite element analysis (FEA), and additive manufacturing evaluation. Individual components of the implant—including the acetabular cap, liner, femoral head, and stem—were parametrically designed in Fusion 360 and subsequently assembled to form the complete implant geometry. The assembled model was imported into ANSYS Workbench, where meshing and application of physiological boundary conditions were performed. Contact interactions between implant interfaces were modeled under bonded, frictional (coefficients of 0.1 and 0.2), and frictionless scenarios to assess their influence on stress distribution, deformation, and strain under a 20 N applied load, corresponding to a pressure of 18.9 kPa on the femoral head surface. FEA results reveal that frictional contacts produce more localized stress concentrations compared to the bonded case, while the friction coefficient significantly affects the magnitude and spatial distribution of deformation and strain. Additionally, the study compares the manufacturability of the implant using two desktop 3D-printing platforms—Ultimaker 3 and Bambu Lab—by analyzing print times, infill patterns, and geometric fidelity from slicing snapshots. The Bambu Lab printer demonstrated a 25% reduction in print time without compromising surface quality. The strong correlation observed between Fusion 360's initial stress estimates and ANSYS's detailed FEA outcomes validates the CAD-based preliminary analysis approach. These findings underscore the critical role of contact modeling in implant performance prediction and highlight the potential of rapid prototyping workflows for efficient implant development.

Keywords: Femoral implant, Finite element analysis, Contact modeling, Additive manufacturing, Rapid prototyping

### Introduction

The acetabular-femoral joint, a critical component of human biomechanics, is fundamental to the body's ability to perform complex movements and bear significant loads. It facilitates a wide range of activities, from walking and running to lifting and carrying, making its structural integrity and functionality vital for overall mobility. As such, optimizing the design and performance of this joint is not only essential for improving quality of life but also pivotal in addressing challenges in prosthetics and orthopedic treatments. Advances in this area can lead to more durable and efficient joint replacements, enhanced rehabilitation outcomes, and better prevention of joint-related disorders.

Total hip replacement is a widely performed orthopedic procedure aimed at restoring mobility and reducing pain in patients with degenerative hip disorders. Central to the success of this intervention is the design and biomechanical performance of the femoral implant, which must replicate the natural articulation of the hip joint while withstanding physiological loads over time. A typical femoral implant comprises four key components: the acetabular cap, liner, femoral head, and femoral stem. Each of these components plays a critical role in achieving joint



stability, motion, and load distribution.

Recent advances in computer-aided design (CAD) and finite element analysis (FEA) have enabled detailed modeling and simulation of implant behavior under realistic loading and boundary conditions. Tools such as Fusion 360 allow for precise geometric modeling and assembly of implant components, while platforms like ANSYS Workbench facilitate high-fidelity simulations that predict stress, strain, and deformation across complex contact interfaces. Modeling these interfaces—whether bonded, frictional, or idealized ball joints—is essential to capturing the real-world mechanical behavior of the implant.

In addition to computational modeling, the integration of additive manufacturing (AM) has opened new avenues for rapid prototyping and patient-specific implant development. 3D printing technologies, such as those provided by Bambu Lab and Ultimaker, enable the fabrication of complex geometries with varying infill structures, material properties, and resolution. Evaluating the performance and print efficiency of these systems is essential for selecting the most suitable platform for biomedical prototyping.

This study aims to design a complete femoral implant assembly in Fusion 360, analyze its mechanical performance in ANSYS under different contact and boundary conditions, and compare the efficiency of two consumer-grade 3D printers. By examining both the virtual simulation and physical prototyping aspects, this work offers a holistic perspective on the design, analysis, and manufacturing of orthopedic implants.

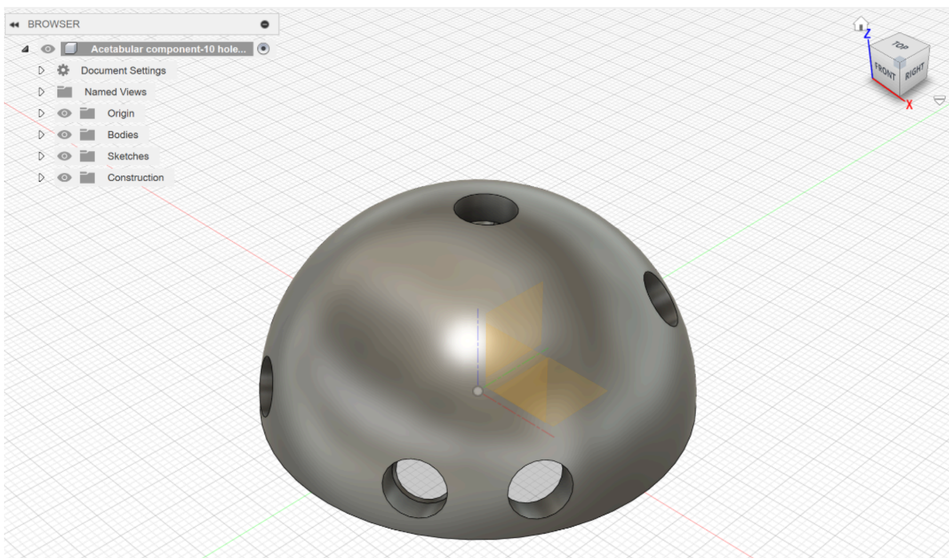
## Method

This study employed a structured approach involving geometric modeling, assembly, finite element simulation, and additive manufacturing comparison to evaluate the mechanical performance and prototyping potential of a femoral implant system. The methodology consisted of four key phases: CAD modeling, joint assembly and contact condition definition, FEA simulation using ANSYS Workbench, and 3D printer performance comparison.

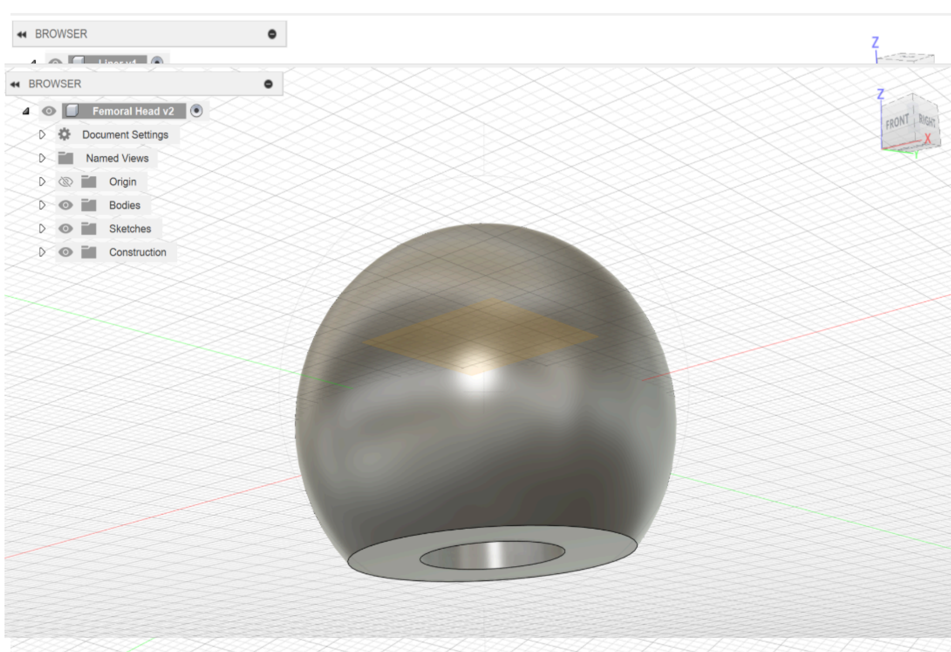
### 1. Geometric Modeling in Fusion 360

The femoral implant was decomposed into four primary components:

a.) Acetabular Cap

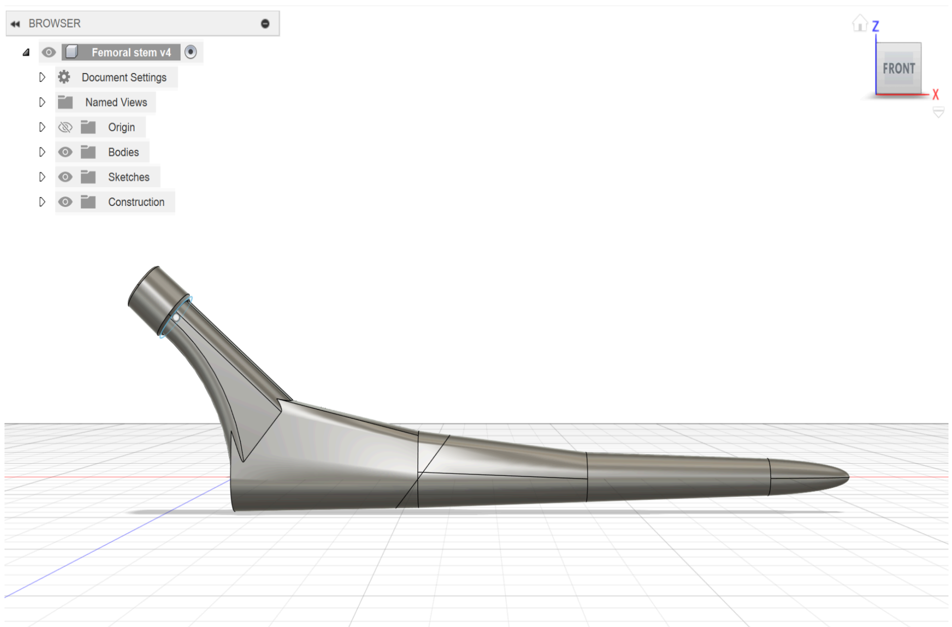



b.) Liner




c.) Femoral Head

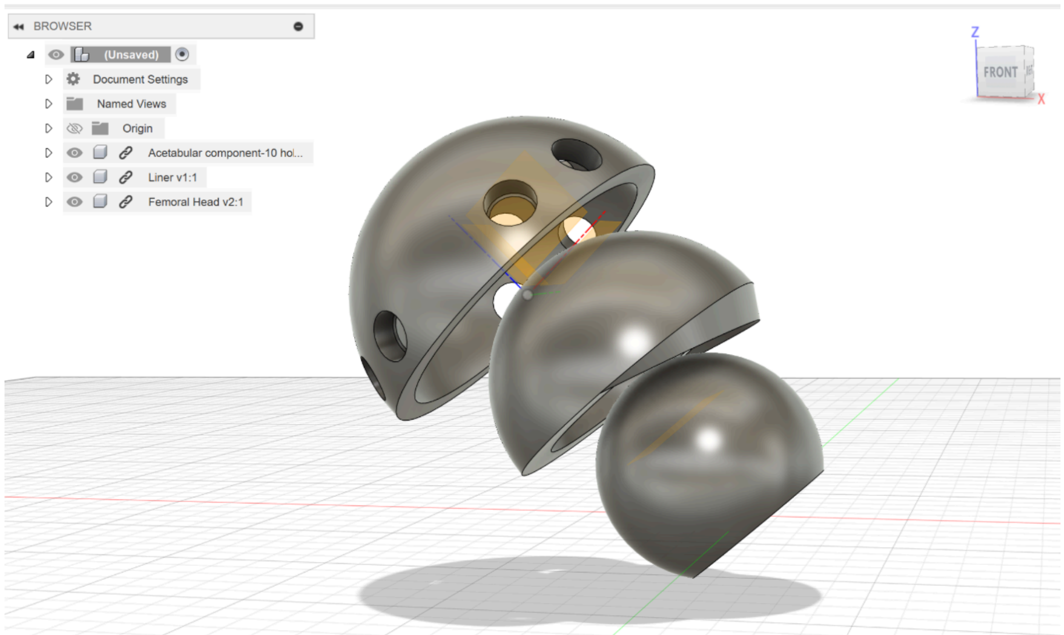
d.) Femoral Stem




Each part was individually modeled using Autodesk Fusion 360 based on anatomical proportions and mechanical design considerations. Parametric features were used to ensure consistency and facilitate future design modifications. After individual modeling, the components were assembled within Fusion 360 to form a complete femoral implant.

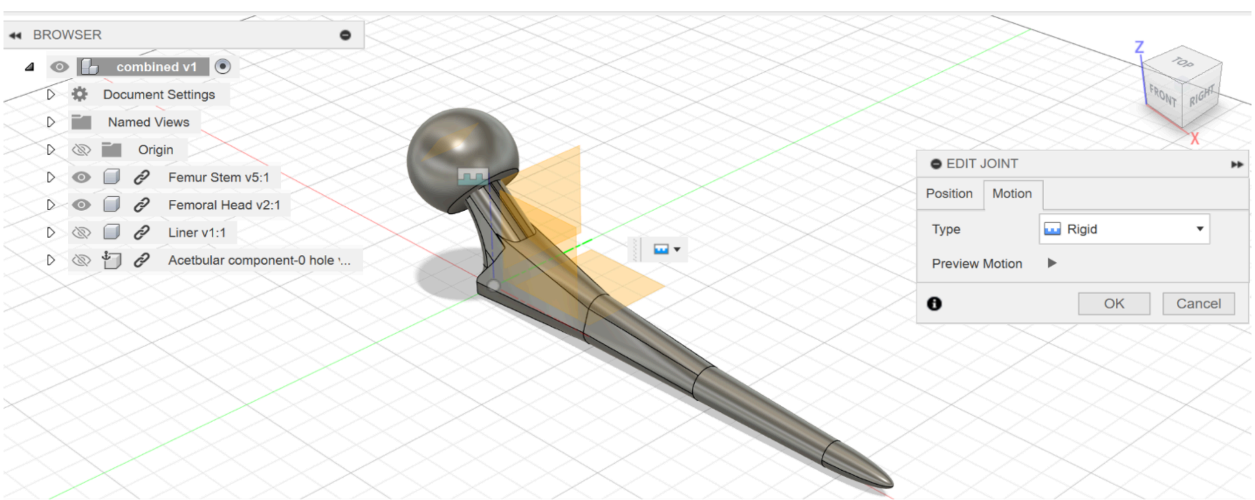
## 2. Assembly and Contact Definition

The assembled model was imported into Fusion 360 to define mechanical interactions and simulate realistic joint behavior.

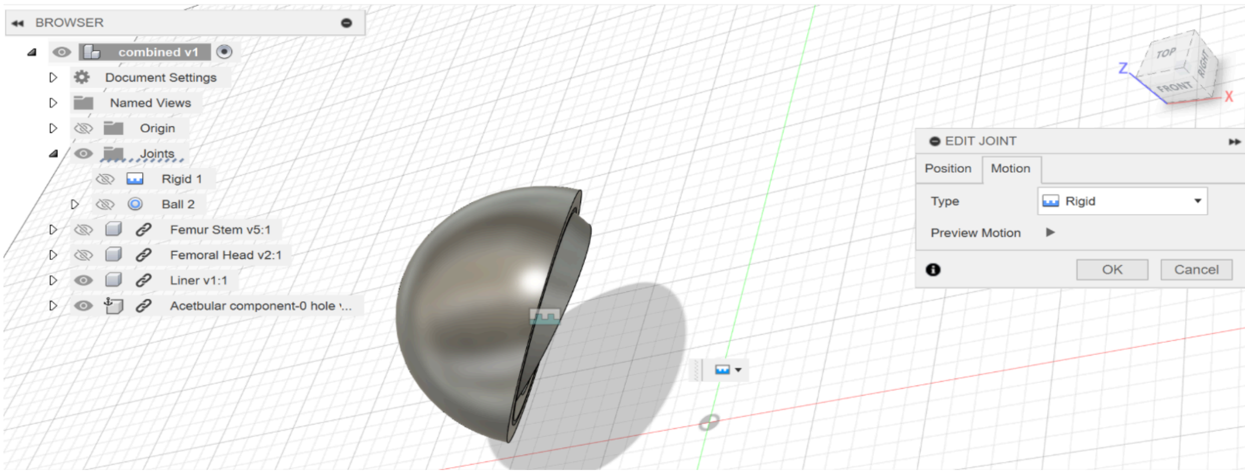



Three key types of contact conditions were defined:

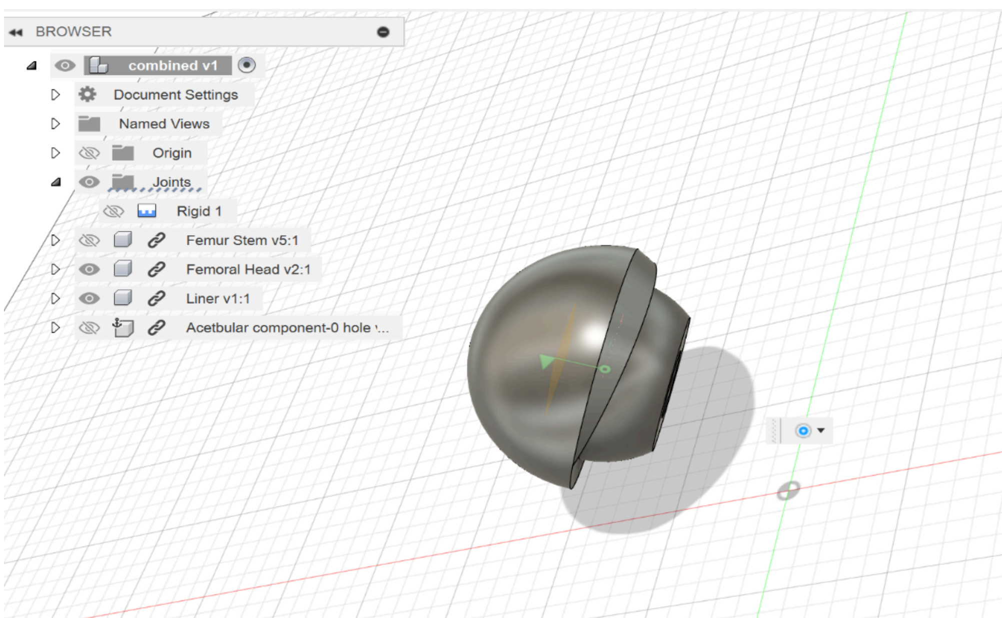
- **Rigid Joint 1** between the femoral head and stem

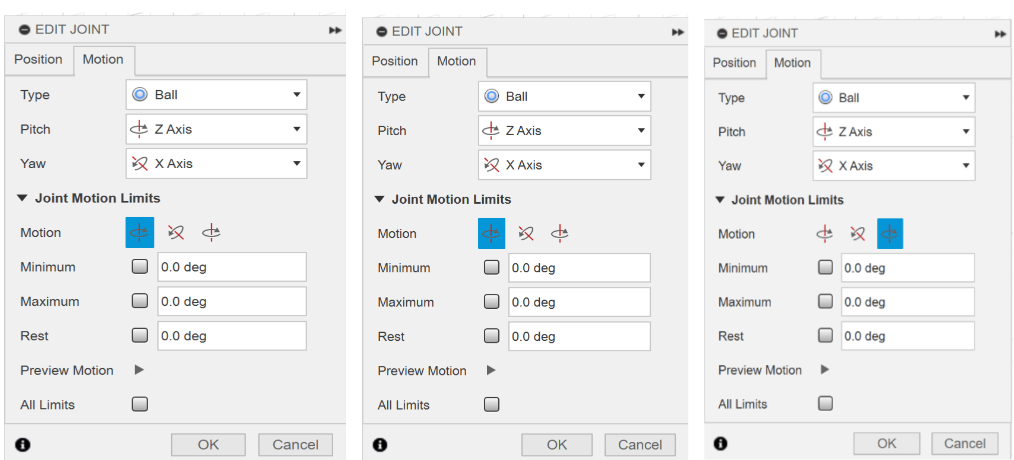


- **Rigid Joint 2** between the acetabular cap and liner



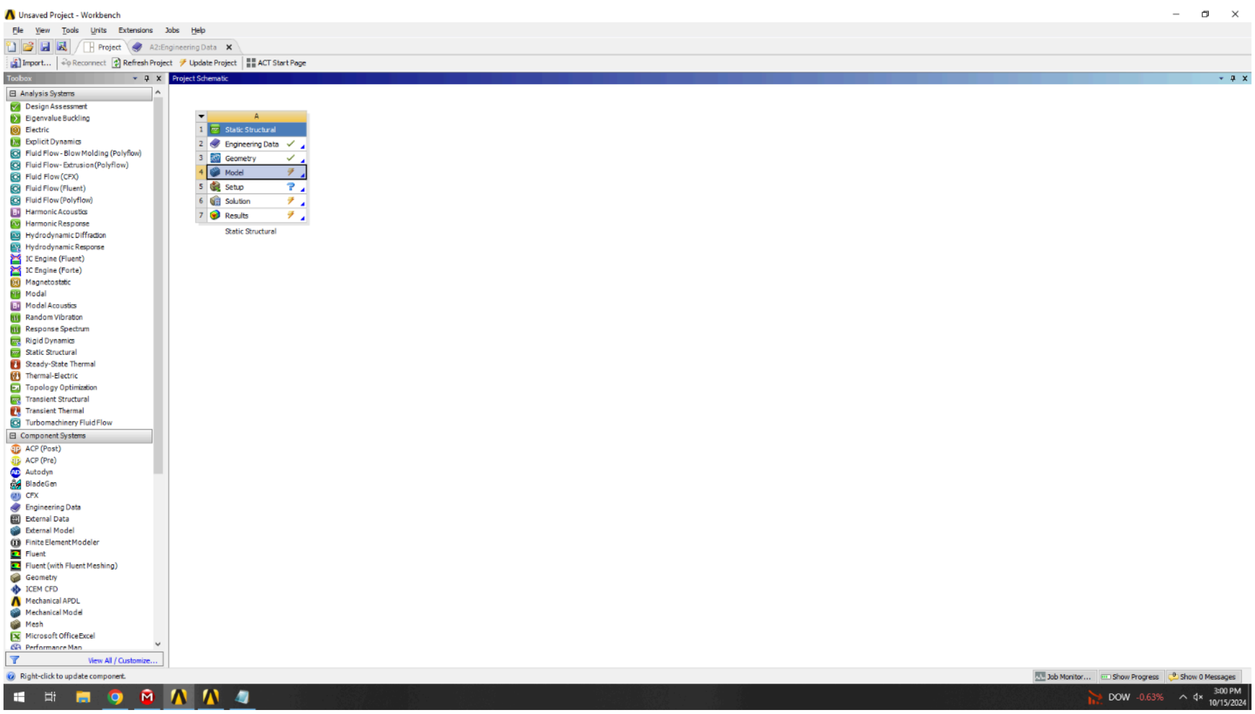

- **Ball Joint** between the liner and femoral head

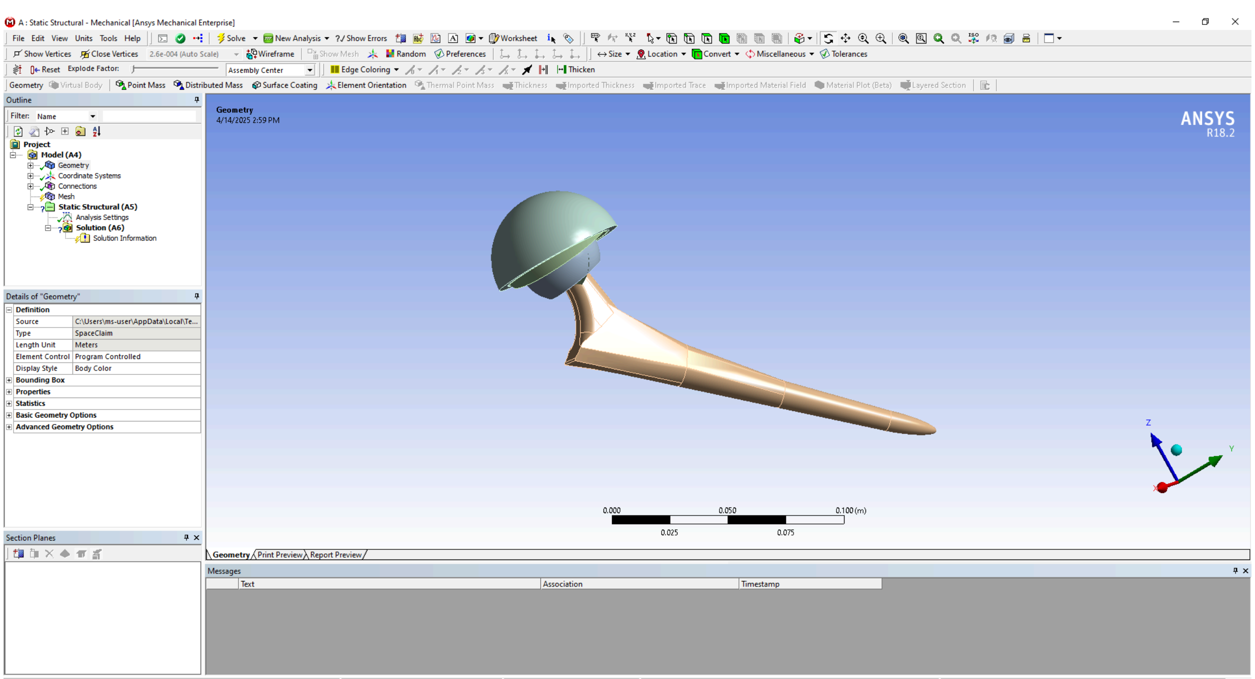


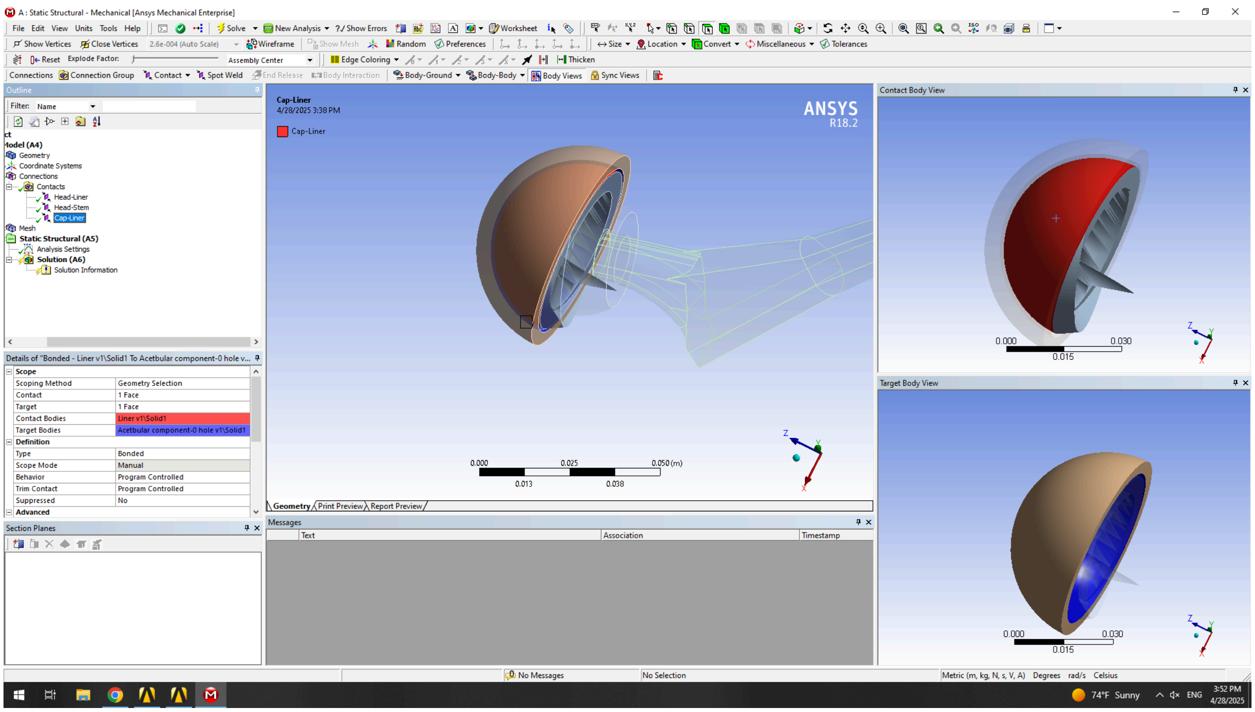
These bonding and articulation conditions aimed to mimic the real physiological constraints of the hip joint. The assembled implant was meshed using tetrahedral elements with refined control in regions of high contact complexity.

The fully assembled model was exported from Fusion 360 in f3d format and imported into ANSYS Workbench for simulation. Upon import, the bonding conditions defined in Fusion 360 were replicated in ANSYS to maintain consistency in the mechanical interactions among components. Specifically, the following contact conditions were re-applied in ANSYS.

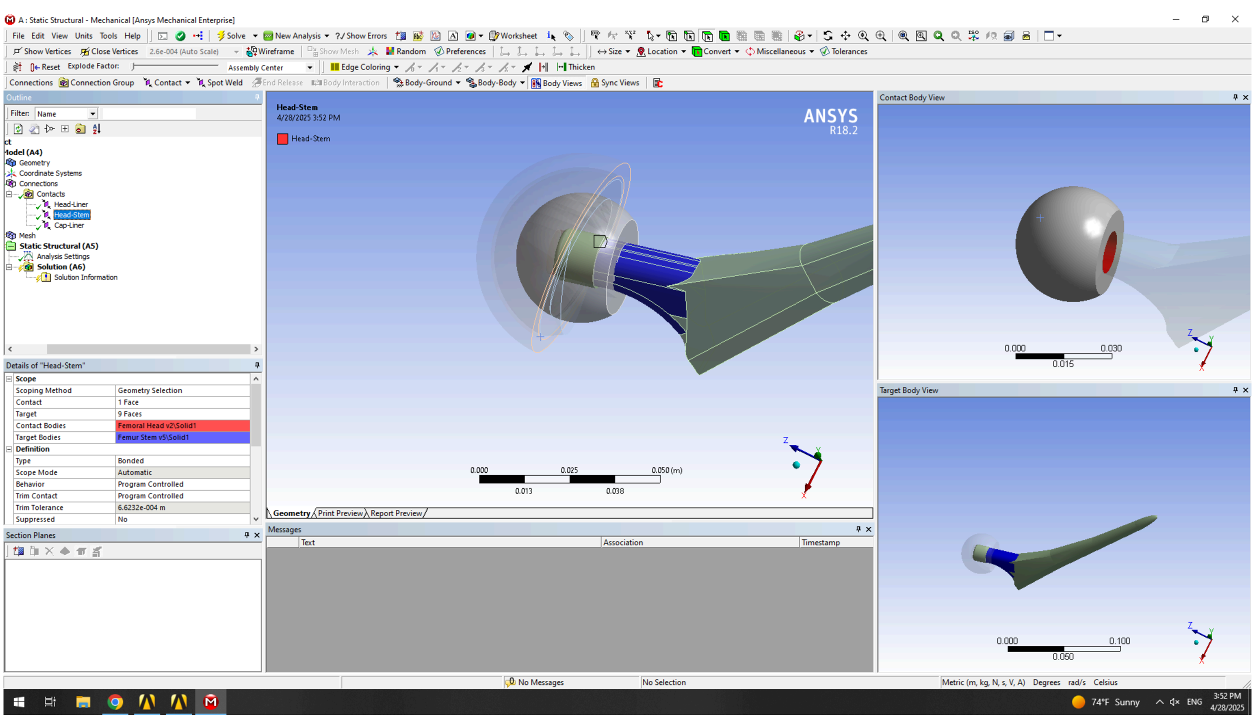




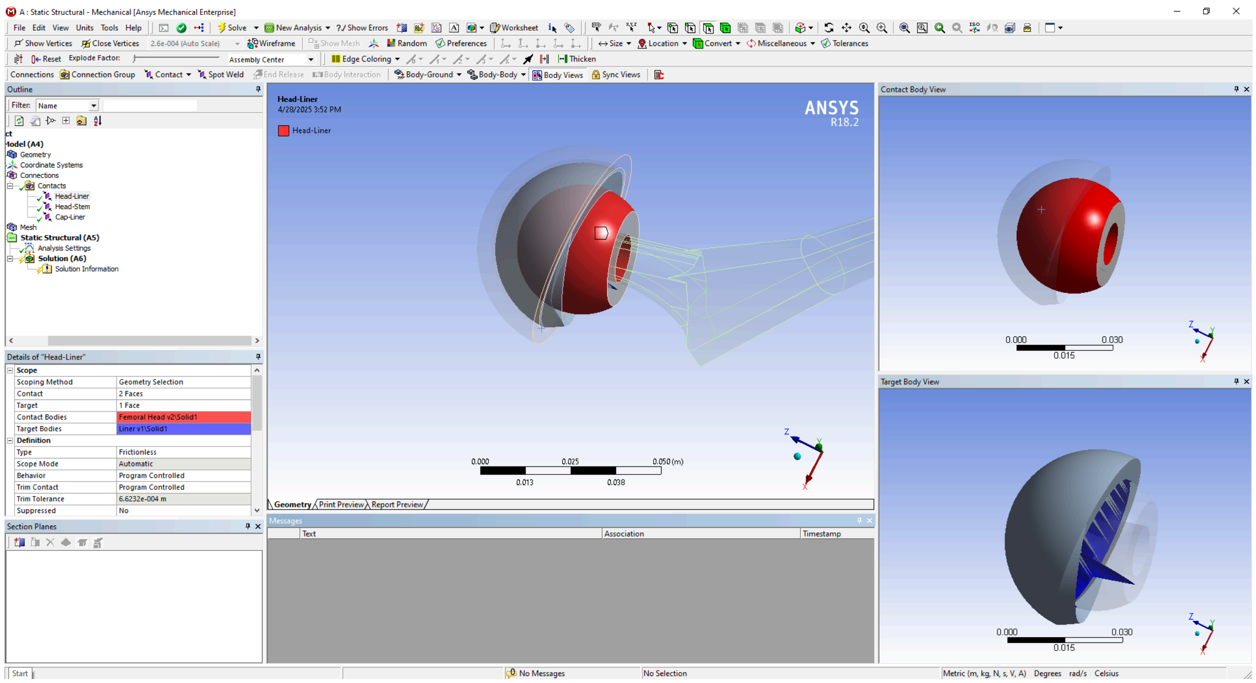
a.) Cap and Liner Bonded Connection:



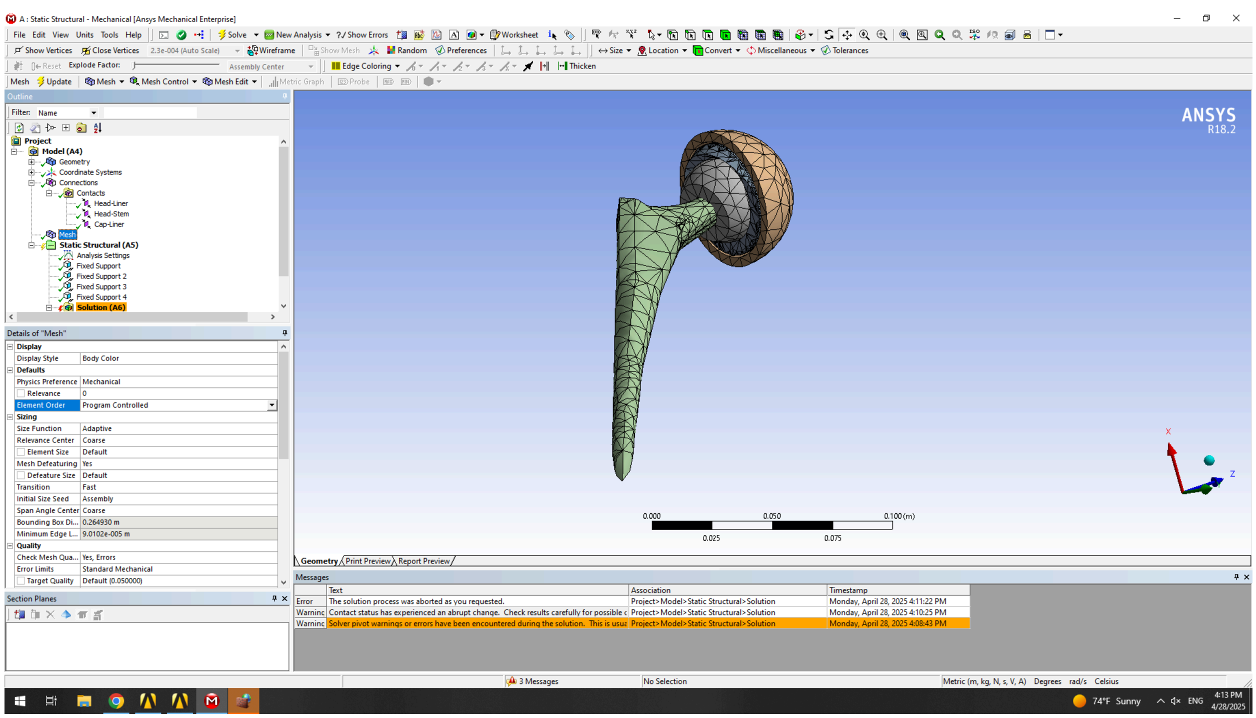

b.) Head and Stem bonded condition:



c.) Head and Liner Frictional/Frictionless contact:




Mesh conditions:




## 3. Finite Element Analysis in ANSYS

Boundary conditions were applied to simulate realistic loading:

- **Fixed supports** were defined on the acetabular cap and distal femoral stem surfaces.
- **A distributed load** of 20 N was applied to the femoral head, corresponding to a pressure of 18.9 kPa, calculated using the relation P=F/A, where r=29.

Three simulation cases were conducted under different contact settings:

- **Case 1:** Bonded contact between all interfaces
- **Case 2:** Frictional contact with a coefficient of 0.2
- **Case 3:** Frictional contact with a coefficient of 0.1

For each case, results for von Mises stress, equivalent strain, and total deformation were analyzed to evaluate the mechanical behavior of the implant under varying contact assumptions.

**4. Additive Manufacturing Comparison**

The finalized implant model was exported as an STL file and prepared for 3D printing using two desktop additive manufacturing platforms:

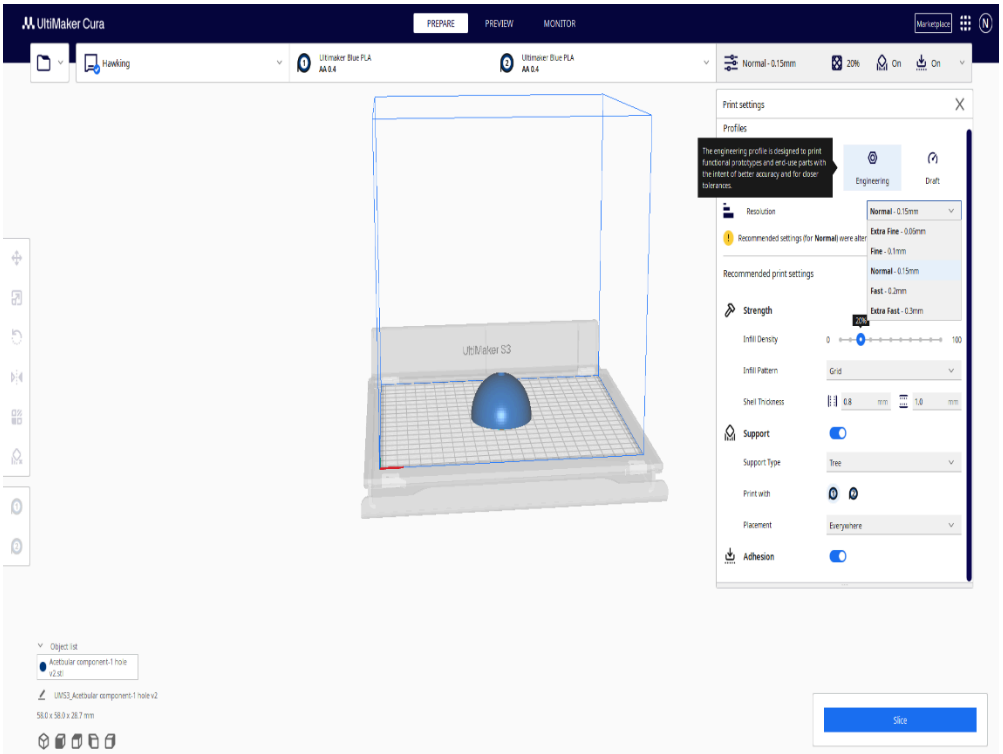
- **Ultimaker 3**
- **Bambu Lab X1-Carbon**

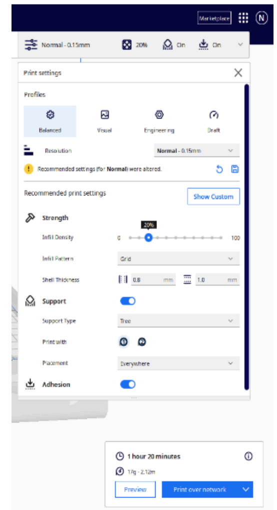
Using their respective slicing software, the model was analyzed for:

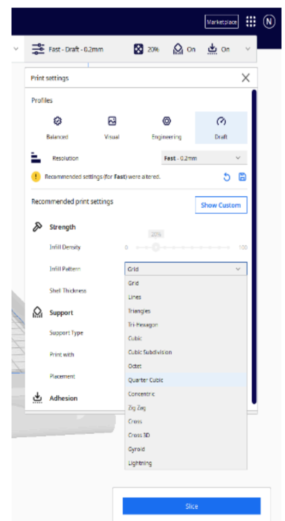
- Infill pattern
- Print time after slicing
- Surface quality and resolution

Snapshots and slicing data were compared to assess the time efficiency, fidelity, and print performance of each printer.

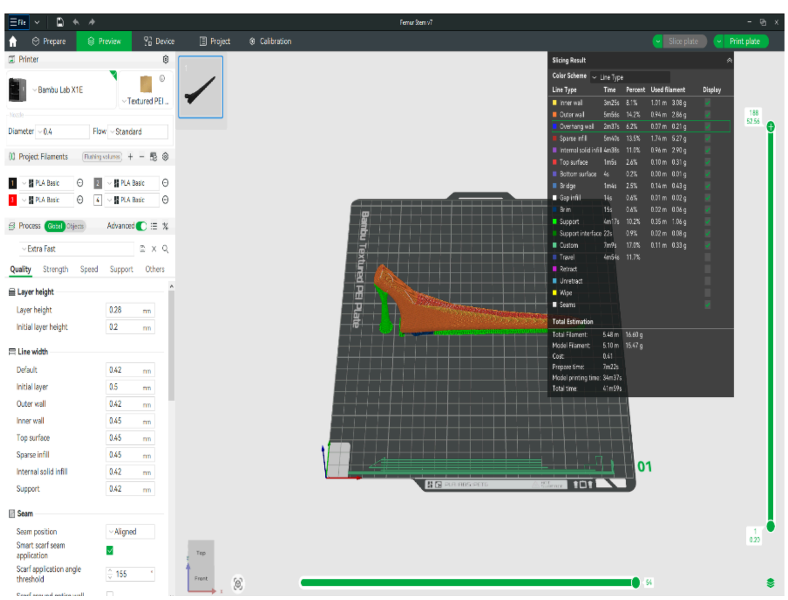
Shown below are some of the snapshots for the Ultimaker:







Shown below are some of the snapshots for Bambu Lab 3D printer:



# Results and Discussion

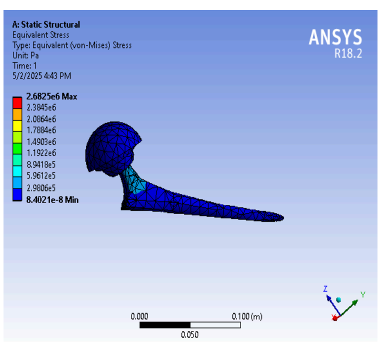
The femoral implant assembly underwent finite element analysis (FEA) in ANSYS under three distinct contact conditions to evaluate its mechanical behavior in terms of stress, strain, and deformation. Additionally, a comparative evaluation of two 3D printers—Ultimaker 3 and Bambu Lab X1-Carbon—was conducted to assess print efficiency and prototyping feasibility.

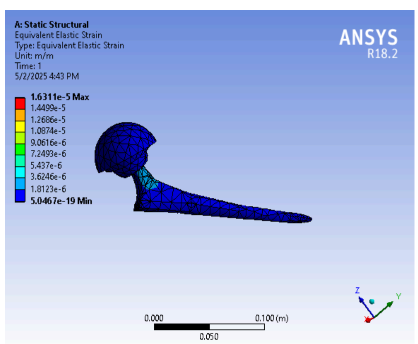
### 1. FEA Results in ANSYS

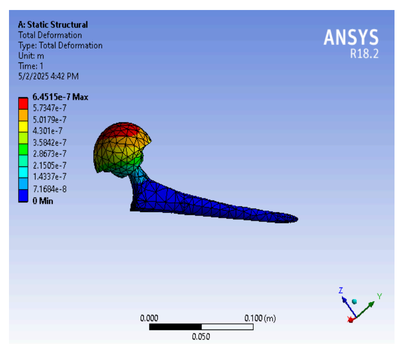
#### Case 1: Bonded Contact

Under fully bonded conditions, the implant demonstrated a relatively uniform stress distribution across the interfaces.

- **Maximum von Mises stress:** Concentrated at the neck region of the femoral stem.
- **Strain and deformation:** Minimal, with negligible sliding or separation at contact surfaces due to rigid bonding.
- **Interpretation:** Bonded conditions simulate an idealized scenario with no relative motion, potentially underestimating local stress concentrations seen in real implants.





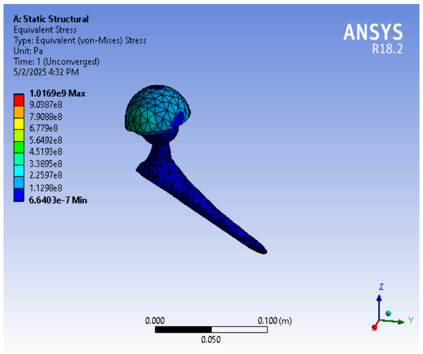


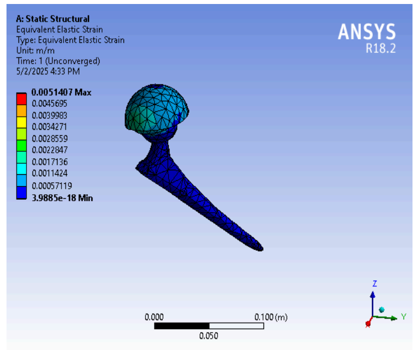

#### Case 2: Frictional Contact (μ = 0.2)

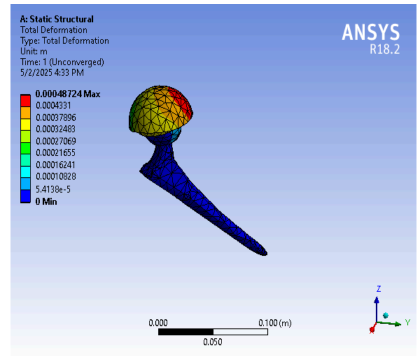
This condition introduced tangential resistance at contact interfaces, more closely approximating real-life implant biomechanics.

- **Stress:** Increased compared to the bonded case, particularly at the head-liner and liner-cap interfaces.
- **Strain and deformation:** More pronounced at contact regions, indicating relative motion and higher internal resistance.

- **Interpretation:** A friction coefficient of 0.2 produced higher but more realistic stress values, capturing the complexities of load transfer between implant parts.



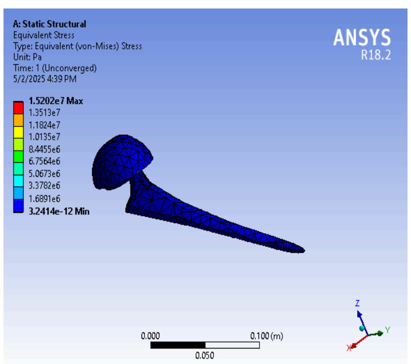



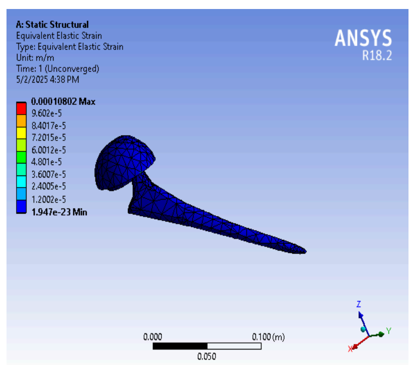



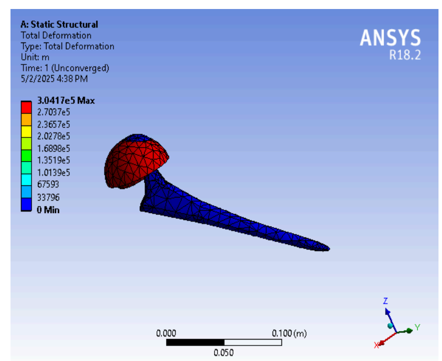

**Case 3: Frictional Contact (μ = 0.1)**

Reducing the friction coefficient resulted in intermediate mechanical responses.

- **Stress and strain:** Slightly lower than Case 2, but higher than bonded case.
- **Deformation:** Slightly increased due to reduced resistance to tangential motion.
- **Interpretation:** Lower friction allows more relative motion at joints, which can be beneficial for wear distribution but may increase micromotion risk.



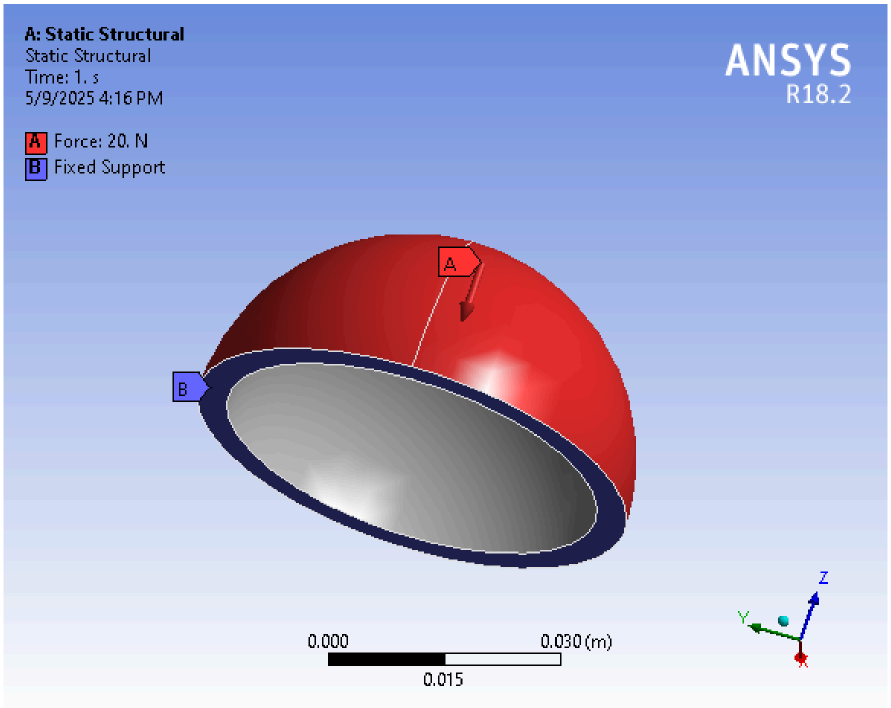




Overall, the bonded condition underestimated realistic contact behavior, while frictional cases revealed the importance of selecting appropriate material interface properties in implant design. The stress concentration in all cases consistently occurred near the neck and stem interface, emphasizing the need for robust material selection in this region.

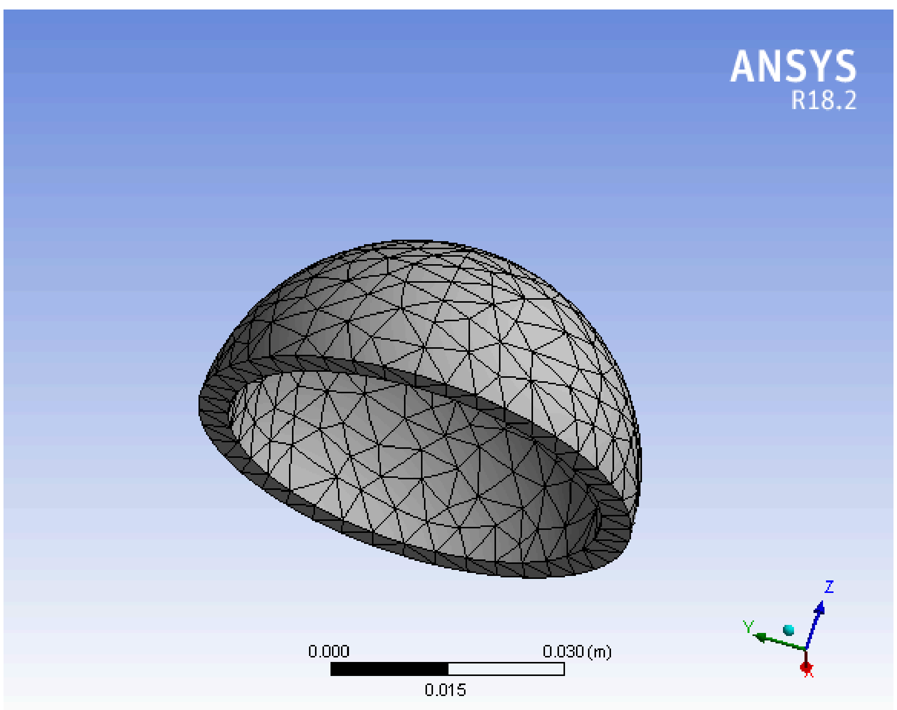
Moving forward we also carried forward FEA analysis on individual components of the Hip-Implant under different bonding conditions. With a force of 20N acting normal to the surface for each component and the corresponding stress, strain and deformation distribution.

1.) Acetabular Cap:

The boundary conditions as defined in the Figure below:

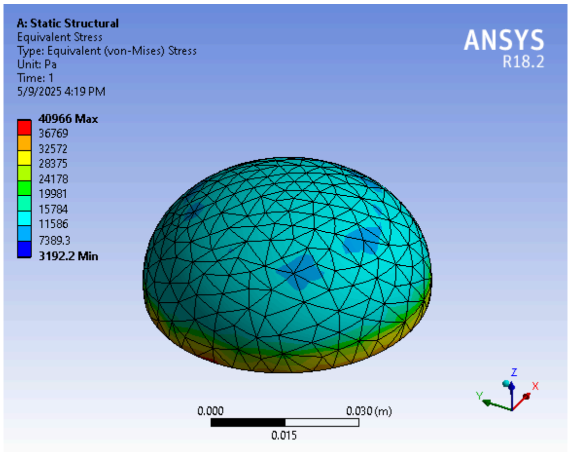


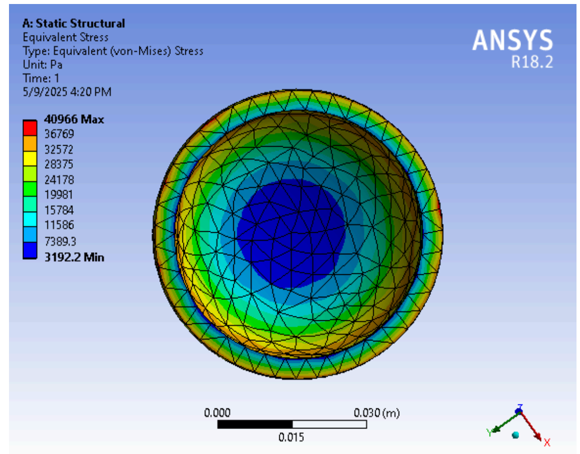

The Mesh conditions:



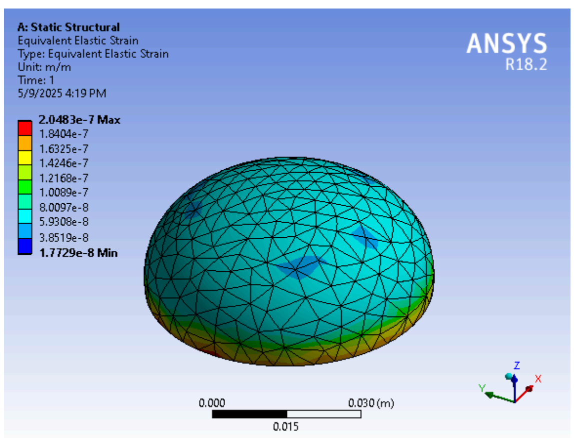
The FEA results:

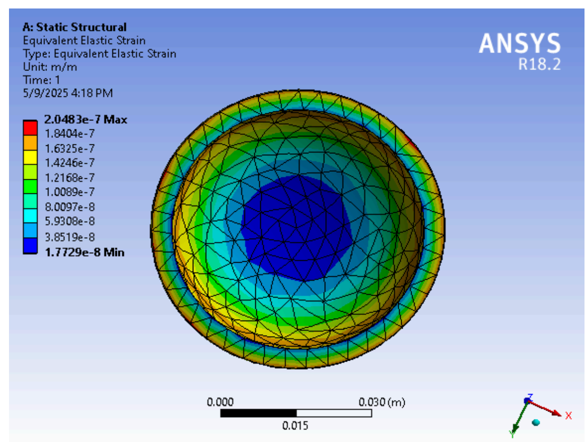
a.) Stress:



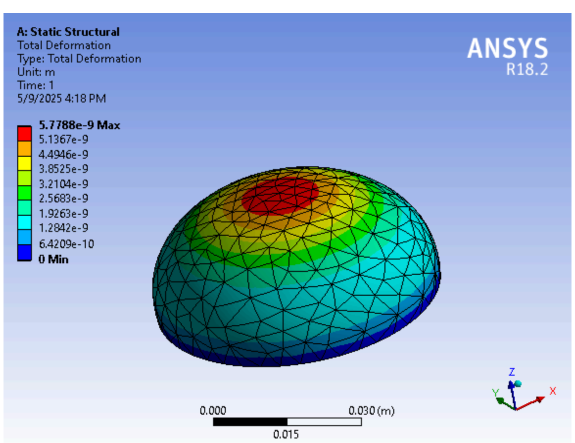



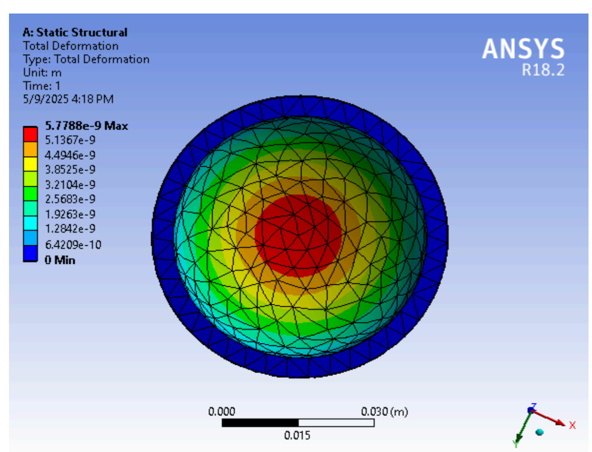

b.) Strain:



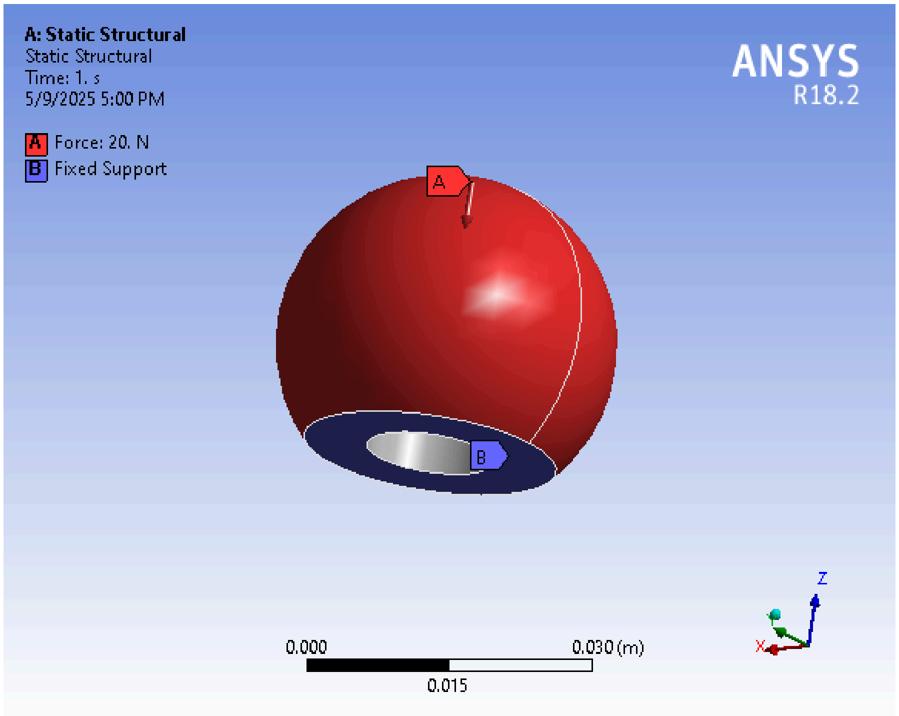

c.) Deformation:



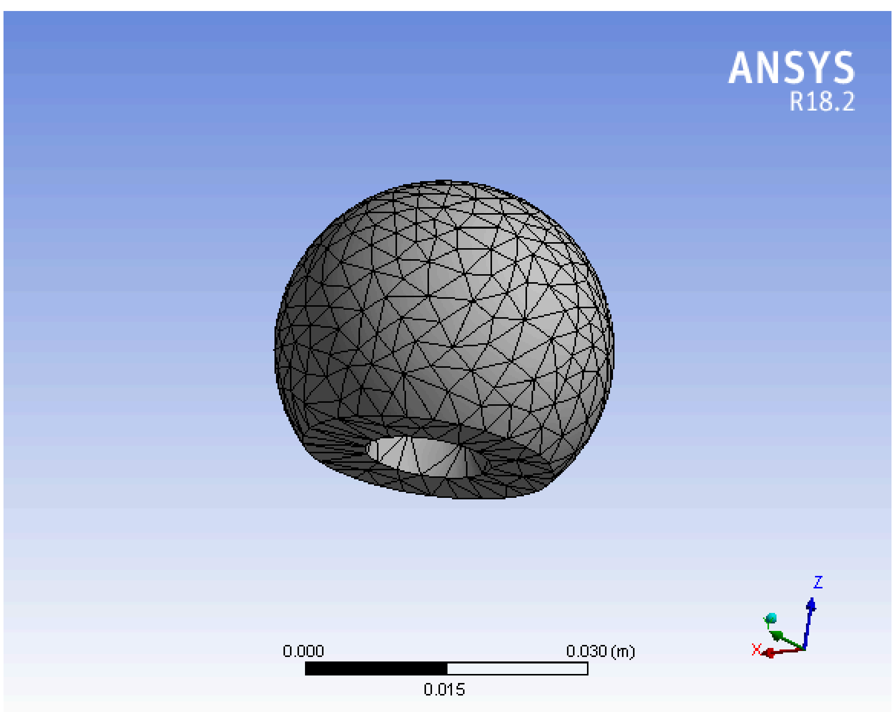



2.) Head

The boundary conditions as defined in the Figure below:

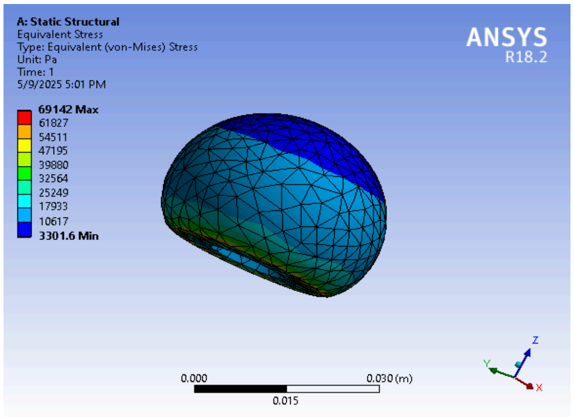


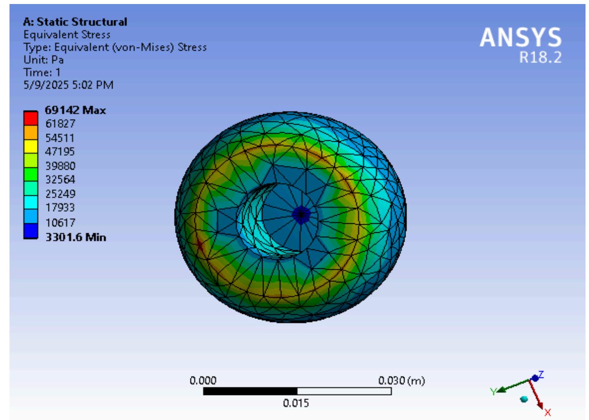

The Mesh conditions:



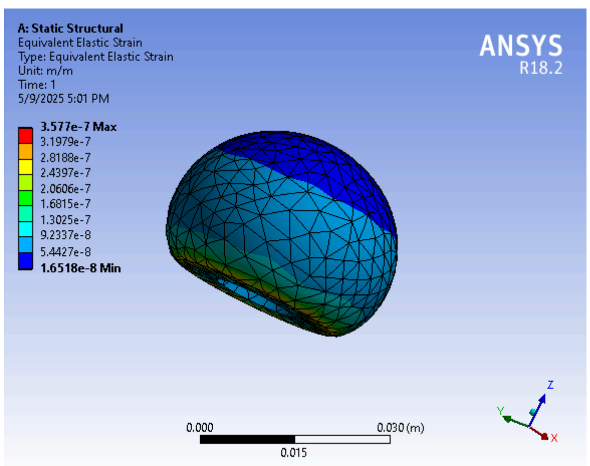
The FEA results:

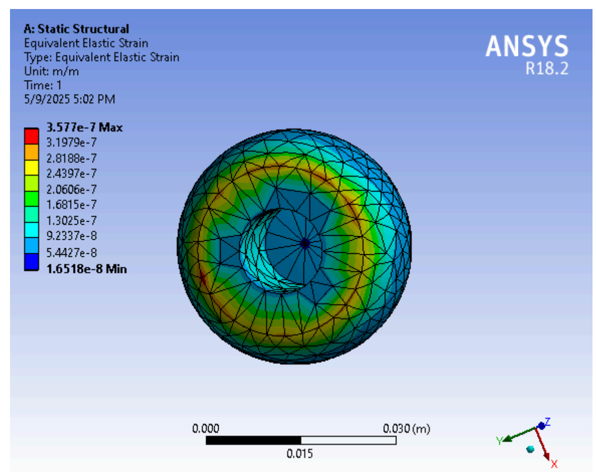
a.) Stress:



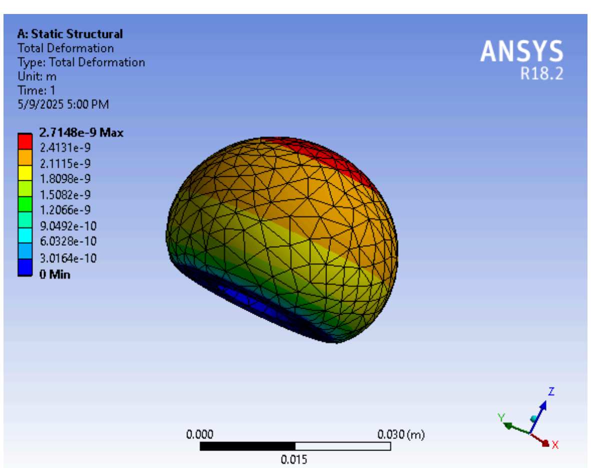



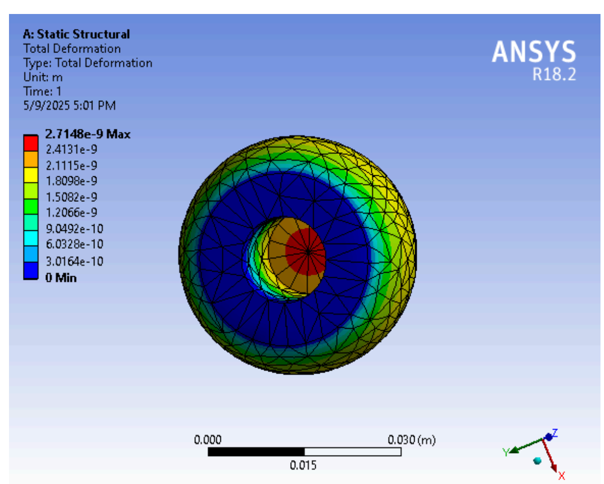

b.) Strain:



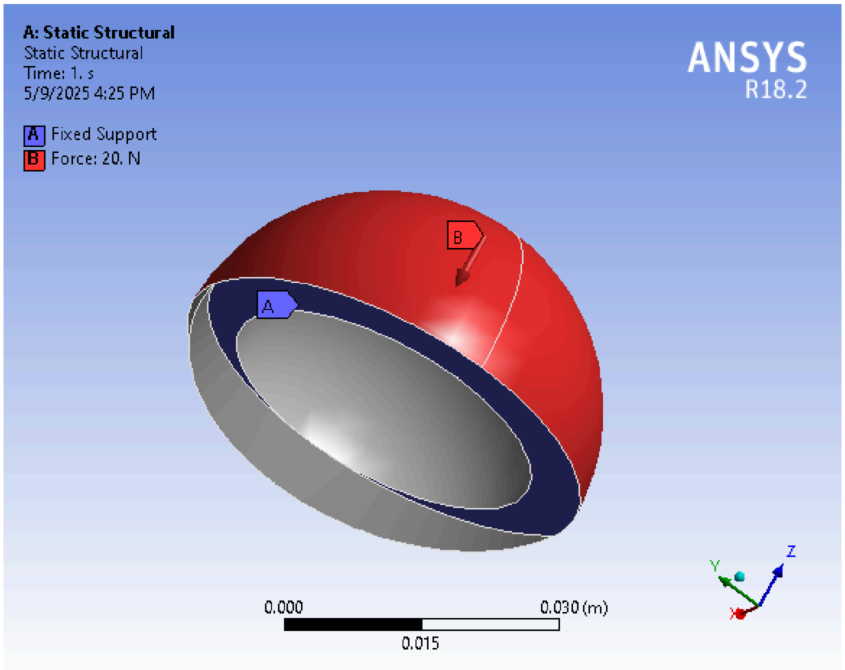

c.) Deformation:



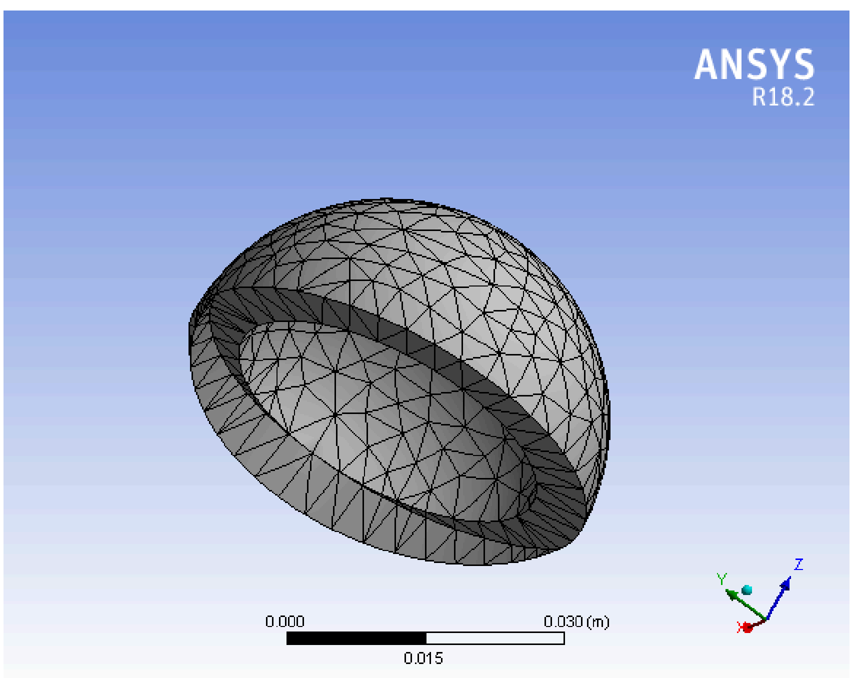



3.) Liner

The boundary conditions as defined in the Figure below:

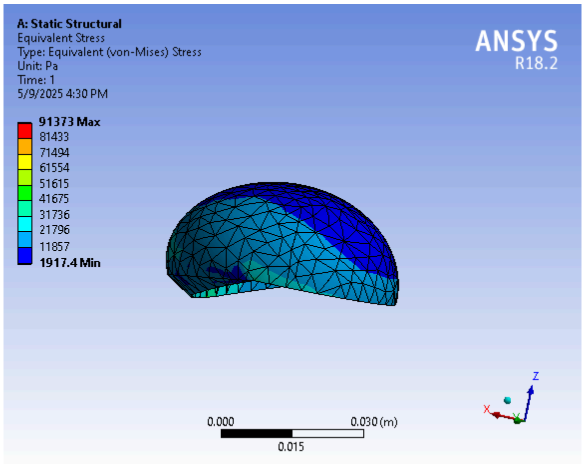


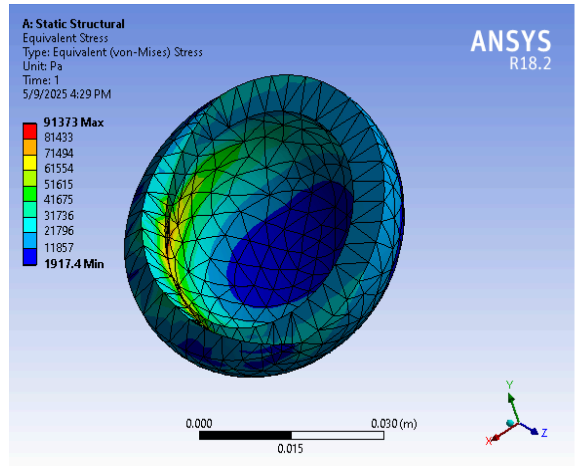

The Mesh conditions:



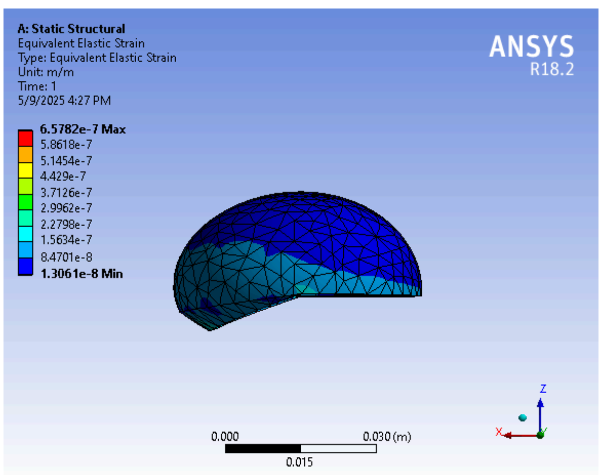
The FEA results:

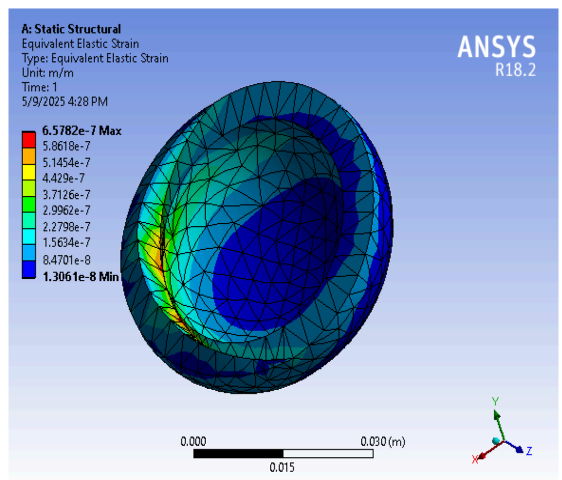
a. Stress:



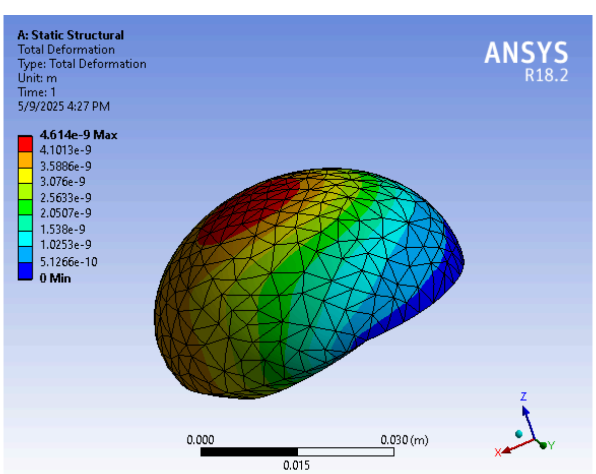



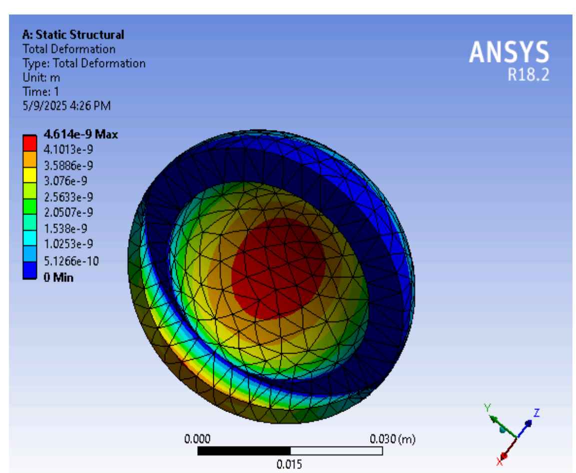

b. Strain:



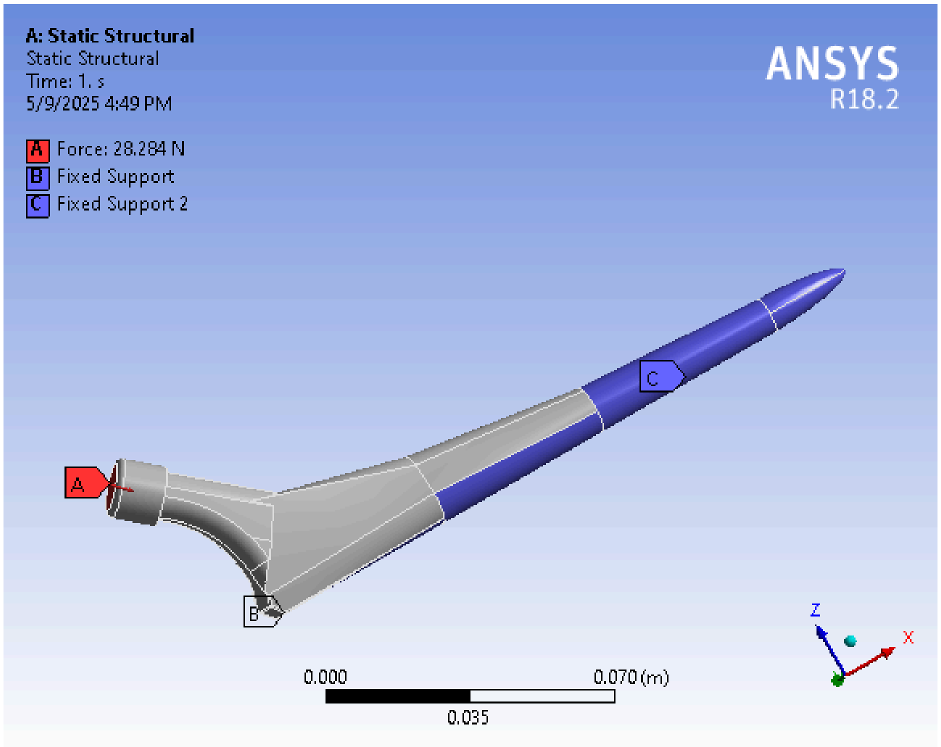

c. Deformation:



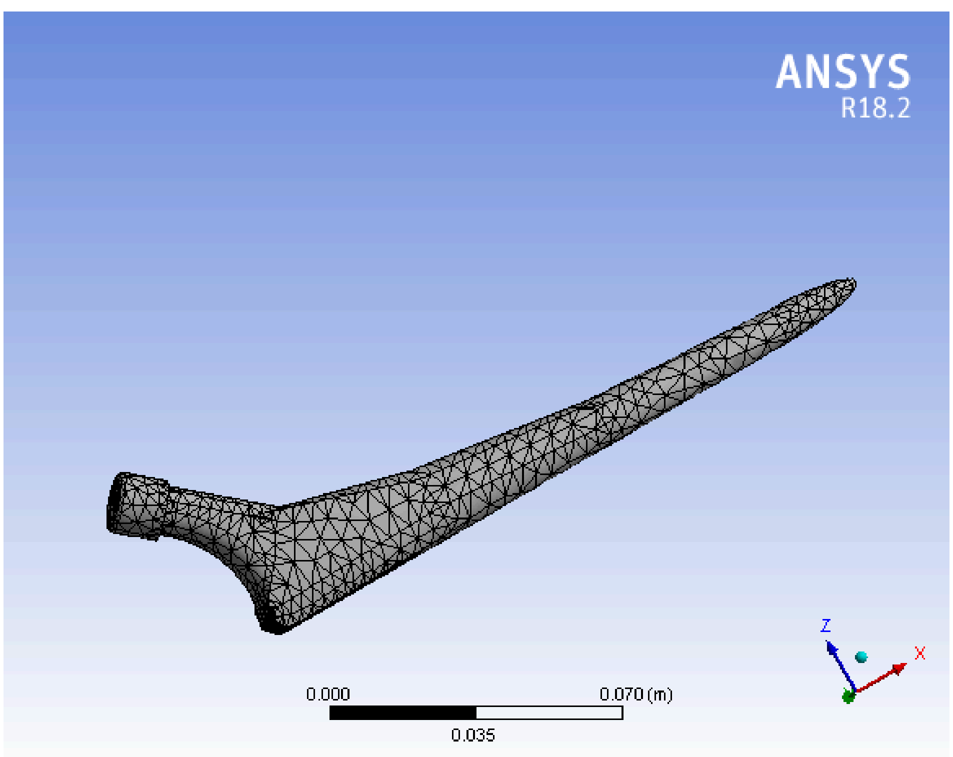



4.) Stem

The boundary conditions as defined in the Figure below:

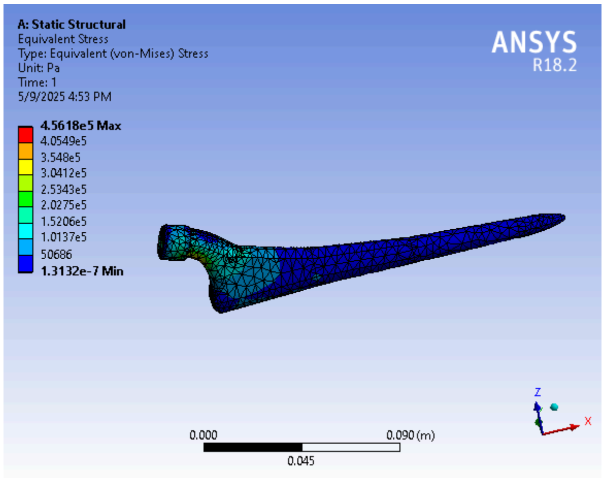


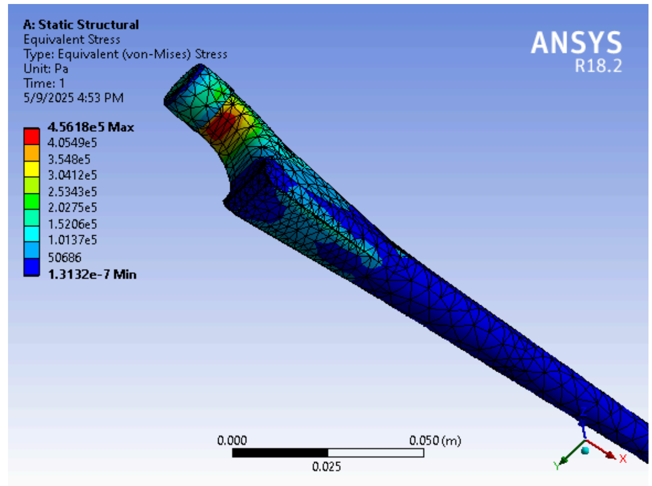

The Mesh conditions:



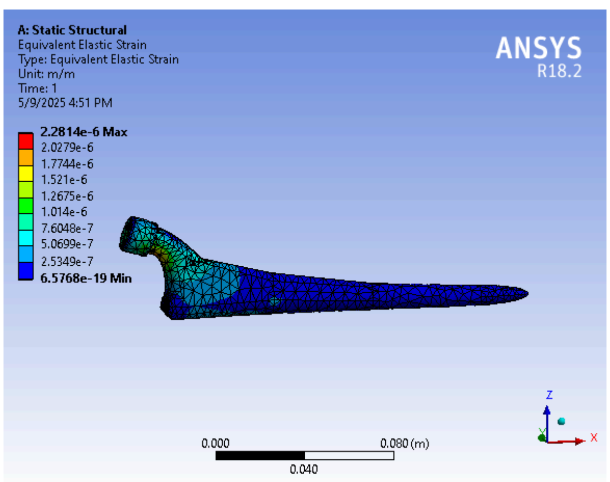
The FEA results:

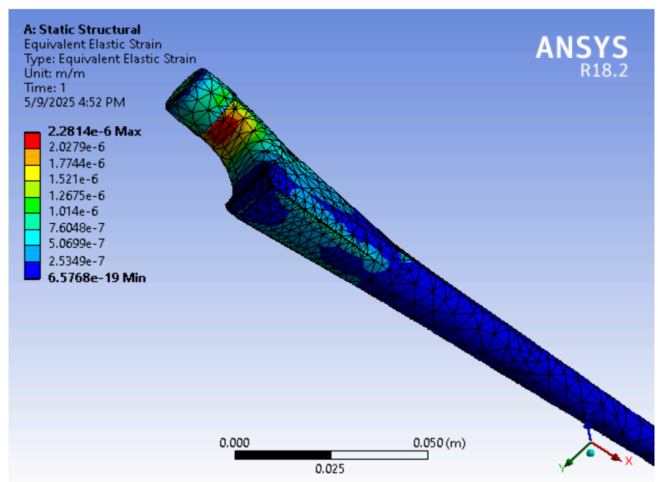
a.) Stress:



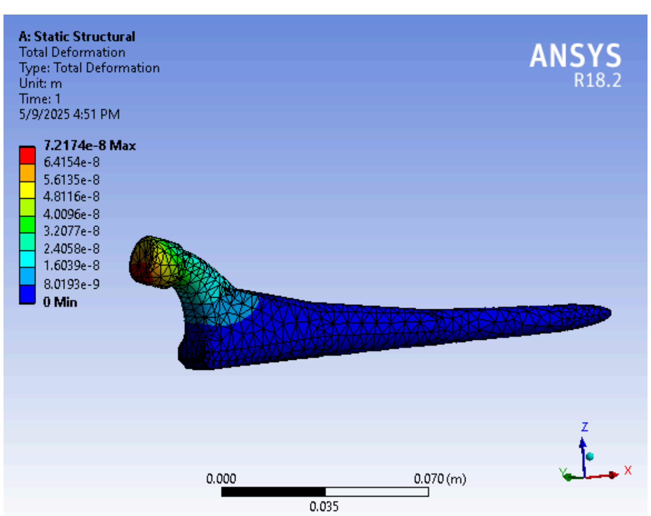




b.) Strain:




c.) Deformation:



The Table below shows the print time for each component in two different 3D printers for different print quality.

| PART | Normal (BL) | Fine<br><br>(BL) | Very Fine<br><br>(BL) | Normal<br><br>(UM) | Fine<br><br>(UM) | Very fine<br><br>(UM) |
|---|---|---|---|---|---|---|
| Acetabular Cap | **42 mins** | **47 min** | **1 hr** | **1 hr 4min** | **1 hr 58 mins** | **3 hr 16 min** |
| Liner | **47 mins** | **53 mins** | **57 mins** | **47 mins** | **1 hr 24 mins** | **2 hrs 23 mins** |
| Femoral Head | **34 mins** | **31 mins** | **42 mins** | **42mins** | **1 hr 7 min** | **1 hr 28 mins**<br><br>**(Failed)** |
| Femoral Stem | **40mins** | **1hr 1min** | **1hr 51min** | **2hr 4mins** | **4 hr 3 min (Failed)** | **7hr 42mins**<br><br>**(Failed)** |

Legend:
BL: Bambu Lab; UM: Ultimaker

**Discussion Summary:**

The integration of CAD, FEA, and additive manufacturing allowed a comprehensive evaluation of the femoral implant design. The mechanical simulations highlighted the critical role of contact modeling in assessing implant performance, with frictional conditions revealing more realistic stress profiles. Meanwhile, the 3D printing comparison underscored the importance of selecting the right hardware for iterative design processes—especially in time-sensitive clinical applications.

This study reinforces the utility of simulation-driven design workflows and rapid prototyping tools in modern orthopedic implant development.

## Conclusion

This study successfully demonstrated an end-to-end design, simulation, and prototyping workflow for a modular femoral implant, incorporating both computational modeling and additive manufacturing techniques. By utilizing Fusion 360 for geometric modeling and ANSYS Workbench for finite element analysis, we were able to explore the mechanical performance of the implant under various bonding and contact conditions. The results revealed that frictional interactions (particularly with a coefficient of 0.2) more accurately captured stress and deformation patterns at the implant interfaces than idealized bonded conditions, emphasizing the need for realistic biomechanical modeling in implant design.

Additionally, the comparative assessment of the Bambu Lab and Ultimaker 3D printers highlighted the importance of efficient and high-fidelity prototyping tools in biomedical engineering. Bambu Lab demonstrated superior performance in terms of reduced print time and improved reliability, particularly under fine and very fine print settings, making it a preferred option for rapid prototyping of complex anatomical components.

Overall, this integrated approach underscores the value of simulation-driven design and prototyping in optimizing implant structures. Future work may involve dynamic loading simulations, incorporation of patient-specific anatomical data, and validation through physical mechanical testing to further advance implant optimization and translational application in orthopedic surgery.

## References

[1] Timoshenko, S.P. and Gere, J.M., 1972. Mechanics of Materials. *McGraw-Hill*.

[2] Beer, F.P., Johnston, E.R., DeWolf, J.T. and Mazurek, D.F., 2015. Mechanics of Materials. *McGraw-Hill*.

[3] Reddy, J.N., 2004. Introduction to the Finite Element Method. *Wiley*.

[4] Cengel, Y.A. and Boles, M.A., 2007. Thermodynamics: An Engineering Approach. *McGraw-Hill*.