Fatigue Analysis of Varying Femoral Head Diameters in Total Hip Implants Using Finite Element Method

Komonibo, Ernest Bolouebi[1*] and Brisibe Endurance[2]
[1.] Department of Mathematics/Computer Science,
Faculty of Basic and Applied Sciences,
University of Africa, Toru–Orua, Bayelsa State, Nigeria.
*Email: komoniboernest@gmail.com
[2]. Department of Mechanical Engineering,
Faculty of Engineering, University of
Agriculture, Nembe Bassambiri, Bayelsa State, Nigeria.

https://doi.org/10.56201/ijemt.vol.12.no11.2026.pg62.71

Abstract:
Mechanical endurance is recognized as a critical design criterion for ensuring the optimal operational performance of hip implants. Consequently, the primary objective of this research is to assess the fatigue behaviour of hip implants across a range of femoral head diameters. Specifically, the investigation aimed to elucidate how variations in prosthetic femoral head dimensions affect the cyclic durability of total hip replacement prostheses. To achieve this, seven distinct hip implant models, fabricated from the Ti-6Al-4V alloy and featuring femoral head diameters ranging from 28mm to 52mm, were developed using SolidWorks Premium 2025. These models were then subjected to finite element analysis (FEA) via ANSYS 2022 R-2 to characterize their respective fatigue responses. The FEA results demonstrated an inverse correlation between the fatigue life (expressed as the number of cycles) and the femoral head diameter. Specifically, cyclic endurance diminished proportionally with increasing femoral head dimensions. The peak cyclic longevity, recorded at 1.120e7 *cycles, was associated with the 28mm femoral head diameter, while the lowest cyclic longevity of* 5.016e6 *cycles corresponded to the 52mm femoral head diameter. Thus, the findings of this research comprehensively illustrate the impact of varying femoral head diameters on the fatigue life of hip implants.*

Keywords: *Femoral Head Diameter, Hip Implant, Finite Element Analysis, Ansys, Fatigue life.*

1. Introduction

While it is inherently challenging to completely eliminate issues with hip implants, contemporary research increasingly focuses on developing advanced design tools to more precisely predict potential complications. The successful design of hip implants requires close collaboration between engineers and surgeons to ensure optimal stability and efficacy. To lessen the adverse effects of implant failure and predict such occurrences, various investigations utilise numerical methods to improve the overall dependability of orthopaedic implants. Significant advancements in hip implant design, material characteristics, robotic surgical techniques, and clinical experience have substantially extended the functional lifespan of implants, establishing partial and total hip

arthroplasty as effective surgical solutions for managing arthritis. Nonetheless, instances of implant fatigue failure or loosening continue to arise. Aseptic loosening of the implant is a frequent cause of implant failure. The forces generated by human movement impose fluctuating stresses on hip implants, potentially leading to dynamic failure. Consequently, ensuring the implant's resistance to dynamic failure is crucial. Fatigue resistance is therefore a vital design criterion to ensure a high level of safety for implants. Although standard fatigue tests can estimate a material's fatigue service life, experimental methods are often expensive and time-consuming for material selection or optimization, frequently requiring retesting for minor variations. However, comparative studies between fatigue-life simulation analyses and experimental results indicate that a well-configured Finite Element Analysis (FEA) can accurately represent the actual stress variations within a hip implant. FEA has proven to be a powerful and dependable numerical technique for addressing complex engineering challenges, enabling the analysis of structures under diverse loading conditions. It is understood that any finite element solution approximates the real-world scenario. Past evaluations of hip implants have explored various stem geometries, including oval, trapezoidal, elliptical, and circular, fabricated from different materials such as Ti-6Al-4V alloy, CoCrMo alloy, and UHMWPE. The findings suggested that trapezoidal stems exhibited the least stress and deformation, leading to recommendations to use CoCrMo alloy in stem designs. Furthermore, experiments utilizing Ti-6Al-4V alloy investigated the development of fatigue fracture, stress, and strain within the hip implant. This research highlighted varying stress distributions, with the implant's neck region and the area surrounding a crack displaying the highest stress concentrations. In the present investigation, a trapezoidal cross-section stem is analyzed using finite element analysis (FEA) with ANSYS 2022 R-2, systematically varying the femoral head diameter for a homogeneous material. This study assesses the fatigue performance of hip implants with femoral head diameters ranging from 28 mm to 52 mm. The primary objective is to determine how changes in femoral head diameter influence the fatigue life of a lightweight hip implant.

2. Materials and Methods

This research utilizes a homogeneous material. The choice of this material was guided by the mechanical properties of the human hip bone and its physiological environment. The mechanical characteristics of human skeletal tissue are outlined in Table 1 [12].

Table 1: Mechanical Properties of Human Bone

Density (Kg/m^3)	Yield Strength (Pa)	Poisson's Ratio	Young's Modulus (Pa)
310 ± 60	1.14e 8	0.62±0.26	1.1e+10 - 2e+10

When selecting biomaterials, two primary factors must be considered. The first factor concerns medical requirements related to the material's biocompatibility. The second factor pertains to mechanical considerations, including wear resistance, stress concentrations, and implant stability [12]. Consequently, Table 2 presents the mechanical properties of biomaterials to aid in selecting a material that is compatible or nearly compatible with human bone [13].

Table 2: Mechanical Properties of Biomechanical Materials (MPMM)

Materials	Density (Kg/m^3)	Yield Strength (N/m^2)	Poisson's Ratio	Young Modulus (N/m^2)
Ti-6Al-4V	430	8.8e+08	0.28	1.09e+11
UHMWPE	930	2.45e+07	0.46	6.5e+08
HDPE	964	2.95e+07	0.4	2.2e+09
Stainless Steel	8000	2.15e+08	0.27	2.1e+11

This investigation primarily considers the Ti-6Al-4V alloy, owing to its inherent physical properties and excellent biocompatibility with human osseous tissue. It is the most widely used titanium alloy, comprising nearly 50% of all titanium-based products currently on the market [14]. In contrast to materials such as stainless steel or cobalt-chromium alloys, titanium and its alloys facilitate direct bone integration, thereby obviating the requirement for a cementing layer. Furthermore, these alloys demonstrate superior wear resistance, a characteristic attributed to their comparatively low shear strength [15]. While both commercially pure titanium (often referred to as sterling titanium) and Ti-6Al-4V are prevalent implant materials, the latter is increasingly replacing the former due to its superior mechanical properties. Consequently, Ti-6Al-4V was selected as the material for the hip implant (HI) designs in this investigation. This alloy is characterized by substantial strength, contributing to its durability, a low density (430 kg/m³) that imparts a lightweight profile, and a Poisson's Ratio of 0.28. It exhibits a Young's Modulus of 1.09e+11 N/m², and its overall performance is generally comparable to that of stainless steel. The alloy's excellent corrosion resistance is also highly advantageous for biomedical applications, further enhancing its significant biocompatibility. For the purposes of this investigation, seven distinct artificial hip implants were computationally modelled using SolidWorks Premium 2025. Across all implant configurations, the dimensions of the femoral stem and acetabular cup were maintained uniformly, with variations exclusively applied to the femoral head's dimensions. Specifically, the acetabular cup featured an internal diameter of 14mm and a material thickness of 7mm. Each implant also incorporated a trapezoidal femoral stem with a total length of 122.45mm and a consistent neck-shaft angle (NSA) of 140 degrees. The femoral head is a critical component of hip implant design. In this study, to facilitate stress analysis, the seven femoral heads were designed with diameters ranging from 28mm to 52mm, increasing by 4mm between iterations. A representative CAD model of an implant featuring a 28mm femoral head, developed in SolidWorks, is presented in Figure 1.

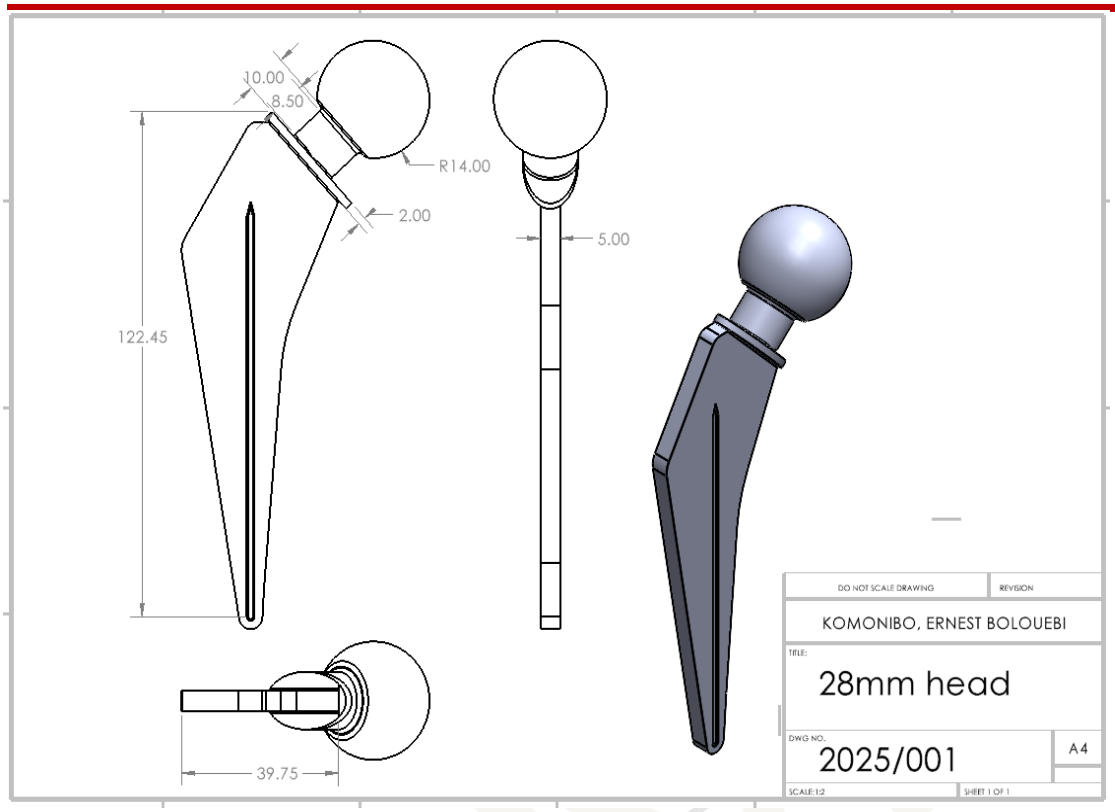


Figure 1: CAD Model for the 28mm Diameter HI

The developed computational models were subsequently transferred to a specialized finite element analysis (FEA) software platform for the execution of static structural analyses. To evaluate and interpret the derived FEA outcomes, a statistical software package and regression modelling were employed.

3. Mesh and Boundary Conditions

An unstructured triangular mesh was utilized to discretize the computational model for the numerical simulation. The femoral head and stem components underwent independent discretization, employing distinct element sizes of 0.0012 m for the femoral head and 0.002 m for the stem. This discretization resulted in a computational grid comprising 60,092 nodes and 34,715 elements, as depicted in Figure 2. To establish mesh independence, a hip implant with a 28-mm femoral head component was investigated. The boundary conditions for this investigation adhered to the specifications outlined in ASTM F2996-13. Furthermore, the applied loading conditions during the assessment conformed to ISO F7206-4:2010. The distal aspect of the stem was fully immobilised. Subsequently, a resultant force of 1680 N, resolved into components of 378 N along the X-axis and -1624 N along the Y-axis, was applied to the surface of the acetabular backing cup, situated within the femoral head, for the evaluation.

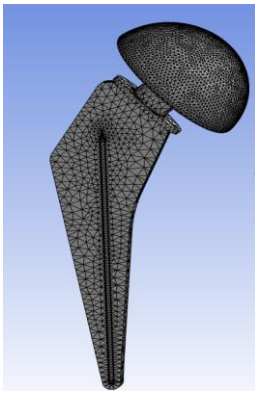


Fig 2: Mesh Report

Computational analyses were performed using Ansys 2022 R-2. The predicted fatigue life for each of the seven distinct models is tabulated in Table 3, while Figure 3 depicts the finite element analysis (FEA) outcomes for these seven diverse hip implant designs, which incorporate varying femoral head diameters.

4. Results and Discussion

Table 3 presents the computational findings, specifically the simulated fatigue life for each of the seven distinct models. Concurrently, Figure 3 illustrates the finite element analysis (FEA) conducted for the seven varied implant configurations, each distinguished by a different femoral head diameter.

Table 3: Femoral Head Diameters with corresponding Fatigue Life

S/N	FEMORAL HEAD DIAMETERS (mm)	FATIGUE LIFE (cycles)
1	28	1.120e7
2	32	1.022e7
3	36	9.613e6
4	40	8.379e6
5	44	7.306e6
6	48	6.425e6
7	52	5.016e6

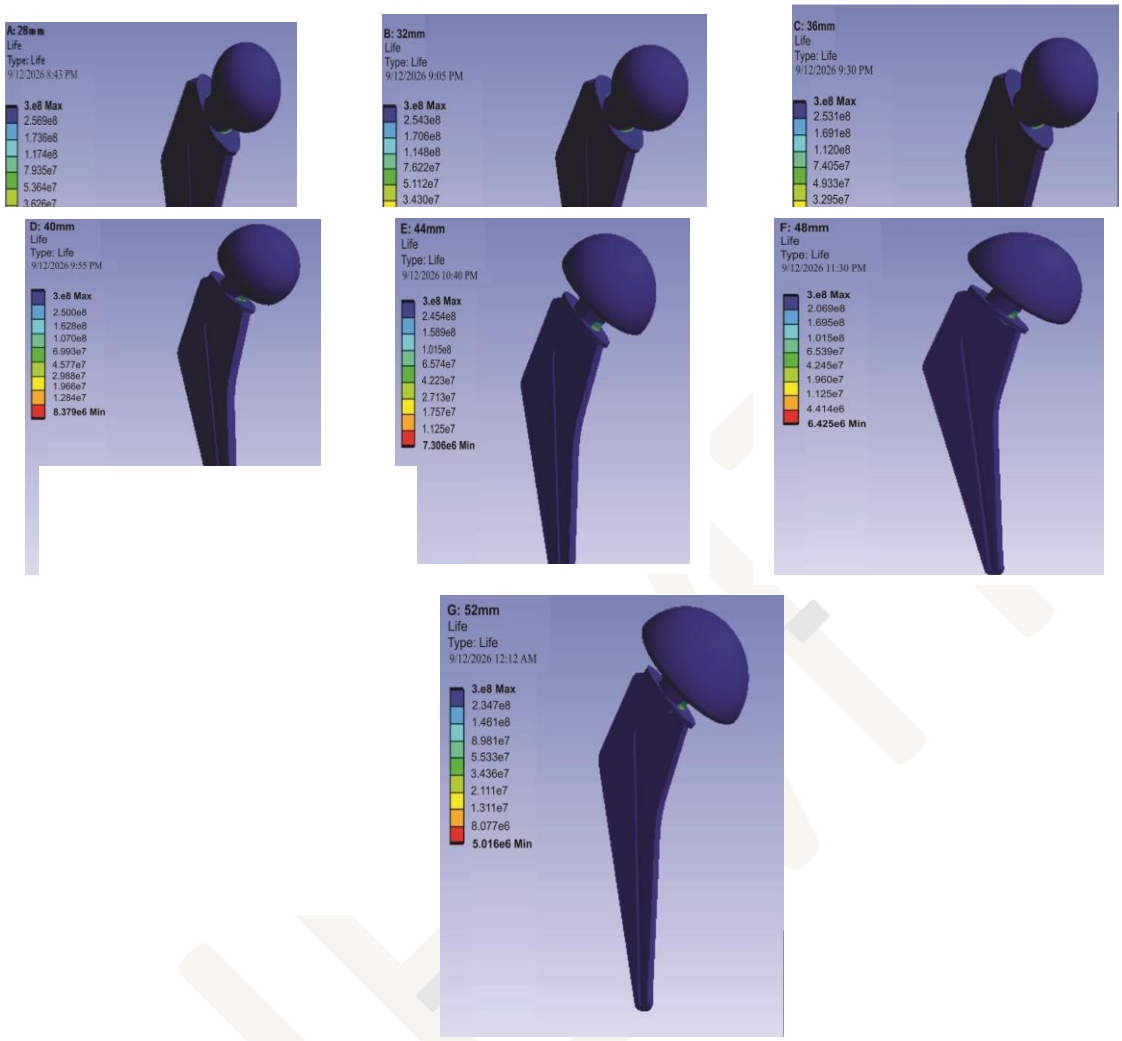


Fig 3

The computational findings from the Finite Element Analysis (FEA) indicate an inverse relationship between fatigue life and femoral head diameters. Specifically, the 28 mm-diameter head exhibited the greatest fatigue endurance, registering 29,477 cycles, while the 52 mm-diameter head demonstrated the lowest fatigue endurance, at 13,096 cycles.

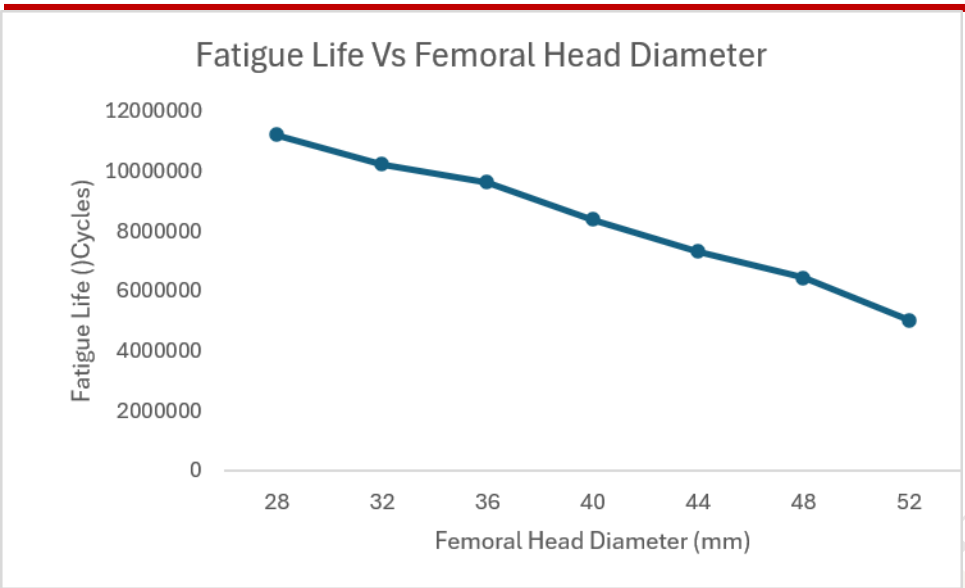


Figure 4: Graph of Fatigue Life against the Femoral Head

Figure 4 illustrates the interrelationship between the observed number of cycles and diverse femoral head diameters. To delve deeper into these findings, the dataset presented in Table 3 was subsequently subjected to a rigorous statistical assessment utilizing SPSS software (version 23). The primary objective of this investigation was to determine the level of statistical significance and delineate the nature of the association between differing femoral head dimensions and their respective cycle counts. More precisely, the number of cycles, identified as the response variable, was regressed on the femoral head diameter, which served as the explanatory (predictor) variable.

Table 4: Model Summary for Number of Cycles and Femoral Heads

Model	Ṙ	R^2	Adj R^2	Std Error of the Estimates	Durbin Watson
1	0.996	0.993	0.991	204552.792	1.874

Source: SPSS output 2025.

The computed correlation coefficient (R) of 0.996 signifies a remarkably strong positive association between the independent variable (femoral head diameter) and the dependent variable (Number of Cycles to Failure). This robust value not only indicates a pronounced linear relationship but also suggests that the linear alignment between the two variables is nearly perfect, effectively accounting for 99.6% of the potential linear relationship. Furthermore, the coefficient of determination (R-squared), assessed at 0.993, elucidates that 99.3% of the observed variation in the response parameter is attributable to the explanatory parameter.

Table 5: ANOVA for Number of Cycles and Femoral Heads

Model		Sum of Squares	df	Mean Square	F	Sig
1	Regression	2.891e13	1	2.891e13	690.753	0.001
	Residual	2.092e11	5	41845935714		
	Total	2.911e13	6			

Source: SPSS output 2025

The Analysis of Variance (ANOVA) table serves as an instrument to gauge the global utility of a statistical model by ascertaining its inferential validity. More precisely, it evaluates whether the proposed model, which incorporates at least one explanatory variable, yields a significantly improved representation of the observed data compared to a more parsimonious null model predicated solely on the mean of the dependent variable. In the present analysis, the obtained p-value (0.001) falls below the conventional threshold for statistical significance of 0.05 ($p < 0.05$). This result indicates that the model, featuring a single independent variable, substantially contributes to elucidating the variability within the outcome measure. Consequently, a statistically robust linear association is demonstrated, the relationship linking the predictor variable to the outcome variable, thereby substantiating the efficacy of the proposed linear model.

Table 6: Shows the Coefficients for Number of Cycles and Femoral Heads

Model	Unstandardized Coefficient β	Sig
	18468785.714	0.001
(Constant)	-254008.929	0.001
Femoral_Head_Diameter		

Source: SPSS Output 2025

Table 6: illustrates the functional principles governing the model's operation.

$$y_0 = \beta_0 + \beta_1 x_1 + \cdots + \epsilon \quad (1)$$

This implies that

$$y_0 = 18468785.714 - 254008.929 x_1 + \epsilon \quad (2)$$

The established model indicates that a one-millimetre increment in femoral head diameter correlates with a substantial reduction in the number of cycles to fatigue failure for the HI, estimated at approximately 254,008.929 units, ultimately accelerating the onset of fatigue failure.

5. Conclusion

In the current investigation, a Finite Element Analysis (FEA) was performed on a monolithic material, specifically Ti-6Al-4V. This alloy is widely employed in the fabrication of hip implants owing to its exceptional biomechanical attributes. The analysis incorporated a prosthetic stem characterized by a trapezoidal cross-section, with femoral head diameters ranging from 28 mm to 52 mm. Throughout the study, the dimensions of the acetabular cup and its associated backing component were kept invariant. Loading and boundary conditions were prescribed consistent with

established ASTM guidelines. The FEA results revealed that the cyclic fatigue life decreased proportionally as the femoral head diameter increased. Specifically, the maximum fatigue life, observed at 1.120e7 cycles, was associated with the 28 mm femoral head diameter, whereas the minimum fatigue life, recorded at 5.016e6 cycles, was associated with the 52 mm femoral head diameter.

References

Kaya, F., & İnce, G. (2024). Fatigue Analysis on a Newly Designed Hip Implant with Finite Element Method. *International Journal of Engineering and Innovative Research*, *6*(3), 162-178.

Marsh, M., & Newman, S. (2021). Trends and developments in hip and knee arthroplasty technology. *Journal of rehabilitation and assistive technologies engineering*, *8*, 2055668320952043.

Eachempati, K. K., Dannana, C. S., Apsingi, S., Ponnala, V. K., Boyapati, G., & Parameswaran, A. (2020). Trunnion fracture of femoral prosthesis following a large metal-on-metal uncemented total hip arthroplasty: a case report. *Arthroplasty*, *2*(1), 32.

Grupp, T. M., Weik, T., Bloemer, W., & Knaebel, H. P. (2010). Modular titanium alloy neck adapter failures in hip replacement-failure mode analysis and influence of implant material. *BMC musculoskeletal disorders*, *11*(1), 3.

Janssen, D., Aquarius, R., Stolk, J., & Verdonschot, N. (2005). Finite-element analysis of failure of the Capital Hip designs. *The Journal of Bone & Joint Surgery British Volume*, *87*(11), 1561-1567.

Łapaj, Ł., Woźniak, W., Wiśniewski, T., Rozwalka, J., Paczesny, Ł., Zabrzyński, J., ... & Kruczyński, J. (2019). Breakage of metal hip arthroplasty components: Retrieval and structural analysis. *Bio-Medical Materials and Engineering*, *30*(3), 297-308.

Mierzejewska, Ż. A. (2015). Case study and failure analysis of a total hip stem fracture. *Advances in materials science*, *15*(2), 5-13.

Joshi, T., Sharma, R., Mittal, V. K., Gupta, V., & Krishan, G. (2022). Dynamic fatigue behavior of hip joint under patient specific loadings. *International Journal of Automotive and Mechanical Engineering*, *19*(3), 10014-10027.

Pekedis, M., & Yildiz, H. (2011). Comparison of fatigue behaviour of eight different hip stems: a numerical and experimental study. *Journal of Biomedical Science and Engineering*, *4*(10), 643-650.

Zameer, S., & Haneef, M. (2015). Fatigue life estimation of artificial hip joint model using finite element method. *Materials Today: Proceedings*, *2*(4-5), 2137-2145.

Milovanović, A., Sedmak, A., Grbović, A., Mijatović, T., & Čolić, K. (2020). Design aspects of hip implant made of Ti-6Al-4V extra low interstitials alloy. *Procedia Structural Integrity*, *26*, 299-305.

Bhawe, A. K., Shah, K. M., Somani, S., Shenoy B, S., Bhat N, S., Zuber, M., & KN, C. (2022). Static structural analysis of the effect of change in femoral head sizes used in Total Hip Arthroplasty using finite element method. *Cogent Engineering*, *9*(1), 1-13.

Dimple, D., Shruti, M., Badgayan, N. D., & Sahu, S. K. (2021). Finite element analysis of hdpe-based hybrid nanocomposite for potential use as liner material for total hip prosthesis. In *Advances in Engineering Design: Select Proceedings of FLAME 2020*, 305-313.

Liu, S., & Shin, Y. C. (2019). Additive manufacturing of Ti6Al4V alloy: A review. *Materials & Design*, *164*, 107552.

Sachin, G., & Yogesh, j. (2022). Biomaterials in Hip Joint Replacement. *Journal of Engineering Research and Sciences1*, *1*(8), 01–11.