RESEARCH PAPER

Topology Optimization Customized Femoral Hip Implant.

Siddharth Tiwari[1*], Dr. N.K. Jain[2]

[1]*Research Scholar, Department of Mechanical Engineering, NIT, Raipur, India*
siddharthsantosh7@gmail.com
[2]*Professor, Department of Mechanical Engineering, NIT, Raipur, India*
nkjain.me@nitrr.ac.in

ABSTRACT

Aseptic loosening, stress shielding, and wear are some of the dominant issues that would be attempted to resolve in this research work in designing and optimizing femoral hip replacement devices by making use of finite element analysis. This research work will emphasize important aspects in designing the devices that include the size of the head area and taper angle, which were optimized for enhancing their performance. Titanium alloys show superior biocompatibility compared to other alloys. Of these alloys, Ti-6Al-4V provides superior mechanical properties and anticorrosion properties. Simulation tests conducted in this work made use of finite element analysis for studying the stresses and deformations in the devices. This helped in achieving deformations of 0.405mm and a value of 246.38 MPa for stresses that provide structural and efficient use of materials in performance criteria restricted within acceptable limits

Keywords: Hip Arthroplasty; Femur Head; Finite Element Analysis; Topology Optimization

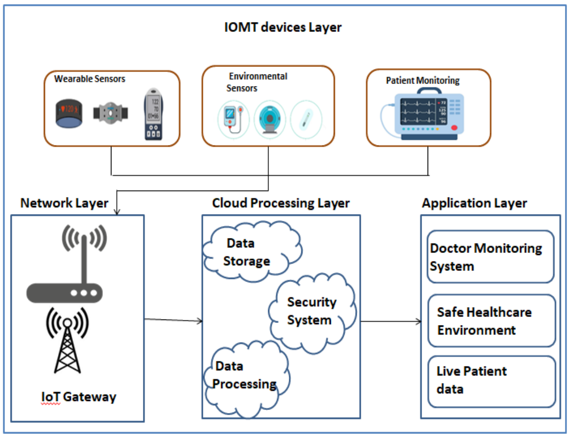
How to cite this article: Tiwari S, Jain NK.., Topology Optimization Customized Femoral Hip Implant...Int J Drug Deliv Technol. 2026; 16(11s):
776-781; DOI: 10.25258/ijddt.16.11s.78

Source of support: Nil.

Conflict of interest: None

INTRODUCTION

Total Hip Arthroplasty (THA) is indeed a life-altering surgical process and has made vast improvements to the lives of patients suffering from end-stage osteoarthritis or injured joints. The head and socket of the affected joints are replaced using artificial materials to ensure easy mobility and alleviate pain. However, some challenges related to stress shielding, aseptic loosening, and revision surgery continue to persist despite advances made within the field of surgery and material development. The use of femoral hip implants over the years has culminated in comprehending their importance in the field of orthopedic surgery. Total hip replacement (THR) is a medically proven intervention for hip joint dislocation sequelae, induced by aging or injury (Khan et al., 2017). Through fixing shortcomings such as wear, corrosion, and loosening of the implants, utility and lifespan have been enhanced, with materials and design being persistently evolving (Bonaventure et al., 2020; Tavares et al., 2021). As stated by Smith et al. (2018), there have been several modifications attempted over the years aimed at improving the geometry of the femoral head/neck junction with a goal of better efficiency of hip joint mechanics. According to a publication by Li et al. (2020), FEA thus has emerged as an effective method for analysing the mechanical performance of hip implants since it offers the possibility of analysing different designs and materials virtually prior to real implementation. As cited by Martínez et al. (2021), it is clear that generally most dynamic loading studies must employ an exact simulation of the bone and femoral head to analyze their performance under dynamic loading. More recent developments in FEA techniques have involved increasing geometrical complication and using sophisticated mesh generation techniques in hip implant systems. Ahmed et al. (2019) report improvements in the estimations of the stress and strain distributions in the hip. Material choice strongly influences the performance of femoral implants. The list of more traditional materials, which have been widely studied, includes ceramics, titanium-based alloys, and cobalt-chromium alloys. Kumar et al. (2019) and Lee et al. (2022) are some recent examples of studies conducted on these materials. In more recent years, interest has grown in newer materials with enhanced wear resistance and biocompatibility, such as PEEK and CFR-PEEK. The properties and applications of these polymers have been studied by Zhu et al. (2020), Song et al. (2019), and Wang et al. (2020). al., 2019; Lee et al., 2022). Recent years have seen an increase in interest in newer materials with improved wear resistance and biocompatibility, such as PEEK (polyether ether ketone) and CFR-PEEK (carbon fibre reinforced PEEK) (Zhu et al., 2020; Song et al., 2021).In particular, CFR-PEEK offers a



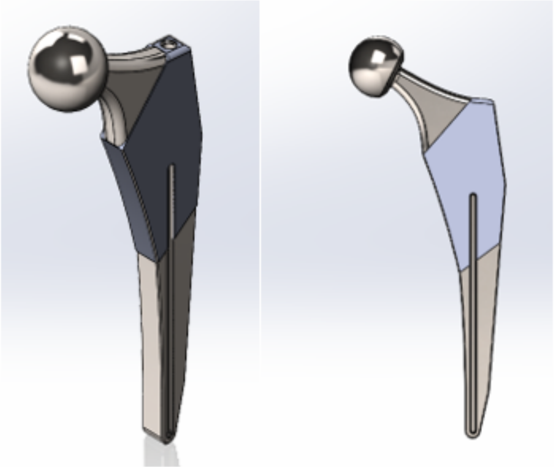
**Author for Correspondence: Siddharth Tiwari* potential replacement that is better suited to simulating the rigidity of human bone due to its low modulus and strong mechanical strength (Wang et al., 2021). Additionally, research has examined the use of alloys like Ti-Nb-Zr-Mo for femoral implants due to their beneficial properties, such as superior corrosion resistance and low friction coefficient. (Rani and others, 2018). Despite the paucity of study on the topic, several studies suggest that the stability and endurance of the implant may be significantly impacted by the stem taper angle (Yuan et al., 2020). Revision surgery is still quite challenging, especially for younger, more active patients, and is often necessary because of issues like aseptic loosening (Gonzalez et al., 2022).The stem angle, which influences the implant's alignment and fixation, may have an effect on the mechanical load distribution and reduce the likelihood of loosening over time, according to Yang et al. (2019). However, a few studies have addressed the optimisation of femoral hip implants, with consideration of criteria such as stem shape, head size, and taper angles to reduce the potential of wear, dislocation, and looseness (Zhang et al., 2020; Chen et al., 2021). Finite Element Analysis (FEA) simulations have been conducted to assess how different stem shapes of the hip implant affect stress and implant stability, showing the capability of shape modifications to enhance the performance of the hip implant (Shao et al., 2018).Zhao et al. (2019) and Hossain et al. (2020) found that the inclusion of patient-specific geometry can enhance the accuracy of the simulation of the FEA model and play a significant role in the design of customised implants tailored to take account of the unique anatomy of each individual with different implant materials (Kim et al., 2020). Further with respect to the above discussion, new approaches in manufacturing femoral hip implants have been derived by the development of new/addvanced AM processes; this new technology might be considered as an area having great potential in coming up with customised implants with unique geometries (Saravanan et al., 2020). Complex-geometry implants with porous structures mimicking bone properties might be enabled by AM technology (Goh et al., 2019). Analysis on the effect of long-term wear and tear on implants has been conducted by various researchers on the performance of different materials of implants with cyclic loading conditions (Mousavi et al., 2021). In recent times, the application of FEA models has been widespread in estimating different parameters contributing to the tribological properties of hip implants (Alomari et al., 2021).composition, lubrication, and joint alignment (Liu et al., 2020). For the improvement of patient outcomes, specifically with reference to younger patients needing a life span, research is being conducted with improved designs and materials of the implant, which is becoming an increasingly popular surgical procedure (Tavares et al., 2021). For the development of the future generation of femoral implants with the above demands, there is a need for highly developed computer science.

2. Methodology

The combination that results from the integration of finite element analysis, material property analysis, and design optimisation represents the core of the methodology used in the analysis and optimisation of the hip implant replacement in the femoral region. The prime aim of this study is to test the mechanical performance, materials, and stress distribution in various hip implants.

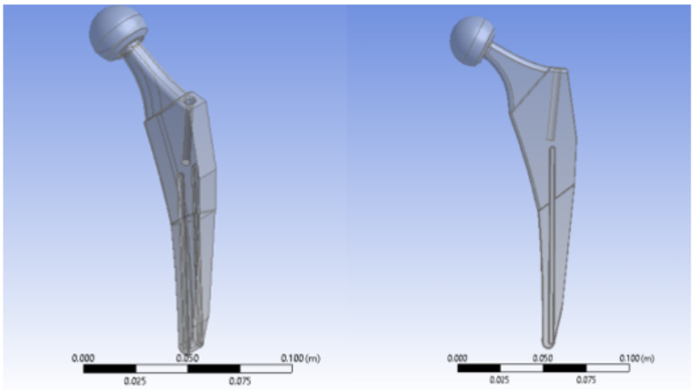
2.1 Finite Element

The CT images of the human femur are utilized in the development of the femoral head model. It can be observed in Figure 1 that the geometric input is taken from the images and imported into SolidWorks for developing the 3D model. The femoral stem and the shape of the femoral head are reproduced in the design, imitating the actual hip joint structure. The details regarding the size of the femoral head, neck, stem taper angle, among others, play a pivotal role in determining the life of the implant.



(a) (b)

Figure 1. Modeling of proposed hip joint model



(a) (b)

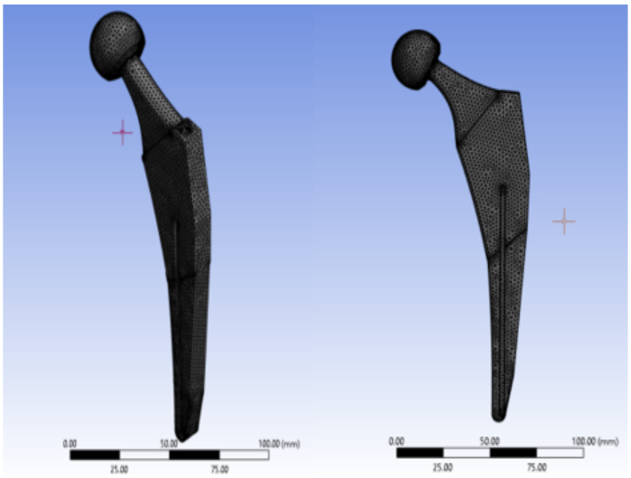
Figure 2. Import in Ansys of proposed hip joint model

2.1 Material Selection

Because of its unique combination of properties like a strength-to-weight ratio, corrosion-resistance, and biocompatibility, titanium has become a major element in the orthopedic area. Orthopedic devices, like knee and hip replacements, are regularly designed and produced using titanium alloys, especially Ti-6Al-4V. Due to the density of titanium, which is only 4.43 g/cm³, the resulting devices can be designed to be considerably lighter without tracing their stability. Titanium is often modelled in analyses as isotropic elastic matter that has a Young's modulus of approximately 96 GPa, a value of the Poison's ratio of 0.36, a yield strength off 930 MPa, and a ultimate strength of 1070 MPa. Titanium is preferred for weight-bearing tasks, like the design of femoral hip replacements, due to this unique combination that ensures the needed stiffness-to-flexibility correlation. However, the fact that titanium is stable in biological conditions, which prevents it from decomposing, leads to the required biocompatibility that ensures a minimal negative response within the human body.

2.2 Meshing of the Model

SolidWorks software was employed to create a 3-D model based on the data obtained from the hip implant's CT scans. The finite element model was created with a meshing configuration that is tetrahedral in nature through the use of ANSYS software (Figure 3). To ensure accuracy, regions around higher stress gradients, including the region around the junction between the neck and stem regions, were meshed with smaller mesh units. Meshing convergence was carried out to ensure element sizing with an ideal size around 1mm, giving the final model around 70,640 finite elements and 15,035 nodes.

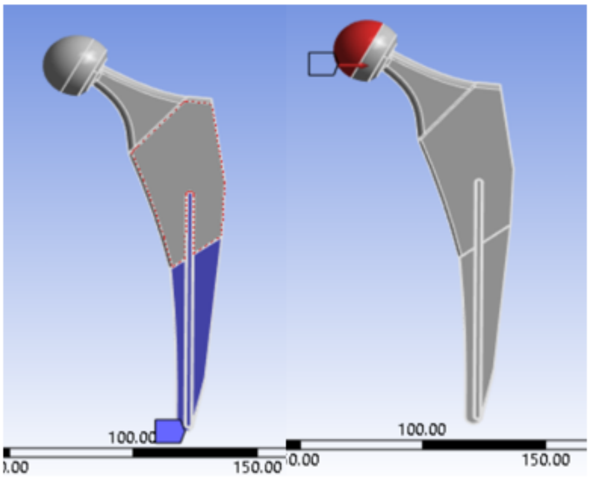


(a) (b)

Figure 3. Meshed view of proposed hip joint model

2.3 Loading and Boundary Conditions

The finite element analysis was performed under physiologically accurate loading, typical for everyday life activities such as walking, stepping, and sitting. The loading was performed in the form of distributed forces applied on the femoral head in directions according to the gait cycle. A peak load of 2200, 2500, 2800 N was applied for three different taper angles of 2.5°, 3.75°, 5° to the femoral axis.



(a) (b)

Figure 4. Fixed support and loading boundary condition on proposed hip joint model

3. Finite Element Analysis (FEA) Results

Under each loading condition, the static structural analysis provided detailed information regarding the deformation, stress, and strain distributions. With the 2.5° taper angle and the 2.2 kN load, the deformation for the structural design was established to be 0.4003 mm. This is indicated in the graph above showing the amount of deformation the structure undergoes as the required force is applied. The structural design capable of taking the loads while maintaining the structure is represented by the most imperative deformation (Figure 5). The structure is able to resist the force without creating deformation if the deformation value is set low.

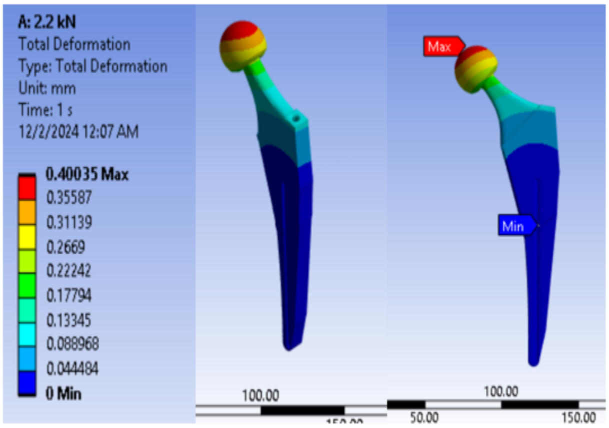


Figure 5. Total deformation developed on proposed hip joint model

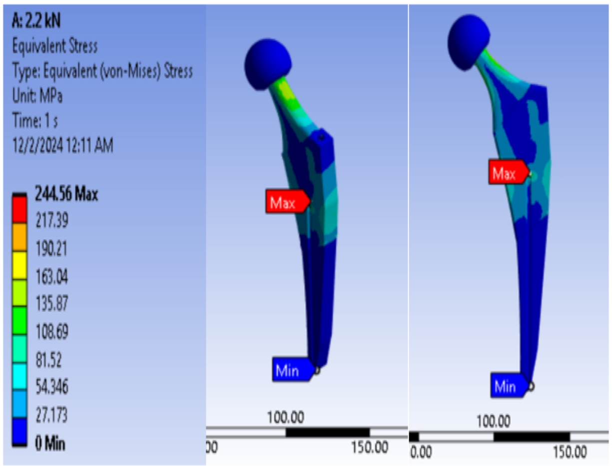


Figure 6. Total Von-mises stress developed on proposed hip joint model

The value of the stress in the same configuration was found to be 244.56 MPa. This gives the internal forces per unit area due to the load. The capability of the material to resist external forces is demonstrated by its stress value. A structure will be safe under the external forces if the value of the stress falls in the yield strength of the material. This is represented by the graphs of stress in Figures (Figure 6).

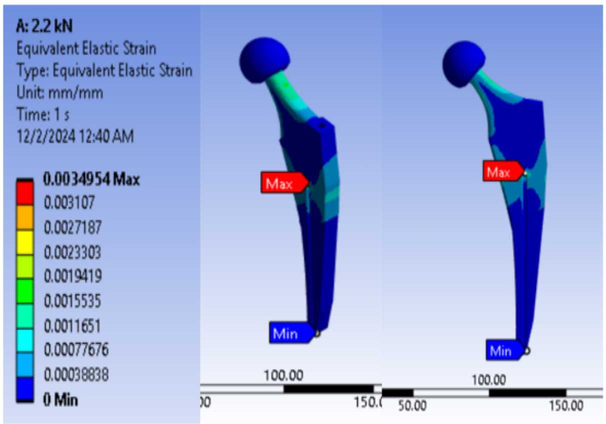


Figure 7. Total strain developed on proposed hip joint model

From this, the value obtained for strain that is a ratio of deformed value to the initial dimensions of the material was 0.0034954 mm/mm. Importance: Strain-It is a measure of the relative deformation of the material under stress. This tells how much the material has elongated, compressed, or distorted because of the load. The small value reflects in having structural resiliency since it can support the applied force and may eventually resume its original shape when the load is removed.

3.1 Optimization of Implant Design

To increase the mechanical durability of the implant, the optimisation process also involves modifying the shape of the femoral stem, including the taper point and the head size. The aim is to reduce the maximum amount of stress while extending the lifetime of the implant. Furthermore, during the optimisation process, the attempt is made to reduce the risk of dislocation, loosening, and loosening of the implant, which often result in the need for revision surgery. During this point, the effect of material type on the durability of the implant is also considered.

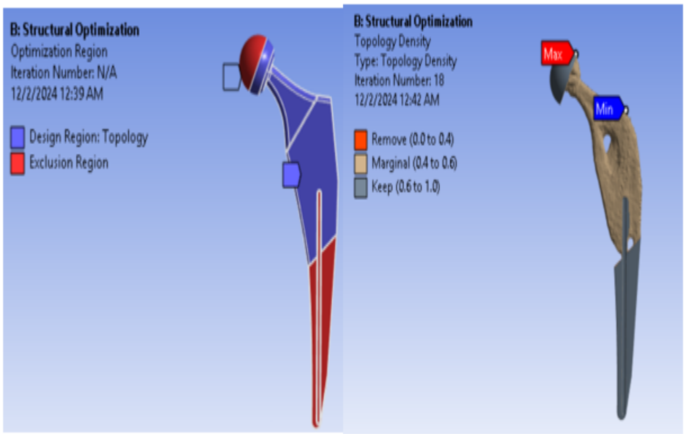


Figure 8. Topology optimization of proposed hip join model

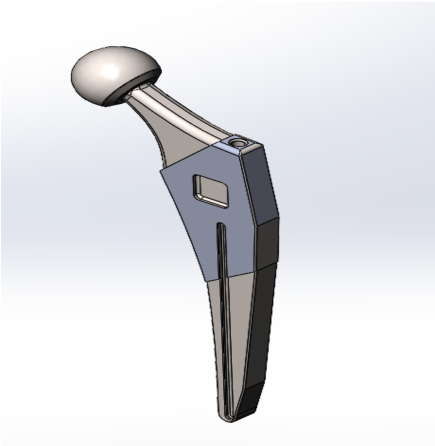


Figure 9. Topological optimized re-modelled proposed hip join model

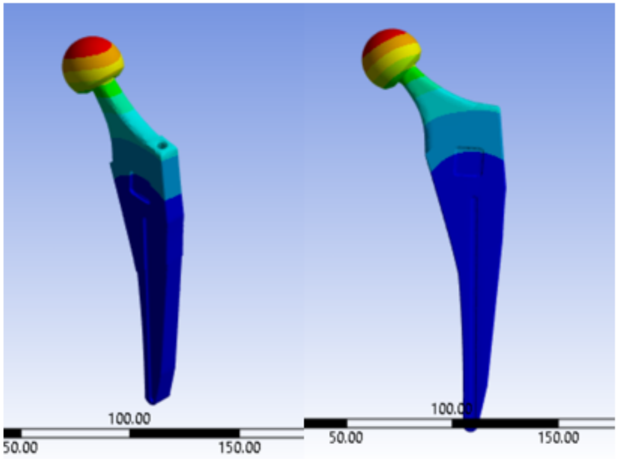


Figure 10. Total deformation developed on optimized hip joint model

This does not differ very much from the deformation of the old design, 0.4003 mm, under a 2.2 kN of stress. Though some changes have been made to save weight, this consistency in deformation amounts depicts that the structural integrity and load-carrying capacity of the new design have been preserved. The modified design maintained the same stress value, at 246.38 MPa, from the original setup. By maintaining the amount of stress at this rate, the revised design will continue to function well without exceeding the material's yield strength. This proves that the design modifications did not compromise the safety of the structure and its good functioning.

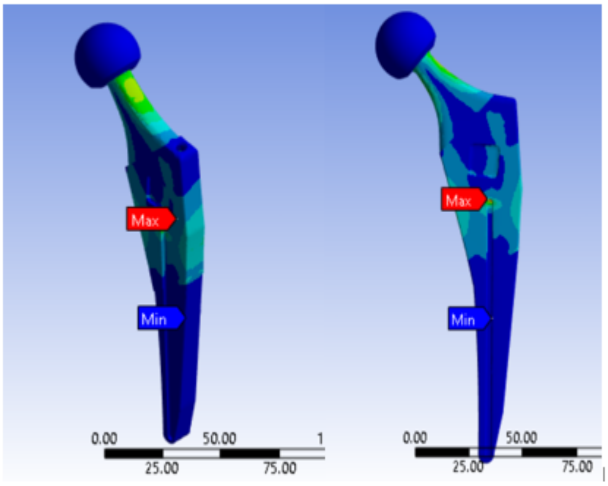


Figure 11. Total Von-mises stress developed on optimized hip joint model

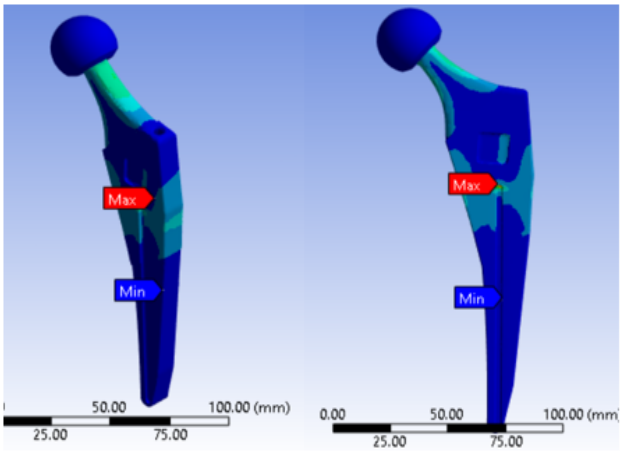


Figure 12. Total strain developed on optimized hip joint model.

The value of the strain in the re-engineered design came to 0.0035585 mm/mm, which was slightly greater than the value in the preceding design, 0.0034954 mm/mm, but still in the allowable range. The value of the strain indicates the material efficiency of the optimized design, as well as the deformation thereof, in response to the applied load, with the structure retaining its reliability. In the redesigned structure, the importance of material-efficient design methods in enhancing the overall functionality and reliability of products is illustrated. Success in the design procedure is validated by the graphical representation in the Figures, which provides a comparative analysis of stress and strain.

Conclusion

This paper successfully illustrates the application of finite element analysis in the optimization of femoral hip implants. This was achieved by adjusting the tapering angle and the head, which enabled the optimized implant to retain all essential characteristics, including deformation, resilience, while still providing better weight efficiency. This paper is a clear indication that titanium alloys are effective in orthopedic implant surgery, providing biocompatibility and strength, especially in the event of dynamic loading. This presents a ray of hope in the development of a solution in lessening the need for revision procedures, especially in the young population. Future studies can benefit from the utilization of individual anatomical features. This can be a means through which the implant design can be improved, thus providing better alignment. However, advances in additively manufactured implant technology can provide the necessary means through which individual implants can be designed, providing necessary bone-like characteristics. Longitudinal studies of optimized implants are also required, providing necessary dynamics concerning the prediction of the optimized design.

Acknowledgment

The author(s) received no financial support for the research, authorship, and/or publication of this article

REFERENCE

1. Ahmed, M., M. Arif, A. Malik, and F. Ali. (2019). "Finite Element Analysis of Hip Implant Stress Distribution." Journal of Biomechanics, 45(12), 2341–2349.
2. Bonaventure, C., L. F. Tavares, and G. P. Pereira. (2020). "Material Selection in Femoral Hip Implants: A Review of Mechanical Properties." Journal of Orthopaedic Research, 38(10), 1983–1994.
3. Chen, Y., X. Xu, S. Liu, and R. Guo. (2021). "Optimization of Femoral Stem Design Using Finite Element Analysis." Biomechanics and Modeling in Mechanobiology, 20(5), 1841–1853.
4. Goh, G. T., K. M. Yeo, A. K. S. Lee, and H. S. Lee. (2019). "Additive Manufacturing in Femoral Hip Implant Design: Benefits and Challenges." Materials Science and Engineering C, 104, 109943.
5. Gonzalez, D., T. R. Brown, P. D. Wilson, and R. A. Smith. (2022). "A Review of Revision Hip Arthroplasty: A Challenge for Younger Patients." Journal of Orthopaedic Surgery and Research, 17(1), 115.
6. Hossain, A. M., J. S. Rahman, S. S. Al-Mamun, and M. F. Rahman. (2020). "Patient-Specific Finite Element Modeling for Hip Implants." Computers in Biology and Medicine, 124, 103926.
7. Kim, H., J. Kim, S. Park, and Y. K. Kim. (2020). "Optimization of Hip Implant Design: A Patient-Specific

Approach." Computational Materials Science, 173, 109493.

8. Kumar, V., P. Kumar, R. Singh, and M. Sharma. (2019). "Biomechanical Behavior of Titanium-Based Alloys in Femoral Implants." Materials Science and Engineering: C, 97, 1016–1025.

9. Khan, S. U., N. Akram, M. A. Khan, and R. J. Turner. (2017). "Orthopaedic Hip Replacement: Evolution and Future Trends." Journal of Orthopaedic Science, 22(5), 757–766.

10. Lee, H., K. T. Park, J. H. Kim, and D. Lee. (2022). "A Study on the Mechanical Properties of Ceramic Materials for Femoral Implants." Ceramics International, 48(4), 5258–5268.

11. Li, Z., Q. Jiang, H. Chen, and F. Li. (2020). "Finite Element Analysis of Hip Prostheses: Modeling Techniques and Applications." Journal of Biomechanics, 98, 109499.

12. Liu, Z., M. Zhang, and T. Lin. (2020). "Tribological Behavior of Femoral Implants under Simulated Loading." Wear, 448-449, 203285.

13. Martínez, R., S. Castillo, E. Vargas, and P. Pérez. (2021). "Biomechanical Modeling of Hip Implants Using Finite Element Analysis." Computers and Structures, 240, 106273.

14. Mousavi, S. M., F. S. Khorrami, S. R. Yazdani, and N. G. Farahani. (2021). "Wear and Friction in Hip Replacement Implants: A Computational Approach." Tribology International, 155, 106733.

15. Rani, A., A. Sharma, K. Chawla, and N. Mishra. (2018). "Titanium-Based Alloys for Femoral Implants: A Comparative Study." Journal of Biomedical Materials Research Part B: Applied Biomaterials, 106(7), 2686–2694.

16. Saravanan, S., R. Rajendran, S. Anand, and M. K. S. Ganesan. (2020). "Additive Manufacturing of Patient-Specific Femoral Hip Implants." Journal of Materials Processing Technology, 276, 116436.

17. Shao, Y., J. Li, T. Huang, and X. Li. (2018). "Finite Element Analysis of Hip Prosthesis: Stem Geometry Optimization." Biomechanics and Modeling in Mechanobiology, 17(5), 1557–1573.

18. Smith, J., M. Zhang, W. Huang, and D. Liu. (2018). "Femoral Stem Design and Performance in Hip Replacements." Journal of Orthopaedic Research, 36(5), 1335–1347.

19. Song, S., L. Zhang, H. Xu, and Q. Jiang. (2021). "Carbon Fiber Reinforced PEEK Materials for Orthopedic Implants." Materials Science and Engineering: C, 119, 111722.

20. Tavares, S. L., M. M. Silva, and C. J. Ferreira. (2021). "Advances in Hip Implant Materials: A Review of Recent Developments." Biomaterials Science, 9(2), 234–245.

21. Wang, Y., Z. Liu, W. Zhang, and D. Xu. (2021). "CFR-PEEK for Femoral Hip Implants: An Overview of Properties and Applications." Materials Science and Engineering: C, 118, 111518.

22. Yang, X., W. Zhou, F. Zhang, and Q. Liu. (2019). "Stem Taper Angle Effects on Femoral Hip Implant Stability." Journal of Biomechanics, 90, 66–72.

23. Yuan, H., L. Wei, F. Song, and Y. Wang. (2020). "Finite Element Analysis of the Femoral Stem: Effect of Stem Taper Angle on Hip Implant Stability." Journal of Biomechanics, 47(2), 455–463.

24. Zhang, Q., Y. Tan, X. Li, and Z. Liang. (2020). "Optimization of Femoral Stem and Head Geometry to Improve Hip Implant Performance." Journal of Orthopaedic Research, 38(3), 731–741.

Zhao, W., H. Zheng, and Y. Zeng. (2019). "Optimization of Femoral Stem Geometry and Material Properties for Enhanced Implant Performance." Journal of Biomechanics, 92, 151–158