





PAPER

Design Optimization of Femoral Hip Implants with Circular Perforations: Structural and Fatigue Evaluation Using Finite Element Analysis

José L. Serna-Landivar¹  ,
Daniela M. Anticona-Valderrama², William C. Algoner¹ , Andrés Eduardo De La Cruz Herrera³, Marleni Mendoza Damas¹ , Jeferson Rayu Vega Zavala¹

¹Universidad Tecnológica del Perú, Lima, Peru

²Universidad Privada del Norte, Lima, Peru

³Universidad César Vallejo, Lima, Peru

U18101283@utp.edu.pe

ABSTRACT

Total hip arthroplasty (THA) is an effective solution for osteoarthritis, but the challenge of designing lightweight implants without compromising their strength remains. This study evaluates the influence of circular perforations in the design of femoral stems using finite element simulations in ANSYS Workbench. Eight geometric configurations were modeled based on a standard stem, incorporating circular perforations of varying diameters in five strategic locations. The analysis considered two materials widely used in orthopedics (Ti-6Al-4V and CoCr alloy) and applied physiological loading conditions according to ASTM F2996-20 and ISO 7206-4 standards. Boundary conditions included distal constraint and an axial load of 2300 N on the femoral head. Total deformation, equivalent stresses, static safety factors, and fatigue resistance according to the Soderberg theory were evaluated. The results showed that, although the V2 and V4 configurations were the most suitable, the titanium V2 design emerged as the best alternative, achieving the greatest mass reduction (19.7%) while maintaining acceptable safety margins. This paper offers a viable and low-cost solution for the design of more sustainable prostheses aligned with the sustainable development goals (SDG 3).

KEYWORDS

stem implant, hip prosthesis, finite element analysis (FEA), prostheses, optimization

1 INTRODUCTION

Hip osteoarthritis is one of the leading causes of disability among the adult population, with a growing global trend that has resulted in a significant increase in total hip arthroplasties (THA) performed over the past decades [1]. This surgical intervention has proven effective in relieving pain, improving functionality, and restoring patients' quality of life [2]. Nevertheless, the durability and mechanical

Serna-Landivar, J. L., Anticona-Valderrama, D. M., Algoner, W. C., De La Cruz Herrera, A. E., Mendoza Damas, M. Vega Zavala, J. R. (2025). Design Optimization of Femoral Hip Implants with Circular Perforations: Structural and Fatigue Evaluation Using Finite Element Analysis. *International Journal of Online and Biomedical Engineering (iJOE)*, 21(12), pp. 106–120. <https://doi.org/10.3991/ijoe.v21i12.57489>

Article submitted 2025-07-03. Revision uploaded 2025-08-11. Final acceptance 2025-08-11.

© 2025 by the authors of this article. Published under CC-BY.

strength of implants remain critical issues, especially in young or active patients, where the goal is to extend the prosthesis lifespan beyond the usual 15–20 years [3].

One of the main biomechanical challenges in femoral implants is the phenomenon of stress shielding [4], caused by the discrepancy in the modulus of elasticity between the bone and the metal implant, which can lead to proximal bone loss and long-term failures [5], [6]. Several studies have shown that stem geometry, material selection, and its internal distribution directly influence the mechanical response under static and dynamic loading conditions. [7], [8]. In this context, the use of finite element analysis (FEA) is a method for evaluating the design and predicting the mechanical behavior of femoral prostheses under different loading scenarios that are closest to real-life conditions [9].

One of the most relevant strategies for improving the structural and functional efficiency of femoral stems is reducing their weight through geometric modifications that preserve the mechanical strength of the implant. In the current literature, the use of ovoid cavities, porous structures, and especially topological optimization techniques has been widely explored to achieve lighter and biomechanically compatible designs [4], [10], [11], [12], [13], [14], [15]. However, these approaches often require metal additive manufacturing, a costly process that remains largely inaccessible in many clinical or industrial settings. Although circular perforations represent a geometric alternative that is easier to implement, since they can be produced using conventional processes with machine tools, there is a notable lack of studies evaluating their influence through parametric optimization. This gap in the scientific literature limits the development of viable solutions for contexts where metal 3D printing is not available. Therefore, it is necessary to investigate the impact of such perforations on stiffness, maximum stress, fatigue resistance, and safety factors in order to validate their structural and functional effectiveness. Moreover, research in other applied technical fields has demonstrated the usefulness of incorporating behavior-focused analysis and iterative refinements to improve the performance of complex systems under real-world constraints [16].

The present study aims to evaluate, through finite element analysis in ANSYS, the impact of different geometric configurations of circular perforations in the femoral stem on its static and fatigue behavior. Two materials commonly used in orthopedic prostheses were considered, Ti-6Al-4V titanium alloy and cobalt-chromium alloy, to compare their structural strength. The methodology includes analysis of deformation, stress distribution, safety factor, and fatigue life, with the goal of proposing an optimized design that contributes to improving the structural strength of the implant, in line with sustainability principles and sustainable development goal 3 (good health and well-being).

2 MATERIALS AND METHODS

2.1 Geometric design of the femoral stem implant

The femoral stem was modeled in CAD software SolidWorks with anatomy that reproduces the natural curvature of the femur's intramedullary canal. Its three-dimensional geometry was designed to achieve a proper fit and stability within the bone cavity, allowing for load transfer and functional integration with the bone [17]. The design aims to adapt to the patient's morphology, contributing to stable fixation and the long-term performance of the implant.

One of the main biomechanical features of this proposal is its ability to evenly distribute physiological loads during dynamic activities, avoiding areas of stress

concentration that could lead to bone resorption or implant loosening. By replicating the natural load transfer patterns of a healthy femur, it promotes the preservation of bone density and minimizes complications associated with stress shielding [17], [18].

This stem, without modifications or cavities, was considered the baseline model for this study. From this version, the geometric variants were developed with the aim of evaluating the structural strength of a weight-reduction strategy using perforations (see Figure 1). The baseline stem model served as the foundation for this study, representing a standard design without any alterations or hollows. This unmodified version provided a crucial reference point for comparing the effects of subsequent geometric variations. As part of the present study, geometric variants of the femoral stem were developed based on the previously designed base model, which did not feature any cavities or perforations.

The primary objective of these geometric variants was to assess the impact of perforations on the structural integrity of the stem. By introducing carefully designed cavities or holes into the stem structure, the researchers aimed to reduce overall weight while maintaining essential mechanical properties. This approach allowed for a comprehensive evaluation of how different perforation patterns and configurations affected the stem's strength, stability, and load-bearing capacity. The various designs, as illustrated in Figure 1, likely encompassed a range of perforation sizes, shapes, and distributions, enabling a thorough analysis of the trade-offs between weight reduction and structural performance.

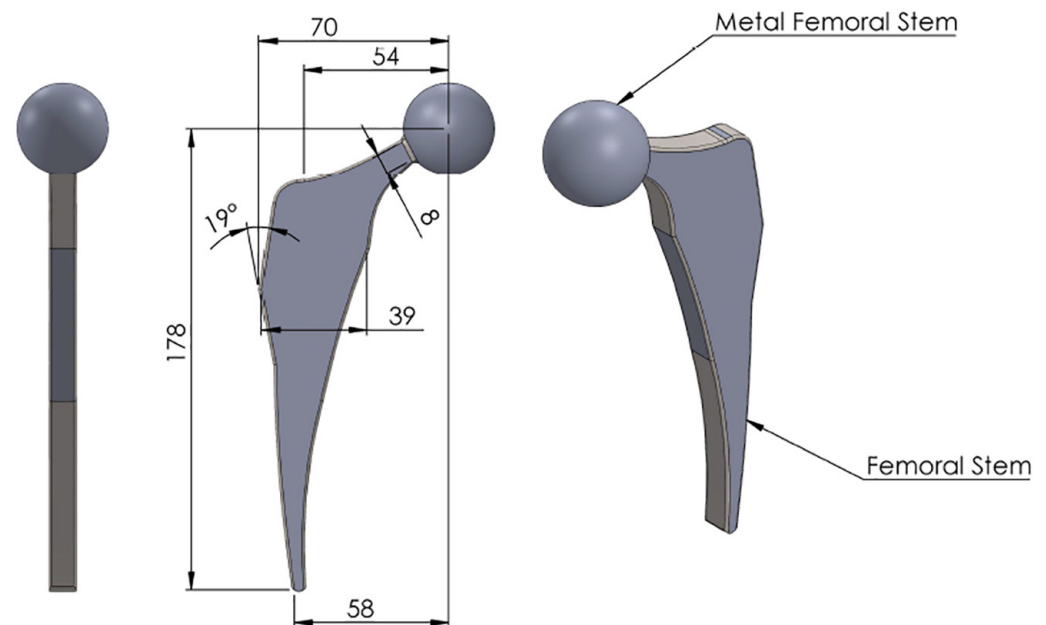


Fig. 1. Conceptual model of the modular femoral stem of the hip implant

2.2 Modified femoral stem with perforations

With the aim of optimizing the structural design of the implant and reducing both its mass and material consumption, a geometric modification of the femoral stem was proposed by introducing circular perforations. These were distributed along the longitudinal axis of the stem body at five strategic locations, designated D1 to D5, situated at 50 mm, 75 mm, 100 mm, 125 mm, and 150 mm from the bottom base of the implant, respectively, as shown in Figure 2.

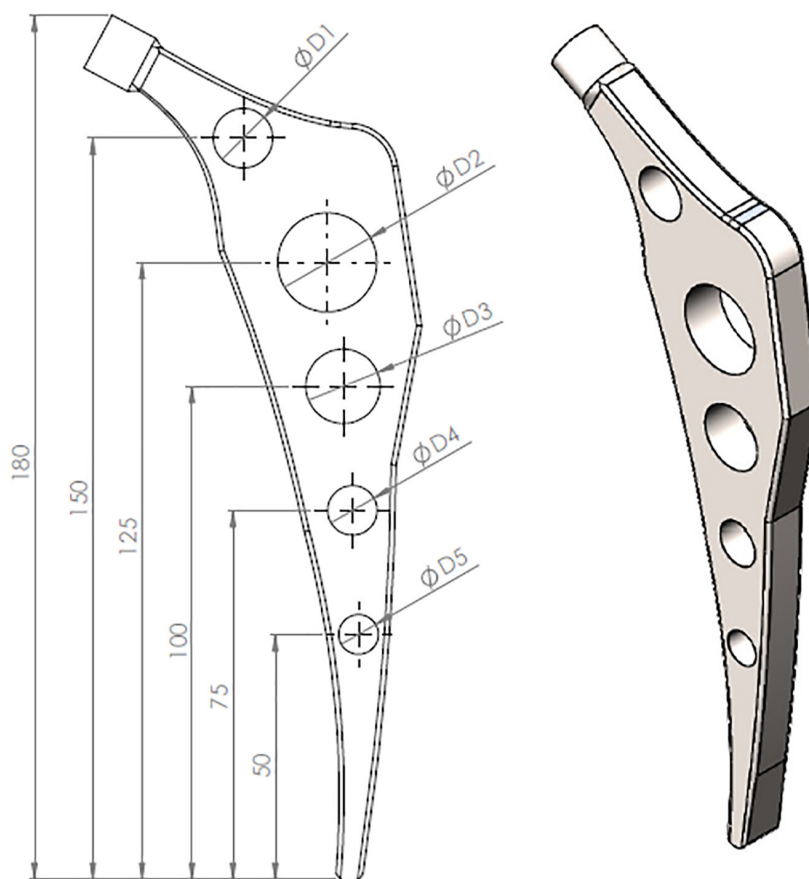


Fig. 2. Representation of the modified femoral stem with the five perforations (D1–D5)

Eight geometric configurations (V1 to V8) were defined, combining two materials used in orthopedic implants with the inclusion or absence of five circular holes distributed axially. Configurations V1 and V5 correspond to the baseline models without perforations for each material, while the other versions incorporate holes in strategic positions, with diameters that progressively decrease from the proximal to the distal region. This geometric variability allows for the evaluation of the impact of the perforations on the structural integrity of the implant and its weight reduction without compromising its mechanical strength. Table 1 presents the details of each iteration, including the type of material, the number of perforations, and the variable diameters.

Table 1. Description of the geometric configurations evaluated for the femoral stem

Iteration	Material	No. of Holes	D1 (mm)	D2 (mm)	D3 (mm)	D4 (mm)	D5 (mm)
V1	Titanium alloy Ti-6Al-4V	0	–	–	–	–	–
V2		5	12	20	15	10	8
V3		5	10	18	13	8	6
V4		5	8	16	11	6	4
V5	CoCr alloy	0	–	–	–	–	–
V6		5	12	20	15	10	8
V7		5	10	18	13	8	6
V8		5	8	16	11	6	4

2.3 Materials

For the analysis of the femoral stem, two biomaterials commonly used in orthopedic applications were selected due to their high biocompatibility, corrosion resistance, and good mechanical performance: titanium alloy (Ti-6Al-4V) and cobalt-chromium alloy (CoCr alloy) [19], [20]. Both materials have demonstrated good outcomes in load-bearing implants, such as joint prostheses. Table 2 summarizes the mechanical properties used in the finite element simulation, which were obtained from scientific literature. These properties are important for accurately representing the structural behavior of the implant under the applied loading conditions.

Table 2. Mechanical properties of the biomedical materials used in the study

Material	Modulus of Elasticity (GPa)	Yield Strength (MPa)	Ultimate Tensile Strength (MPa)	Poisson Ratio	Density (Kg/m ³)	References
CoCr alloy	200	612	1503	0.3	8500	[2], [21], [22]
Ti-6Al-4V	114	880	930	0.31	4500	[5], [21], [22] [23], [24], [25], [26], [27], [28]

Regarding fatigue resistance, bibliographic information was collected on the stress–number of cycles (S–N) curves for Ti-6Al-4V and CoCr alloy materials. These curves make it possible to represent the behavior of materials under cyclic loads, which is important for assessing their resistance under real conditions. The references used to obtain this data come from previous studies [2], [7], [21] widely cited in the scientific literature. The corresponding S–N curves are shown in Figure 3.

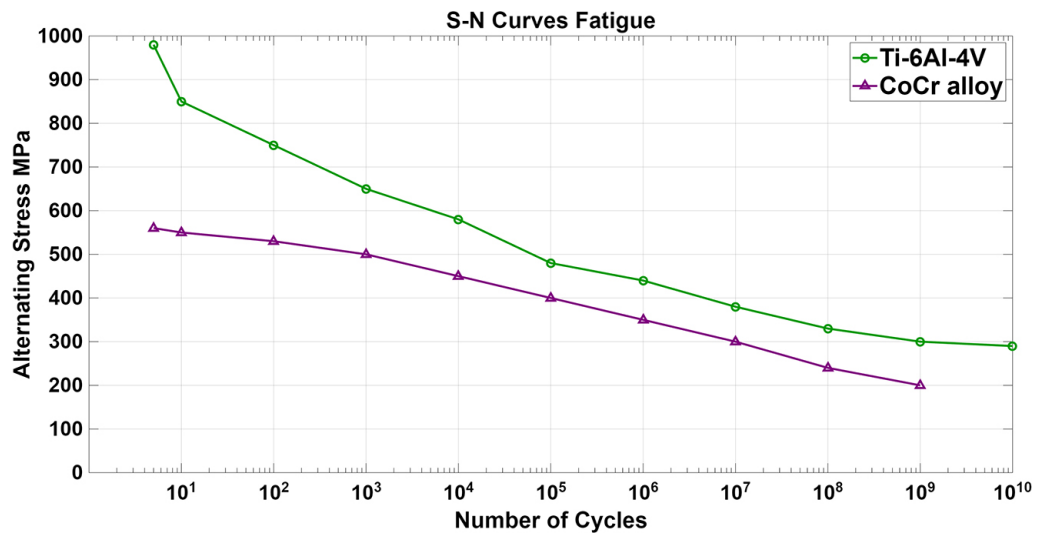


Fig. 3. S–N fatigue curves of Ti-6Al-4V and CoCr alloy materials

2.4 Boundary conditions and meshing

To define the boundary conditions of the femoral stem in the finite element analysis, the guidelines of ASTM F2996-20 standards were adopted [29] and ISO 7206-4:2010 [30], which standardize both the fixation and the load applied in

structural tests of implants. The distal end of the stem was completely restricted to simulate its fixation to the bone, while an axial load of 2300 N was applied to the femoral head, perpendicular to the axis of the implant. Figure 4 shows the application of these conditions in the original and modified models. This methodology has been employed and validated in previous studies related to the analysis of orthopedic stems [31], [32], [33], [34], [35].

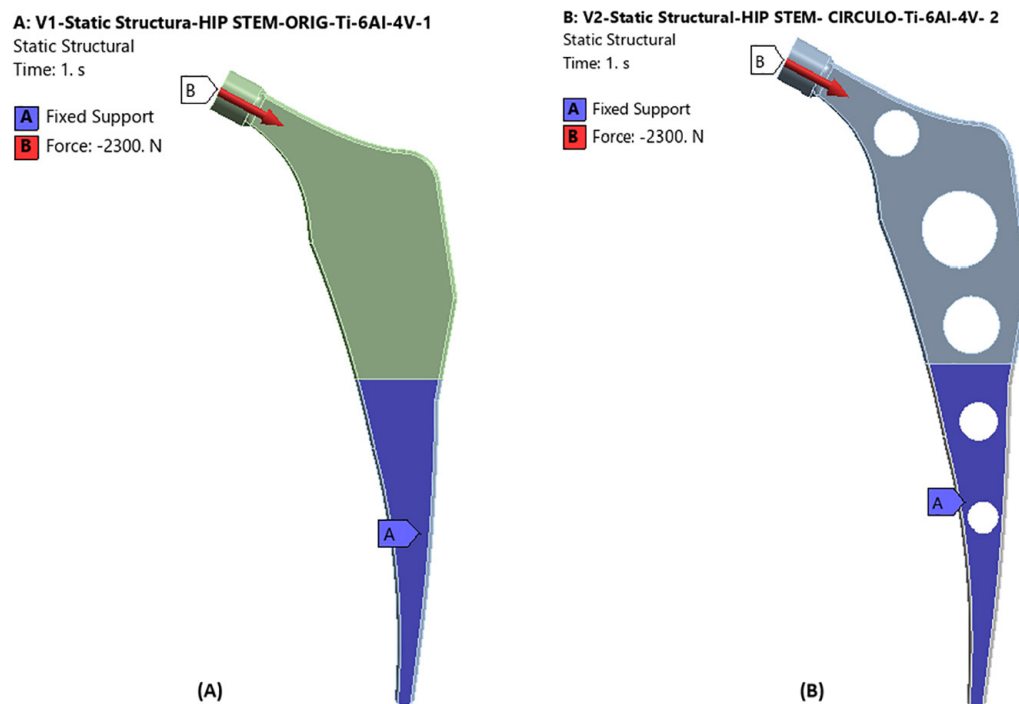


Fig. 4. Boundary conditions and load applied to the femoral stem in the finite element analysis: A) Original stem and B) Modified stem

A mesh convergence analysis was carried out with the aim of ensuring the numerical validity of the results obtained in the finite element simulation. For this purpose, a uniform element size of 1 mm was used, and the relationship between the total number of nodes and the maximum von Mises stress was evaluated. The original femoral stem model reached a final discretization of 15,900 nodes, while the modified model with perforations averaged 19,980 nodes due to the greater geometric complexity introduced by the circular cavities, which generated additional internal surfaces and curved edges that required a denser mesh. Despite the reduction in material volume, this complexity increased the number of elements needed for an accurate representation. In both cases, the average element quality exceeded 0.92, demonstrating reliable meshing. Figure 5 summarizes this process: (A) shows the refined mesh of the original stem, (B) the modified model with perforations, and (C) presents the convergence curve, where it can be seen that, as the number of nodes increases, the von Mises stress values tend to stabilize, validating the adequacy of the meshes used in the study.

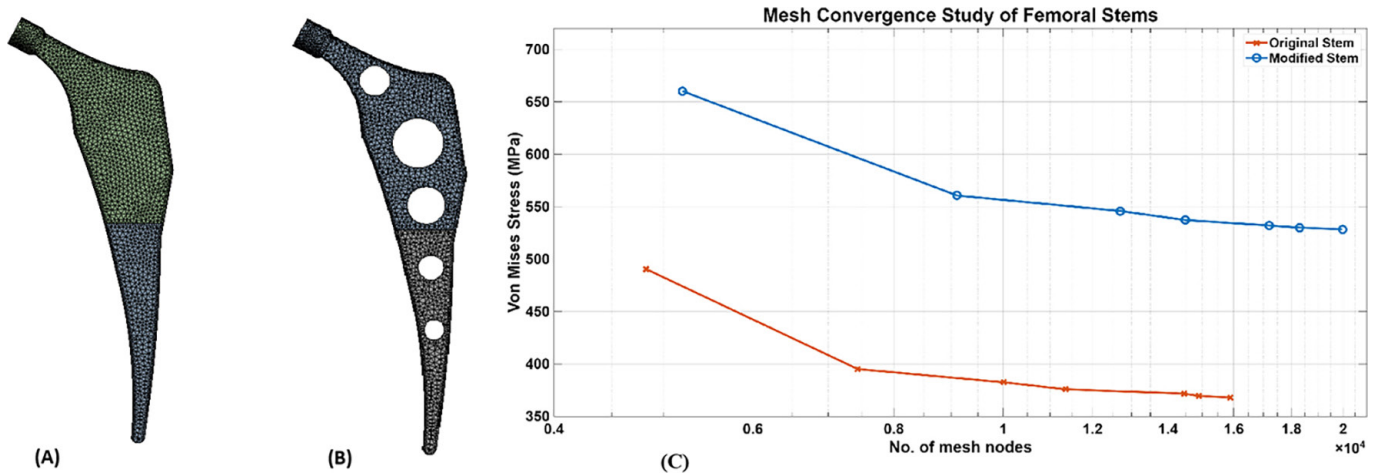


Fig. 5. Mesh convergence analysis and refined meshing of the femoral stems: (A) Mesh of the original stem, (B) Mesh of the modified stem with perforations, and (C) Convergence curve of von Mises stress as a function of the number of nodes

2.5 Static analysis

The static analysis was carried out using the finite element method with the ANSYS Workbench software. This analysis made it possible to determine the distribution of stresses, deformations, and the safety factor of the femoral stem under physiological loading conditions. A linear-elastic material behavior was assumed, using the following equation: [36], [37].

$$\{F\} = [K]\{u\} \tag{1}$$

Where, $\{F\}$ is the force vector, $[K]$ is the stiffness matrix y $\{u\}$: displacement vector.

2.6 Fatigue analysis

To evaluate the durability of the femoral stem under cyclic loads, a fatigue analysis was carried out based on the Soderberg theory. This theory is known for its conservative nature, as it combines the alternating stress σ_a and the average effort σ_m in relation to the yield strength of the material S_y and the fatigue strength limit S_e , according to the following expression:

$$\left(\frac{\sigma_a}{S_e}\right) + \left(\frac{\sigma_m}{S_y}\right) = \frac{1}{N} \tag{3}$$

Where:

N : Safety Factor

σ_a : Alternating Stress

S_e : Fatigue Limit Resistance

σ_m : Average Stress

S_y : Material yield Strength

Fatigue analysis, complemented by the results of static analysis, makes it possible to identify optimal geometric configurations that reduce mass without compromising the strength or safety of the implant [21].

3 RESULTS AND DISCUSSIONS

3.1 Static analysis

Figure 6 shows the total deformation obtained for configurations V1–V4 in two types of alloys: CoCr (top) and Ti-6Al-4V (bottom). In both cases, it can be observed that the models without perforations (V1) exhibit the lowest maximum deformations, while these values progressively increase in the optimized configurations (V2, V3, and V4), due to the reduction in stiffness caused by the presence of holes. In the case of Ti-6Al-4V, the deformation increased from 0.142 mm (V1) to 0.2238 mm (V2), and for the CoCr alloy, it went from 0.0916 mm (V1) to 0.119 mm (V2). Despite the increase in deformation in the lightweight models, all remained within the elastic regime, ensuring safe structural behavior without risk of plastic deformation.

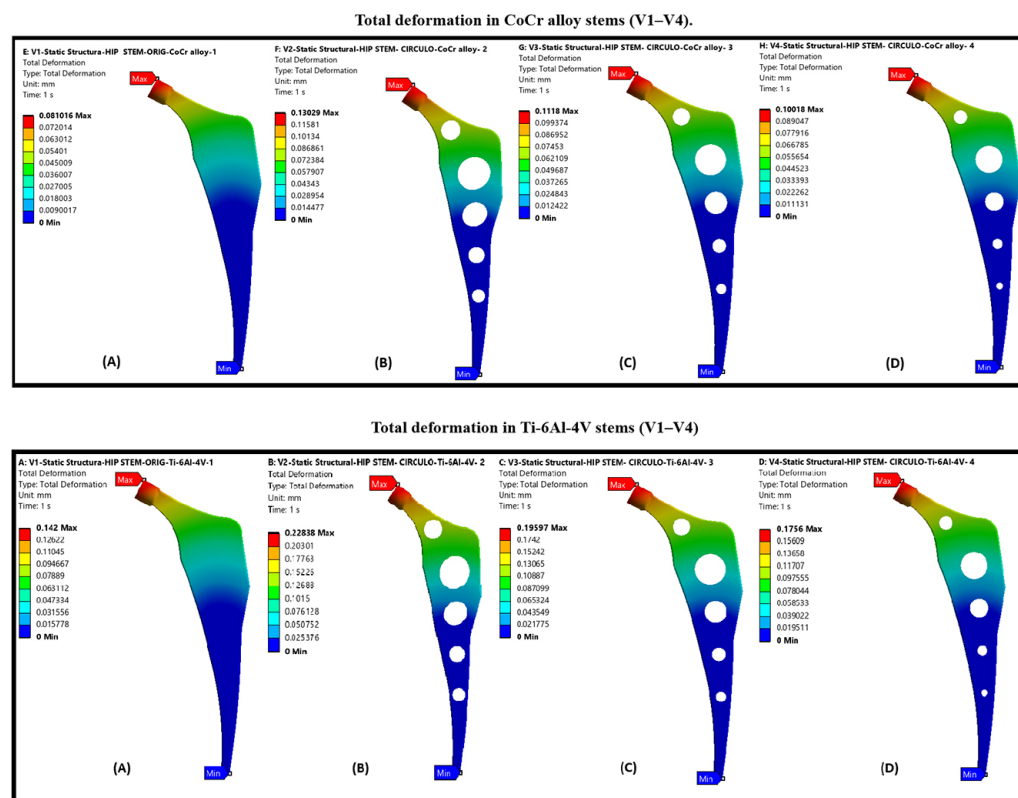


Fig. 6. Total deformation in V1–V4 configurations of stems manufactured from CoCr and Ti-6Al-4V alloys

Figure 7 shows the distribution of equivalent von Mises stress in the stems made of CoCr alloy (top) and Ti-6Al-4V (bottom). In the models without perforations (V1), the maximum stress was 372.28 MPa for CoCr and 368.81 MPa for Ti-6Al-4V. In the configurations with perforations (V2–V4), an increase in stress was observed, reaching a maximum of 533.35 MPa in CoCr and 528.44 MPa in

Ti-6Al-4V, with concentrations mainly in areas near the central holes. Despite the increase, all values remained below the yield strength of the respective alloys, ensuring structural behavior within the elastic regime and no risk of failure due to plastic deformation.

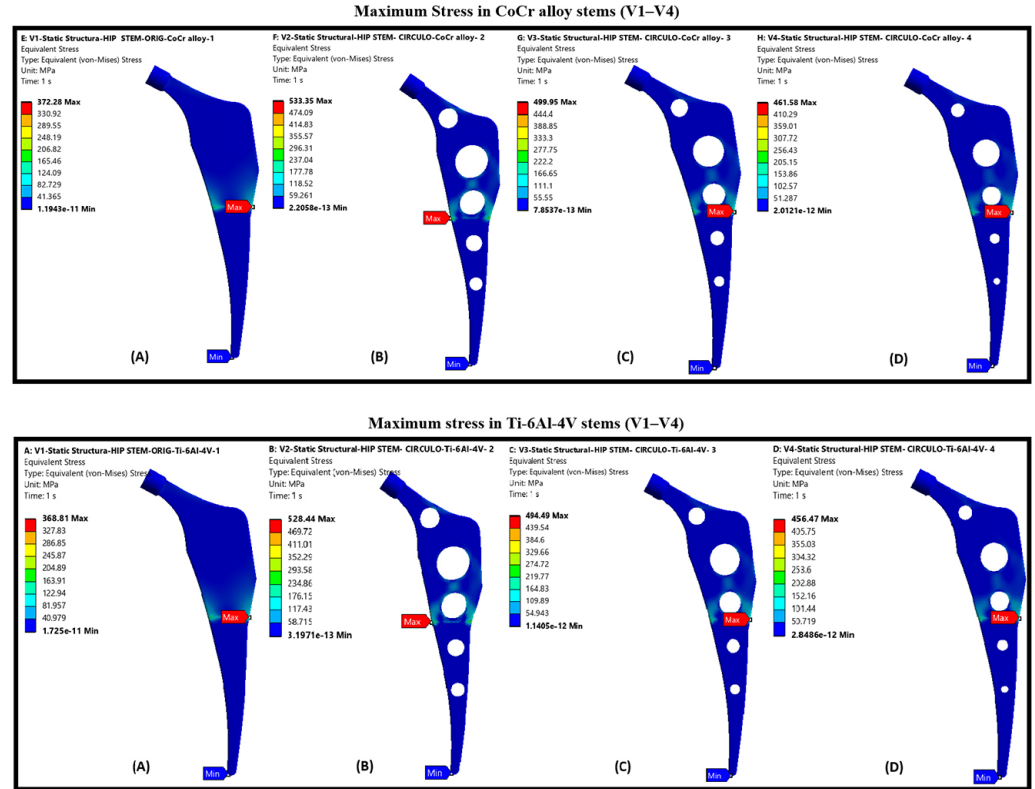


Fig. 7. Distribution of equivalent (von Mises) stress in V1–V4 configurations of stems manufactured from CoCr and Ti-6Al-4V alloys

3.2 Fatigue analysis

In the fatigue analysis of the CoCr alloy stems, the configuration without perforations (V1) showed the highest safety factors, with a minimum value of 1.55, demonstrating excellent structural resistance, as shown in Figure 8. When perforations were introduced (V2–V4), the factors decreased, especially in V2, where critical values below 0.83 were recorded at the edges of the holes. Configurations V3 and V4 offered progressive improvement, with a minimum factor of 1.10 in V3 and a value of 1.25 in V4, indicating that the latter represents an appropriate balance between structural lightening and safety under load.

In the fatigue analysis of Ti-6Al-4V stems (see Figure 8), the configuration without perforations (V1) showed the highest safety factors, with a minimum value of 2.01, indicating high structural resistance under cyclic loads. When perforations were introduced (V2–V4), a reduction in the safety factor was observed, with V2 being the most critical, reaching a minimum value of 1.40. Configurations V3 and V4 showed improvements compared to V2, achieving minimum factors of 1.62 and 1.72, respectively, which indicates that, despite the reduction in material, these optimized versions maintain adequate safety margins for demanding clinical applications.

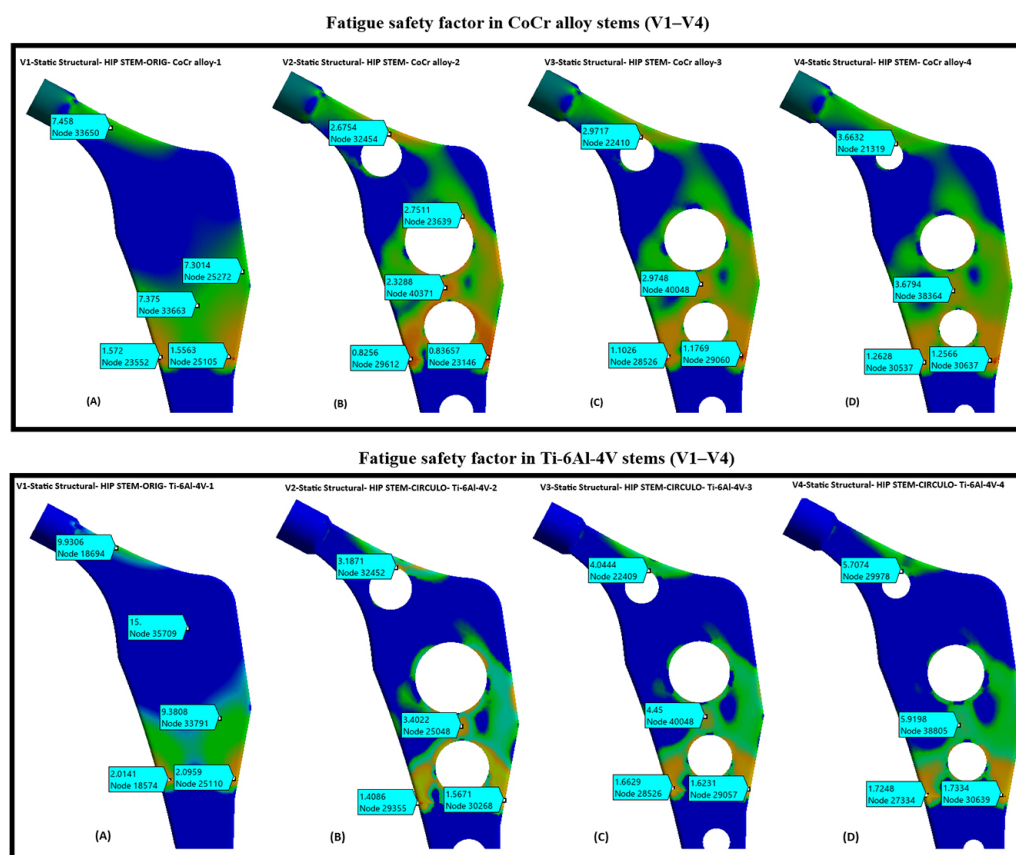


Fig. 8. Fatigue safety factors in femoral stems (V1–V4) manufactured from CoCr alloy and Ti-6Al-4V

3.3 Discussions

The results obtained in this study show that increasing the number and size of perforations leads to a significant rise in von Mises stresses and deformations, especially in the V2 configuration of Ti-6Al-4V, which reached a maximum stress of 528.44 MPa and a deformation of 0.228 mm, with a mass reduction of 19.7%. These data are clearly shown in Figure 9A for von Mises stress and in Figure 9B for total deformation. Despite these geometric modifications, the V2 configuration remained within the elastic regime and showed acceptable safety factors, as demonstrated in Figure 9C for static load and in Figure 9D for fatigue.

This behavior is consistent with the findings reported by [5], who, through static structural analyses in Ansys under load, observed that the Ti-6Al-4V material exhibited lower stresses and deformations compared to CoCr alloys, which they attributed to its lower elastic modulus and a better distribution of maximum stresses.

Additionally, [8] found maximum stresses of 744.91 MPa and a deformation of 0.37 mm for a homogeneous Ti-6Al-4V model under similar loading conditions. In comparison, the V2 configuration in this study exhibited a significantly lower stress (528.44 MPa) and less deformation (0.228 mm), indicating stiffer behavior and better stress distribution, even with an optimized geometry featuring perforations, as shown in Figures 9A and 9B. This confirms that the proposed geometry maintains structural integrity within safety limits.

Regarding fatigue behavior, the results for the V2 configuration also align with the findings of [2], who, through finite element simulation under cyclic walking loads, demonstrated that Ti-6Al-4V exhibits a longer fatigue life and better safety

factors compared to CoCr under dynamic conditions, consistent with what is shown in Figure 9D.

Although the V4 configuration showed better control of stress concentrations and less overall deformation, the titanium V2 configuration offered the best balance between weight reduction and mechanical strength under both loading conditions (static and fatigue), as summarized in Figure 9E for mass reduction and 9F for the total mass of the stem. These results, compared with those obtained by [2], [5], and [8] reaffirm the suitability of Ti-6Al-4V as the preferred material in the design of geometrically optimized femoral stems.

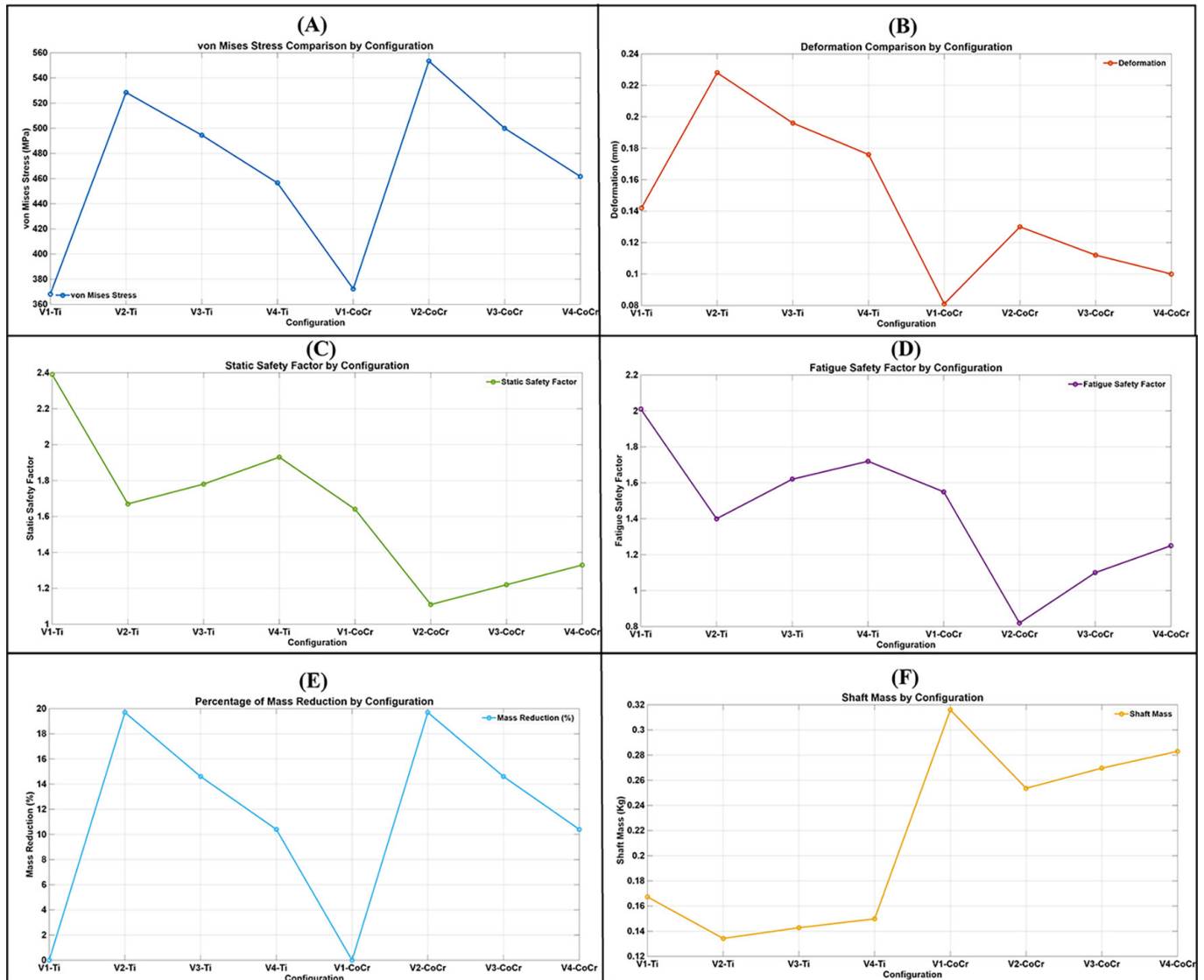


Fig. 9. Comparison of mechanical parameters in femoral stems according to configuration and material: (A) Maximum von Mises stress, (B) Total deformation, (C) Static safety factor, (D) Fatigue safety factor, (E) Percentage of mass reduction, and (F) Stem mass

4 CONCLUSIONS

The present study has demonstrated that the introduction of circular perforations in the design of femoral stems represents a viable geometric strategy to reduce the

mass of the implant without compromising its structural integrity. Through finite element simulations in ANSYS, the static and fatigue behavior of different geometric configurations was evaluated using two materials widely used in orthopedics, Ti-6Al-4V alloy and CoCr alloy.

The results obtained show that, although the incorporation of circular perforations leads to an increase in both maximum stress and deformations due to the reduction in structural stiffness, all the analyzed configurations remained within the elastic range, ensuring safe mechanical behavior. In particular, configurations V2 and V4 were identified as the most suitable. The titanium (Ti-6Al-4V) V2 configuration demonstrated an optimal balance between mass reduction and structural safety, achieving acceptable safety factors under both static conditions (1.67) and fatigue analysis (1.40), together with a notable mass reduction of 19.7%. On the other hand, the V4 CoCr alloy configuration offered an intermediate alternative, maintaining safety factors above the minimum recommended threshold of 1.33 for the static range and achieving a mass reduction of 10.4%, representing an appropriate balance between lightness and strength. These geometrical improvements aimed at weight reduction without compromising structural integrity strengthen the feasibility of the optimized design of femoral stems, contributing to the development of lighter, more sustainable, and biomechanically efficient prostheses. This work aligns with the sustainable development goals (SDG 3) by contributing to the development of more accessible, durable, and functional medical devices.

Future research could address the thermal analysis of the implant in extreme climates, evaluating heat dissipation based on the material and the modified geometry. Likewise, it is recommended to develop advanced topological optimization studies that integrate clinical and manufacturing constraints, with the goal of generating personalized designs. Another promising line consists of simulating the behavior of the stem under dynamic multiaxial loading conditions or in impact scenarios, which would make it possible to broaden the understanding of biomechanical performance in more demanding real-life situations.

5 REFERENCES

- [1] H. Long *et al.*, "Prevalence trends of site-specific osteoarthritis from 1990 to 2019: Findings from the global burden of disease study 2019," *Arthritis and Rheumatology*, vol. 74, no. 7, pp. 1172–1183, 2022. <https://doi.org/10.1002/art.42089>
- [2] J. Reginald, M. Kalayarsan, K. N. Chethan, and P. Dhanabal, "Static, dynamic, and fatigue life investigation of a hip prosthesis for walking gait using finite element analysis," *International Journal of Modelling and Simulation*, vol. 43, no. 5, pp. 797–811, 2023. <https://doi.org/10.1080/02286203.2023.2212346>
- [3] K. Shimasaki *et al.*, "Optimizing stem length in conversion total hip arthroplasty: An expanded finite element analysis," *J Clin. Med.*, vol. 14, no. 4, p. 1141, 2025. <https://doi.org/10.3390/jcm14041141>
- [4] M. Ceddia, B. Trentadue, G. De Giosa, and G. Solarino, "Topology optimization of a femoral stem in titanium and carbon to reduce stress shielding with the FEM method," *Journal of Composites Science*, vol. 7, no. 7, p. 298, 2023. <https://doi.org/10.3390/jcs7070298>
- [5] A. K. Bhawe *et al.*, "Static structural analysis of the effect of change in femoral head sizes used in total hip arthroplasty using finite element method," *Cogent. Eng.*, vol. 9, no. 1, p. 2027080, 2022. <https://doi.org/10.1080/23311916.2022.2027080>

- [6] S. G. Yan *et al.*, “Biomechanical analysis of a short femoral stem used in revision total hip replacement of a standard femoral stem,” *Sci. Rep.*, vol. 15, no. 1, p. 1967, 2025. <https://doi.org/10.1038/s41598-025-86108-6>
- [7] D. M. Anticono-Valderrama, J. L. Serna-Landivar, W. C. Algoner, M. L. Miranda, M. Y. Garcia-Alvarez, and L. K. S. Landivar, “Static, transient, and fatigue design and analysis of a hip femoral stem using the finite element method,” *International Journal of Online and Biomedical Engineering*, vol. 20, no. 16, pp. 89–102, 2024. <https://doi.org/10.3991/ijoe.v20i16.52865>
- [8] V. K. Mittal and V. Gupta, “Homogeneous and heterogeneous modeling of patient-specific hip implant under static and dynamic loading condition using finite element analysis,” *Journal of The Institution of Engineers (India): Series D*, vol. 105, no. 1, pp. 1–20, 2024. <https://doi.org/10.1007/s40033-023-00447-0>
- [9] D. Scherb *et al.*, “Optimizing the individual design of revision hip implants – evaluating the effects for bone and stem using patient-specific finite element models,” in *Proceedings of the 35th Symposium Design for X, DFX 2024*, 2024. <https://doi.org/10.35199/dfx2024.02>
- [10] Z. Xiao, L. Wu, W. Wu, R. Tang, J. Dai, and D. Zhu, “Multi-scale topology optimization of femoral stem structure subject to stress shielding reduce,” *Materials*, vol. 16, no. 8, p. 3151, 2023. <https://doi.org/10.3390/ma16083151>
- [11] X. Wang *et al.*, “Topological design and additive manufacturing of porous metals for bone scaffolds and orthopaedic implants: A review,” *Biomaterials*, vol. 83, pp. 127–141, 2016. <https://doi.org/10.1016/j.biomaterials.2016.01.012>
- [12] M. Fraldi, L. Esposito, G. Perrella, A. Cutolo, and S. C. Cowin, “Topological optimization in hip prosthesis design,” *Biomech. Model Mechanobiol.*, vol. 9, no. 4, pp. 389–402, 2010. <https://doi.org/10.1007/s10237-009-0183-0>
- [13] D. Zhao *et al.*, “Design, fabrication and clinical characterization of additively manufactured tantalum hip joint prosthesis,” *Regen. Biomater.*, vol. 11, 2024. <https://doi.org/10.1093/rb/rbae057>
- [14] V. Tuninetti *et al.*, “Computational shape design optimization of femoral implants: Towards efficient forging manufacturing,” *Applied Sciences (Switzerland)*, vol. 14, no. 18, p. 8289, 2024. <https://doi.org/10.3390/app14188289>
- [15] M. Ceddia and B. Trentadue, “Evaluation of rotational stability and stress shielding of a stem optimized for hip replacements—A finite element study,” *Prosthesis*, vol. 5, no. 3, pp. 678–693, 2023. <https://doi.org/10.3390/prosthesis5030048>
- [16] R. Marar, N. Alhyari, and S. Tedmori, “Improving 3D animation education: A case study of curriculum development in Jordan,” *International Journal of Engineering Pedagogy (ijEP)*, vol. 15, no. 5, pp. 89–107, 2025. <https://doi.org/10.3991/ijep.v15i5.53579>
- [17] M. L. Brandi and L. Cavalli, “Periprosthetic bone loss: Diagnostic and therapeutic approaches,” *F1000Res*, vol. 2, p. 266, 2014. <https://doi.org/10.12688/f1000research.2-266.v2>
- [18] R. V. V. Petrescu, R. Aversa, V. Perrotta, L. M. Ungureanu, A. Apicella, and F. I. T. Petrescu, “News in bone modeling for customized hybrid biological prostheses development,” *Online J. Biol. Sci.*, vol. 21, no. 2, pp. 285–316, 2021. <https://doi.org/10.3844/ojbsci.2021.285.316>
- [19] G. Mani, D. Porter, S. Collins, T. Schatz, A. Ornberg, and R. Shulfer, “A review on manufacturing processes of cobalt-chromium alloy implants and its impact on corrosion resistance and biocompatibility,” *J. Biomed. Mater. Res. B Appl. Biomater.*, vol. 112, no. 6, p. e35431, 2024. <https://doi.org/10.1002/jbm.b.35431>
- [20] E. Marin and A. Lanzutti, “Biomedical applications of titanium alloys: A comprehensive review,” *Materials*, vol. 17, no. 1, p. 114, 2024. <https://doi.org/10.3390/ma17010114>

- [21] A. Z. Senalp, O. Kayabasi, and H. Kurtaran, "Static, dynamic and fatigue behavior of newly designed stem shapes for hip prosthesis using finite element analysis," *Mater. Des.*, vol. 28, no. 5, pp. 1577–1583, 2007. <https://doi.org/10.1016/j.matdes.2006.02.015>
- [22] J. V. Corda, K. N. Chethan, B. Satish Shenoy, S. Shetty, N. Shyamasunder Bhat, and M. Zuber, "Fatigue life evaluation of different hip implant designs using finite element analysis," *Journal of Applied Engineering Science*, vol. 21, no. 3, pp. 896–907, 2023. <https://doi.org/10.5937/jaes0-44094>
- [23] A. D. Oza, N. Gupta, and R. Singh, "Design and non-linear finite element analysis of titanium-based femoral hip-stem for Indian population," *International Journal on Interactive Design and Manufacturing*, vol. 17, no. 5, pp. 2489–2493, 2023. <https://doi.org/10.1007/s12008-022-01047-0>
- [24] L. Xie *et al.*, "Numerical analysis and experimental validation on residual stress distribution of titanium matrix composite after shot peening treatment," *Mechanics of Materials*, vol. 99, pp. 2–8, 2016. <https://doi.org/10.1016/j.mechmat.2016.05.005>
- [25] S. Kapoor *et al.*, "Evaluation of stress generated with different abutment materials and angulations under axial and oblique loading in the Anterior Maxilla: Three-dimensional finite element analysis," *Int. J. Dent.*, vol. 2021, 2021. <https://doi.org/10.1155/2021/9205930>
- [26] O. K. Ajayi, B. O. Malomo, S. D. Paul, A. A. Adeleye, and S. A. Babalola, "Failure modeling for titanium alloy used in special purpose connecting rods," *Materials Today: Proceedings*, vol. 45, no. Part 6, pp. 4390–4397, 2021. <https://doi.org/10.1016/j.matpr.2020.11.852>
- [27] A. Mestar, S. Zahaf, N. Zina, and A. Boutaous, "Numerical study of the effect of elastomer and cement of stress absorbers on the reduction of stresses in tibia and tibial bone analysed by finite element method," *Nano Biomed. Eng.*, vol. 10, no. 1, pp. 56–78, 2018. <https://doi.org/10.5101/nbe.v10i1.p56-78>
- [28] M. G. Gok and O. Cihan, "Numerical analysis of the application of different lattice designs and materials for reciprocating engine connecting rods," *Scientia Iranica*, vol. 29, no. 5 B, pp. 2362–2373, 2022. <https://doi.org/10.24200/sci.2022.59400.6216>
- [29] ASTM, "F 2996-20 Standard practice for Finite Element Analysis (FEA) of non-modular metallic orthopaedic hip femoral stems," ASTM International, West Conshohocken, PA, pp. 1–11, 2020.
- [30] ISO, "ISO 7206-4:2010 – Implants for surgery—Partial and total hip joint prostheses—Part 4: Determination of endurance properties and performance of stemmed femoral components n.d.," 2010.
- [31] S. Karimi, F. Haji Aboutalebi, and M. Heidari-Rarani, "A numerical study on fatigue design of Ti-6Al-4V total hip stem: Infinite-life and damage tolerance approaches using XFEMPN-VCCT," *Meccanica*, vol. 58, no. 5, pp. 959–980, 2023. <https://doi.org/10.1007/s11012-023-01661-6>
- [32] J. Triyono, A. R. Prabowo, and J. M. Sohn, "Investigation of meshing strategy on mechanical behaviour of hip stem implant design using FEA," *Open Engineering*, vol. 10, no. 1, pp. 769–775, 2020. <https://doi.org/10.1515/eng-2020-0087>
- [33] G. Zhang *et al.*, "Parametric analysis of the effect of impaction load on the stability of head-neck junction in total hip arthroplasty," *Clinical Biomechanics*, vol. 94, p. 105633, 2022. <https://doi.org/10.1016/j.clinbiomech.2022.105633>
- [34] P. S. R. S. Maharaj, A. Vasanthanathan, F. B. D. Ebenezer, R. Giriharan, and M. Athithayan, "In situ bio printing of carbon fiber reinforced PEEK hip implant stem," *AIP Conference Proceedings*, vol. 2653, p. 030008, 2022. <https://doi.org/10.1063/5.0110578>
- [35] K. N. Chethan, Z. Mohammad, N. Shyamasunder Bhat, and B. Satish Shenoy, "Optimized trapezoidal-shaped hip implant for total hip arthroplasty using finite element analysis," *Cogent Eng.*, vol. 7, no. 1, p. 1719575, 2020. <https://doi.org/10.1080/23311916.2020.1719575>

- [36] J. L. Serna-Landivar *et al.*, “Static, dynamic, and high cycle fatigue analysis of crossed spherical gearing for robotic arm ball joint: A finite element analysis approach,” *International Journal of Online and Biomedical Engineering*, vol. 20, no. 2, pp. 16–30, 2024. <https://doi.org/10.3991/ijoe.v20i02.46817>
- [37] K. Nabudda *et al.*, “Identification of flexural modulus and poisson’s ratio of fresh femoral bone based on a finite element model,” *International Journal of Online and Biomedical Engineering (iJOE)*, vol. 18, no. 4, pp. 94–105, 2022. <https://doi.org/10.3991/ijoe.v18i04.28939>

6 AUTHORS

José L. Serna-Landivar completed Msc. in Mechanical Engineering at the Pontificia Universidad Católica del Perú (PUCP) and Eng. in Mechanical Engineering from Universidad Tecnológica de Perú. He has experience in computer-aided design in Inventor, SolidWorks, and simulation with ANSYS and SAP2000 computational tools. His areas of interest in research includes applied mechanics, mechanical design, and finite element method for numerical solution of structural problems (E-mail: u18101283@utp.edu.pe).

Daniela M. Anticona-Valderrama has a degree in physical mathematics, master’s degree in education administration, PhD in public management and governance at the Universidad Cesar Vallejo, and is a PhD candidate in didactics of scientific research at the National University of Panama; with experience of 6 years as a professor at Universidad Tecnológica del Perú, Universidad Autónoma del Perú, Universidad César Vallejo and Universidad Privada del Norte and interest in research that includes education, environmental management, mathematics, engineering, and technology (E-mail: daniela.anticona@upn.pe).

William C. Algoner is a physics researcher, engineering educator, and scientific publication coordinator at the Universidad Tecnológica del Perú (UTP). He was a postdoctoral researcher at the Federal University of Parana, Brazil. He is currently an IEEE-UTP Student Branch Advisor (E-mail: walgoner@utp.edu.pe).

Andrés Eduardo De La Cruz Herrera holds a master’s degree in public management from Universidad César Vallejo (UCV), Civil Engineer from the National University of San Luis Gonzaga in Ica. Professor of Civil Engineering, Bridge and Works of Art Design, Structural Analysis using Finite Methods, Shallow and Deep Foundations (E-mail: adelacruz01@ucvvirtual.edu.pe).

Marleni Mendoza Damas holds a Master’s degree in accreditation and evaluation of educational quality from Universidad San Ignacio de Loyola (USIL) and serves as a mathematics professor at the National University of San Marcos (UNMSM). She has extensive experience in higher education teaching, particularly in the scientific disciplines. Her study interests focus on education, design engineering, technology, and health sciences (E-mail: c18968@utp.edu.pe).

Jeferson Rayu Vega Zavala is a business administrator and university professor at the Universidad Tecnológica del Perú (UTP). He holds a master’s degree in business administration (MBA) and specializes in research (E-mail: C28481@utp.edu.pe).