

## PAPER

# Static, Transient, and Fatigue Design and Analysis of a Hip Femoral Stem Using the Finite Element Method

Daniela Milagros Anticona-Valderrama<sup>1</sup>(✉), José L Serna-Landivar<sup>2</sup>, William C. Algoner<sup>2</sup>, Manuel López Miranda<sup>3</sup>, Maria Ysabel Garcia-Alvarez<sup>2</sup>, Leslie Katerine Serna Landivar<sup>4</sup>

<sup>1</sup>Universidad Privada del Norte, Lima, Perú

<sup>2</sup>Universidad Tecnológica del Perú, Lima, Perú

<sup>3</sup>Universidad Ricardo Palma, Lima, Perú

<sup>4</sup>Universidad César Vallejo, Lima, Perú

[daniela.anticona@upn.pe](mailto:daniela.anticona@upn.pe)

## ABSTRACT

The hip femoral stem is vital in ensuring patient support and mobility, and research into the femoral stem is critical for improving the durability and strength of orthopedic prostheses. Using the finite element method, this paper evaluates a femoral stem's static, transient, and fatigue behaviors using three different materials: Ti-6Al-4V, CoCr alloy, and AISI 316L. ANSYS Mechanical software was used to perform the static analysis, evaluate deformations and stresses, and perform the transient analysis to simulate dynamic loading conditions. In addition, a fatigue analysis was conducted to determine the resistance to repetitive load cycles. The results showed that the stresses and strains in the transient analysis were higher than in the static analysis because the transient analysis considers dynamic and time-varying loads. In addition, the Ti-6Al-4V material exhibits higher fatigue resistance, a significantly longer service life, and lower stresses in the static and transient range. These findings reaffirm the importance of selecting materials that balance stiffness, structural strength, and fatigue to optimize the performance and durability of femoral implants.

## KEYWORDS

hip implant, finite element method (FEM), finite element analysis (FEA), prostheses

## 1 INTRODUCTION

The hip is the most essential part of the human body. It supports the body's weight, provides stability, and allows various daily activities, such as walking, standing, sitting, running, climbing, and other movements [1], [2].

Due to various factors, people suffer from degenerative diseases affecting the hip joints, such as osteoarthritis, rheumatoid and infectious arthritis, lupus, and avascular necrosis. Worldwide, osteoarthritis affects more than 527 million people, with significant growth in recent decades [3], which is why total hip replacement (THA) is an essential surgical procedure to relieve pain and restore mobility in people [4], [5], [6]. More than 635,000 THA surgeries are performed in the USA yearly [7].

Anticona-Valderrama, D.M., Serna-Landivar, J.L., Algoner, W.C., Miranda, M.L., Garcia-Alvarez, M.Y., Landivar, L.K.S. (2024). Static, Transient, and Fatigue Design and Analysis of a Hip Femoral Stem Using the Finite Element Method. *International Journal of Online and Biomedical Engineering (iJOE)*, 20(16), pp. 89–102. <https://doi.org/10.3991/ijoe.v20i16.52865>

Article submitted 2024-07-17. Revision uploaded 2024-09-24. Final acceptance 2024-10-05.

© 2024 by the authors of this article. Published under CC-BY.

Hip implant designs and materials have evolved significantly. However, the durability and reliability of these devices remain critical issues in orthopedic practice and biomedical research. Hip implants use biocompatible materials, such as metal alloys and polymers, to improve wear and fatigue resistance [8].

The finite element method (FEM) is a computational tool for numerically simulating the behavior of mechanisms, such as hip stem implants [9], [10]. This technique allows the simulation and evaluation of the mechanical behavior of implants under various static and dynamic loading conditions, providing crucial data for design optimization. Previous research has demonstrated the importance of the femoral stem geometry and implant-bone interaction in the stress distribution, deformation, and implant lifetime [11], [12]. It also allows the simulation of hip joint wear [13]–[18].

The existing literature evaluating the behavior of femoral stem implants primarily addresses three areas: static analysis, dynamic analysis, and fatigue analysis. Static analysis focuses on implant responses under constant body loads [19]–[32], whereas dynamic analysis considers variable loads during activities such as walking [8], [11], [17], [20], [21], [26], [33]–[36]. Fatigue analysis, on the other hand, evaluates the resistance of the implant to failure due to repeated cyclic stresses [8], [11], [14], [15], [34], [37], [38]. However, these investigations did not consider the transient analysis that is part of dynamic analysis, which allows a more accurate evaluation of the stresses and deformations due to the impacts of loads on the joint between the hip and the stem implant; these results cannot be found in static analysis. There is a research gap to be filled by this study.

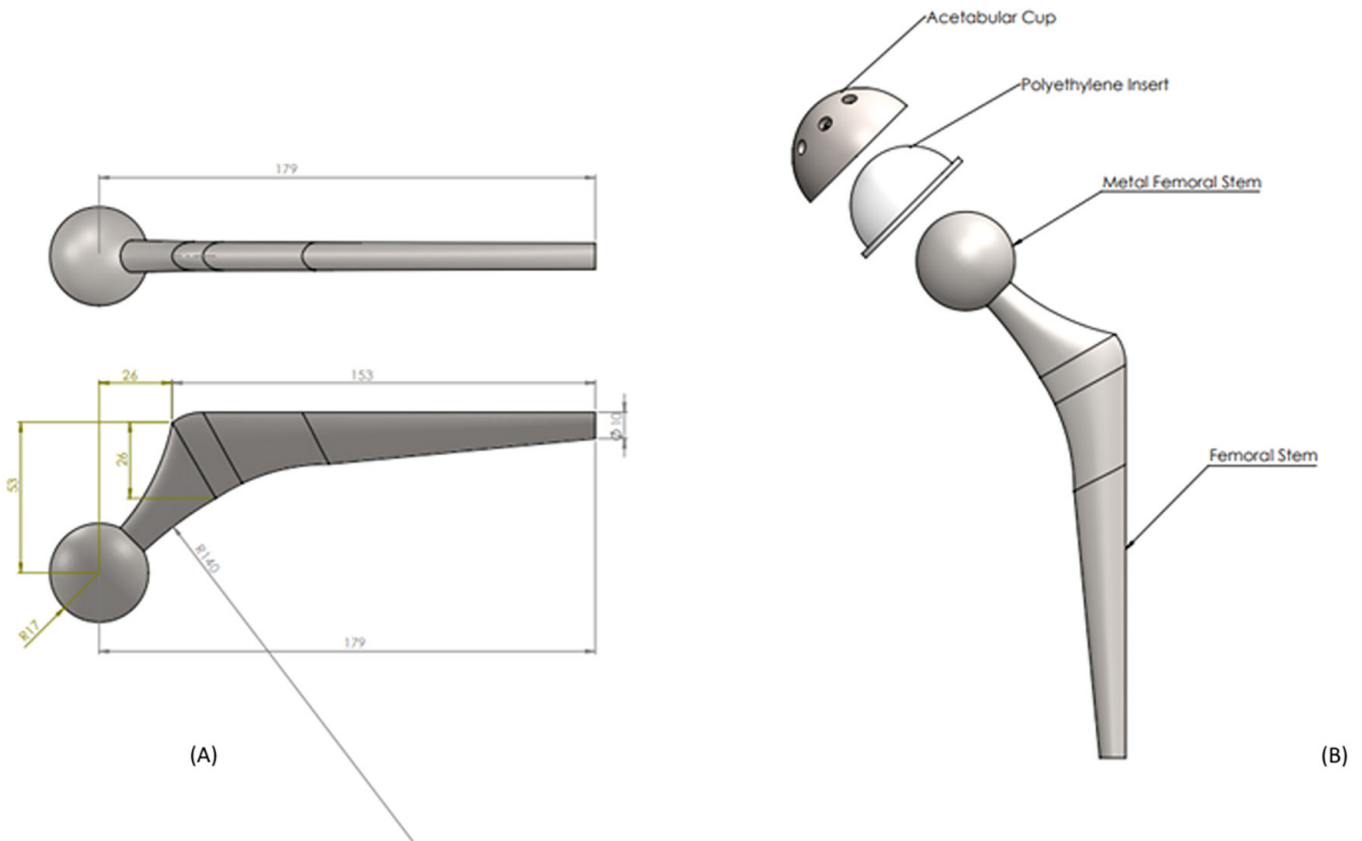
This study is crucial because it can improve the safety and durability of hip prostheses, directly affecting patients' quality of life. The expected results will enhance the design and selection of materials for implant manufacturers to improve their products. In addition, these findings will contribute significantly to the scientific knowledge of biomechanics, providing a solid basis for future research, regulations, and design standards and reducing the need for surgical revisions, thus improving people's quality of life.

In this study, the mechanical strength of the hip femoral stem was evaluated in terms of stress, strain, and safety factors via static, dynamic (modal, transient), and high cycle fatigue analyses using finite element analysis (FEA) (ANSYS). The materials used for the femoral stem were Ti6Al4V titanium and cobalt-chromium alloys. The static, fatigue, and transient analysis results were compared for the two materials. In addition, the dynamic behavior was evaluated against the loads under natural conditions to which the femoral stem is subjected.

## 2 MATERIALS AND METHODS

### 2.1 Hip implant stem model

This study focused on the femoral stem, an important part of the system that should be evaluated in each patient (see Figure 1). The dimensions of the femoral stem are shown in Figure 1a. The hip implant system comprises an acetabular cup, polyethylene inserts, a metal femoral stem, and a femoral stem (see Figure 1b). The modeling was performed using CAD SolidWorks software, and FEA was performed.



**Fig. 1.** (a) Overall dimensions of the “femoral stem”; (b) Components of the hip implant system

## 2.2 Materials

The femoral stem should be composed of biomaterials, and our study used stainless steel (AISI 316L), titanium alloy (Ti-6Al-4V), and cobalt-chromium alloy (CoCr alloy). Table 1 lists the mechanical properties of the materials, which are essential for conducting an accurate simulation.

**Table 1.** Mechanical properties of the Ti-6Al-4V, CoCr alloy, and AISI-316L materials

Material	Modulus of Elasticity (Gpa)	Yield Strength (Mpa)	Poisson Ratio	Ultimate Tensile Strength (MPa)	Density (Kg/m <sup>3</sup> )	References
Ti-6Al-4 V	114	880	0.31	930	4500	[8], [38]–[45]
CoCr alloy	200	612	0.3	1503	8500	[8], [34], [38]
AISI 316	210	266	0.27	647	7969	[46]

Studies [8], [34] also found references to Ti-6Al-4V and CoCr alloy materials regarding the fatigue strength and stress cycle number (S-N) curves of the three materials mentioned; a reference was also found for AISI 3016L stainless steel [47]; see Figure 2.

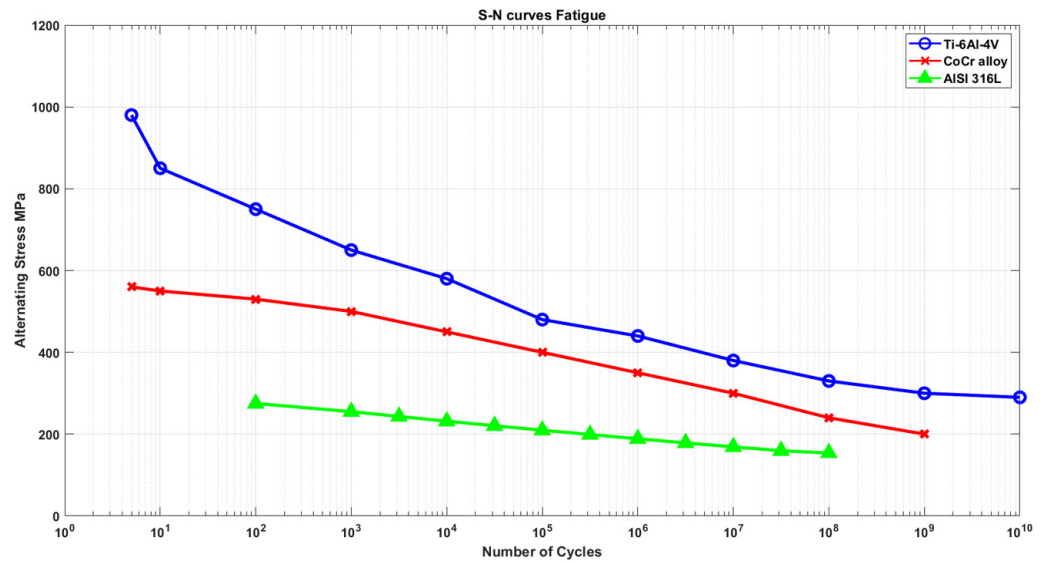


Fig. 2. Fatigue curves S-N de Ti-6Al-4V, CoCr alloy, and AISI 316L

### 2.3 Boundary conditions and meshing

Some standards standardize the appropriate parameters for stem-edge conditions to obtain reliable results (see Figure 3a). For example, ASTM F2996-20 [48] addresses boundary conditions, and ISO 7206-4:2010 addresses loads [49]; (see Figure 3b) To attach the stem, a line is drawn from the top center of the sphere down to 80 mm; an additional 10 mm line is drawn. Finally, from this point, a distance of 97.8 mm was subtracted from the lower tip to determine the fixed part. A load of 2300 N was applied to the cup perpendicular to the femoral head (see Figure 4). Previous investigations [27], [37], [50]–[52] have followed this methodology.

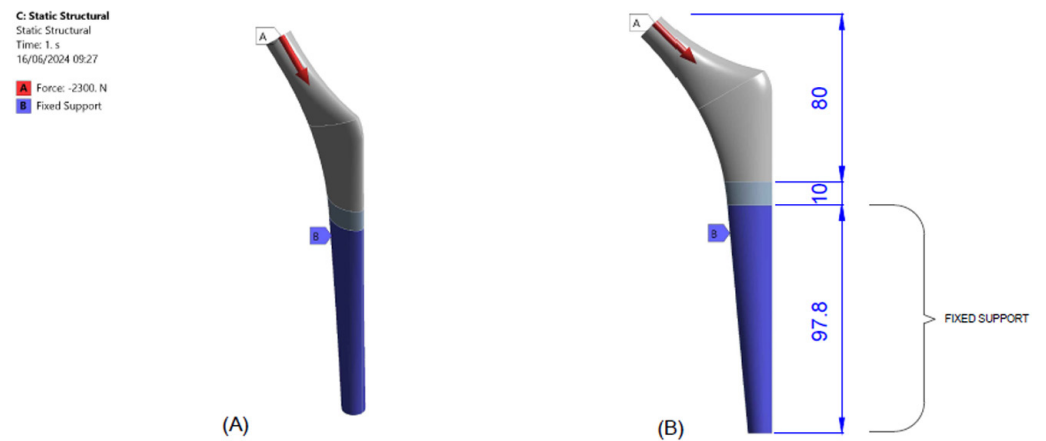


Fig. 3. (a) Stem boundary conditions for finite element analysis; (b) Dimensions and edge conditions according to ASTM F2996-13 and applied force according to ISO 7206-4

During the research object’s meshing process, a mesh convergence analysis was performed to ensure proper discretization of the stem and mesh refinement, thus avoiding significant deviations in the results. As a result, 15900 nodes and 4297 elements were obtained. The average quality of the elements was 0.93, therefore

complying with the recommendation of the Reference [53] for a minimum mesh quality of 0.7. This indicates that the component is of good quality. Figure 4a shows the stem with the refined mesh, and Figure 4b shows the mesh convergence diagram relating the stress to the number of nodes.

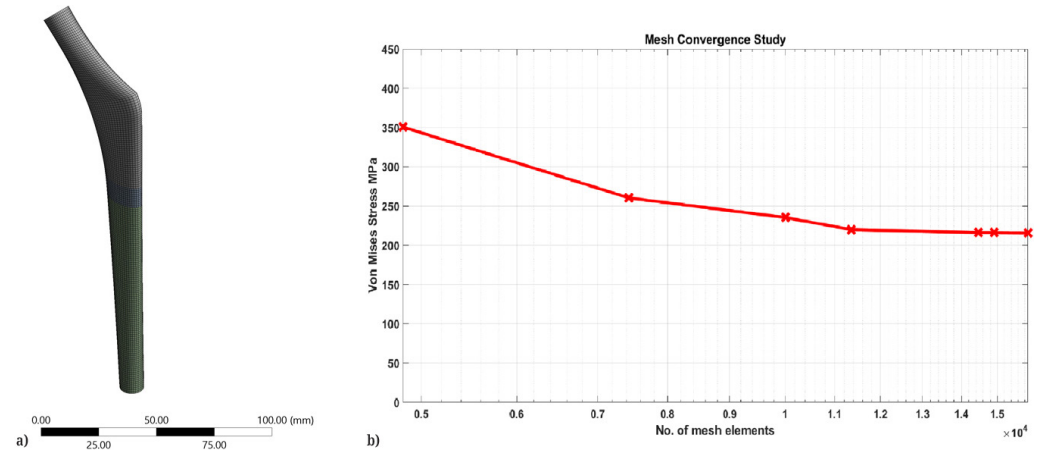


Fig. 4. (a) Mesh shank; (b) Mesh convergence in the finite element analysis

## 2.4 Static analysis

The static simulation performed with ANSYS determines the stresses, strains, and safety factors by linear analysis within the elastic range based on the relevant equation (Eq. 1) [54], [55]. This analysis is fundamental for evaluating the strength and stiffness of components under load; it provides an understanding of how forces affect the target object.

$$\{F\} = [K]\{u\} \quad (1)$$

Here,  $\{F\}$  is the force vector,  $[K]$  is the stiffness matrix, and  $\{u\}$  is the displacement vector.

## 2.5 Transient analysis

In the transient analysis of the femoral stem for hip prosthesis, a dynamic simulation was performed to evaluate the implant's behavior under time-varying loads. This analysis is governed by the differential equation of motion, which incorporates the mass, stiffness, damping matrices, and time-dependent forces (Eq. 2).

$$F_{(t)} = [M\ddot{u}] + [C\dot{u}] + [Ku] \quad (2)$$

Where,  $F_{(t)}$  is the time-varying force vector,  $[M]$  is the mass matrix,  $[C]$  is the damping matrix,  $[K]$  is the stiffness matrix and  $\ddot{u}$ ,  $\dot{u}$  and  $u$  corresponds to the acceleration, velocity, and nodal displacement vectors, respectively [56].

Transient analysis provides a better structural view of the femoral stem under dynamic conditions, ensuring that the design can withstand the impacts of dynamic loads and vibrations without compromising its integrity. This methodology is essential for designing and validating safe and reliable hip prostheses and providing critical information to improve implant durability and functionality.

## 2.6 Fatigue analysis

Fatigue analysis is essential for ensuring the durability and safety of the femoral stem in hip prostheses. Soderberg’s theory is the most conservative on fatigue analysis. It is based on the premise that the combination of alternating stress and the mean stress must not exceed the material strength. This theory uses the material’s elastic limit  $S_e$  instead of the ultimate strength  $S_u$  as in other theories, such as Goodman’s and Gerber’s. The proposed theory provides a more conservative and safe fatigue-resistance assessment method.

Soderberg’s theory was applied to evaluate fatigue life; Eq. 3 details this theory.

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

Where:

- $\sigma_a$ : Alternating stress
- $S_e$ : Fatigue limit resistance
- $\sigma_m$ : Average stress
- $S_y$ : Material yield strength

The integration of fatigue analysis with the finite element method provides a detailed and accurate evaluation of the durability of the femoral stem under cyclic loading, ensuring its reliability and longevity under natural conditions of use and minimizing the risk of failure [8].

## 3 RESULTS AND DISCUSSIONS

### 3.1 Static analysis

The simulation results in Figure 5 indicated that Ti-6Al-4V presented a maximum deformation of 0.079239 mm (see Figure 5a), compared with CoCr alloy (see Figure 5b) and AISI 316L (see Figure 5c), which showed maximum deformations of 0.045245 and 0.043285 mm, respectively. In all cases, the maximum deformation was located at the base of the shank neck because the force was located there. Formation values were within the permissible range for this type of femoral implant.

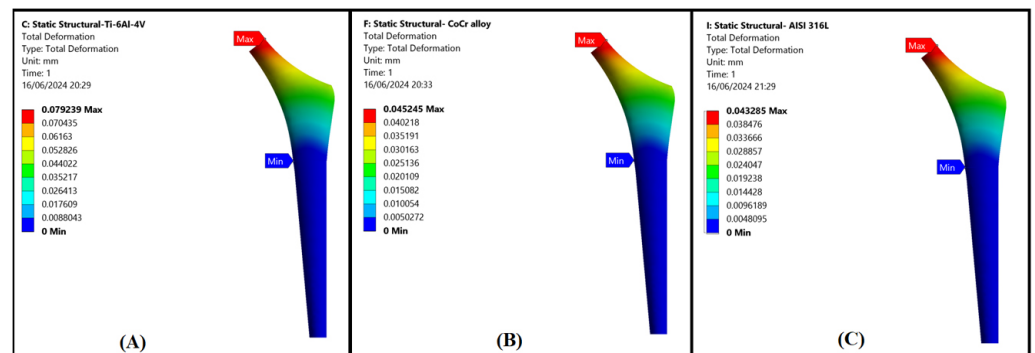
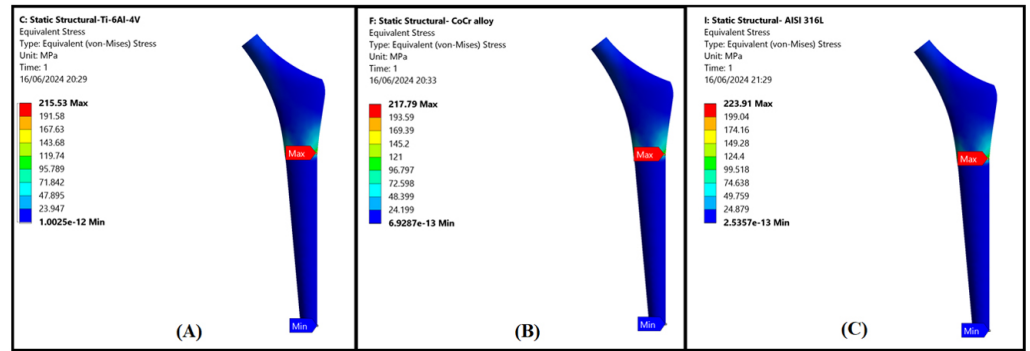


Fig. 5. Static analysis: (A) Maximum deformation Ti-6Al-4V, (B) Maximum deformation CoCr Alloy, (C) Maximum deformation of AISI 316L

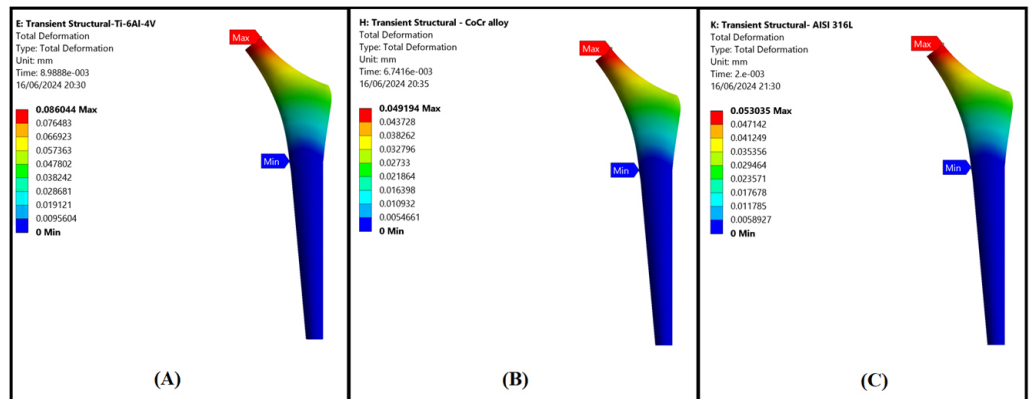


**Fig. 6.** Static analysis: (A) Maximum stresses Ti-6Al-4V, (B) Maximum stresses CoCr Alloy, and (C) Maximum stresses AISI 316L

The maximum stresses (see Figure 6) in the simulation results for the Ti-6Al-4V (see Figure 6a) material presented a maximum stress of 215.55 MPa. In contrast, the CoCr alloy (see Figure 6b) and AISI 316L (see Figure 6c) reached maximum stresses of 217.79 and 223.91 MPa, respectively. In all cases, the highest stress concentration was observed at the base of the shank neck. This stress distribution indicates that this zone is critical for the structural integrity of implants. The analysis reveals that although all three materials withstand significant static loads, AISI 316L exhibits the highest maximum stress, followed by CoCr alloy and Ti-6Al-4V. These results are fundamental for selecting the most suitable material and ensuring that the implant can withstand the applied stresses without compromising its durability and functionality under natural conditions of use.

### 3.2 Transient analysis

A transient analysis (see Figure 7) of the femoral hip stem was performed using the finite element method to evaluate the behavior of three materials: Ti-6Al-4V, CoCr alloy, and AISI 316L. The total strain results indicate that Ti-6Al-4V (see Figure 7a) recorded a maximum strain of 0.086404 mm, CoCr alloy (see Figure 7b) recorded the smallest maximum strain of 0.049134 mm, and AISI 316L (see Figure 7c) recorded a maximum strain of 0.053103 mm. These results suggest that the CoCr alloy exhibits the best structural stiffness with the least degree of deformation. This is crucial in femoral stem design because less deformation could imply better implant stability and less risk of micromovement that can affect osseointegration.



**Fig. 7.** Transient analysis: (A) Maximum deformation of Ti-6Al-4V, (B) Maximum deformation of CoCr alloy, (C) Maximum deformation of AISI 316L

For the von Mises equivalent stresses (see Figure 8), Ti-6Al-4V (see Figure 8a) reached maximum stress of 245.74 MPa, CoCr alloy (see Figure 8b) reached a maximum stress of 224.3 MPa, and AISI 316L (see Figure 8c) presented the highest maximum stress of 275.42 MPa. These results indicate that despite its higher deformation resistance, AISI 316L is more susceptible to fatigue under transient loading conditions because of its higher maximum stresses. On the other hand, the CoCr alloy exhibited the lowest strain and relatively low stress, suggesting an excellent combination of stiffness and stress resistance, making this material an attractive option for the design of femoral hip stems.

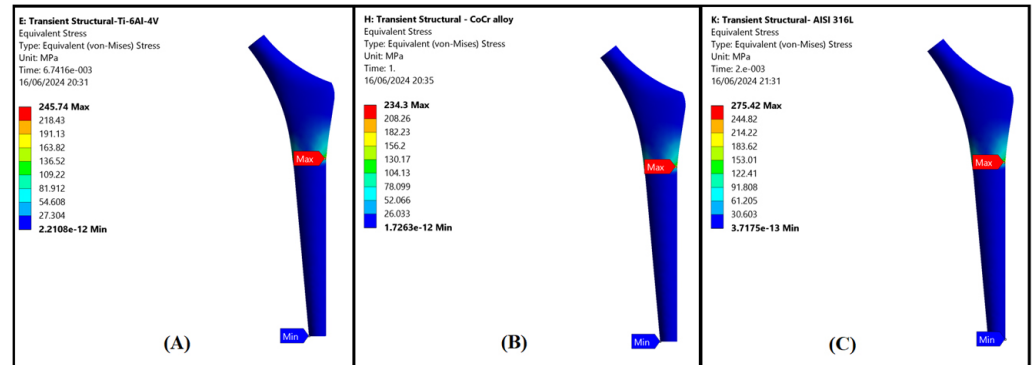


Fig. 8. Transient analysis: (A) Maximum stresses Ti-6Al-4V, (B) Maximum stresses of CoCr alloy, and (C) Maximum stresses of AISI 316L

### 3.3 Fatigue analysis

In the fatigue analysis of the femoral stem (see Figure 9), the results indicate that the Ti-6Al-4V (see Figure 9a) material has a minimum service life of  $10^{10}$  cycles, whereas the CoCr alloy (see Figure 9b) and AISI 316L (see Figure 9c) have minimum service lives of  $1.85 \cdot 10^8$  cycles, and 6658 cycles, respectively. In all cases, the minimum service life was located at the base of the shank neck, which is critical for fatigue resistance. The CoCr alloy and AISI 316L implants demonstrated lower fatigue resistance than the Ti-6Al-4V ones, indicating that the Ti-6Al-4V implant is suitable for ensuring implant durability under cyclic loading conditions by meeting a minimum cycle count of  $10^6$  cycles [37].

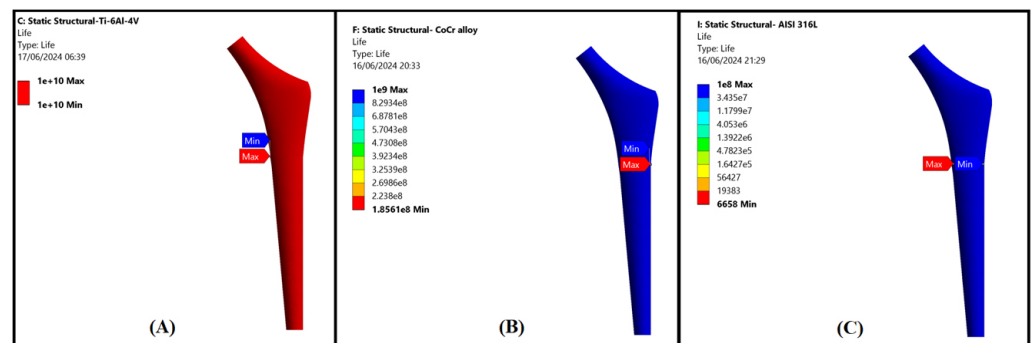
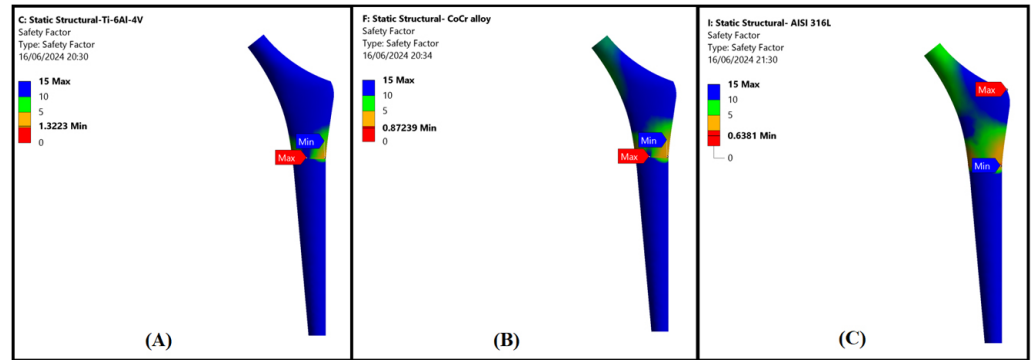


Fig. 9. Fatigue analysis: (A) Ti-6Al-4V implant life, (B) CoCr alloy implant life, (C) AISI 316L implant life



**Fig. 10.** Fatigue analysis: safety factor (A) Implant safety factor Ti-6Al-4V, (B) Implant safety factor CoCr Alloy, (C) Implant safety factor AISI 316L

In the fatigue analysis of the femoral stem (see Figure 10), the results indicate that the Ti-6Al-4V (see Figure 10a) material has a minimum safety factor of 1.3223, whereas the CoCr alloy (see Figure 10b) and AISI 316L (see Figure 10c) have minimum safety factors of 0.87229 and 0.6831, respectively. In all cases, the critical region with the lowest safety factors was the stem neck base. The distribution of the safety factors reveals that although Ti-6Al-4V has the highest safety factor, CoCr alloy and AISI 316L have lower safety factors, indicating higher susceptibility to fatigue under cyclic loading conditions. These findings are essential for material selection and ensure implants can withstand stresses without compromising their long-term structural integrity and functionality in clinical applications. In addition, they provide a solid basis for future design optimization and experimental validation.

**Limitations of the proposed approach:** The approach used in this study has yielded valuable results on the behavior of Ti-6Al-4V, CoCr alloy, and AISI 316L materials in femoral implants. However, as with any simulation-based research, it is important to recognize that the idealized loading and environmental conditions do not include all real-world variables, such as wear or the biological environment. Even so, these results provide a solid basis for future experimental investigations and validation of the models used, contributing to advancing orthopedic implant material design and selection.

### 3.4 Discussions

The results show that the stresses of the static analysis are higher in the CoCr alloy stem than in the Ti-6Al-4V material, in agreement with previous research [30], [31]. Furthermore, our results agree that the highest stress was obtained for the AISI 3016L stainless steel stem [35]; this material has the lowest safety factor for loads subjected to numerical validation. Regarding the deformations obtained, the AISI 316L stem material has the highest stiffness compared with Ti-6Al-4V and CoCr alloy; these results also agree with previous studies [30], [31], [35].

**Table 2.** Comparison of the results for Ti-6Al-4V, CoCr alloy, and AISI-316L

Parameter	Ti-6Al-4V	CoCr alloy	AISI 316L
Maximum Static Deformation (mm)	0.0792	0.0452	0.0433
Maximum Static Stress (MPa)	215.55	217.79	223.91
Maximum Transient Deformation (mm)	0.0864	0.0491	0.0531
Maximum Transient Strain (MPa)	245.74	224.30	275.42
Fatigue Life (Cycles)	$1 \times 10^{10}$	$1.85 \times 10^8$	6656
Minimum Fatigue Factor of Safety	1.3223	0.8723	0.6831

According to our results, the Ti-6Al-4V rod exhibits a higher fatigue resistance than the CoCr alloy and AISI 316L; these results are in agreement with studies carried out with the CoCr alloy material [34], [38]. However, with AISI 316L, there are no precedents for fatigue studies on hip stem implants, and these results will serve as precedents for future research.

Finally, our results suggest that Ti-6Al-4V is the most suitable material for femoral hip stem applications due to its excellent stress ratio, fatigue life, and safety factor. Although CoCr alloy shows good mechanical performance, its fatigue life and lower safety factor limit its potential compared to Ti-6Al-4V. AISI 316L, on the other hand, has a low fatigue safety level. This work not only reinforces the conclusions of previous studies but also opens new lines of research on implants with AISI 316L material (see Table 2).

## 4 CONCLUSIONS

In this study, static, transient, and fatigue analyses of the hip femoral stem were performed using the finite element method with Ansys software, and three materials (Ti-6Al-4V, CoCr alloy, and AISI 316L) were evaluated. The results showed that CoCr alloy exhibits the best structural stiffness and the least deformation, making it an optimal choice for hip implants requiring stability and strength. On the other hand, Ti-6Al-4V stood out for its superiority in terms of fatigue resistance and implant life, being the strongest under cyclic loading conditions. AISI 316L, although it exhibited high maximum stresses, exhibited low fatigue resistance and a lower fatigue safety factor compared to the other two materials. In conclusion, Ti-6Al-4V was identified as the most promising material due to its excellent combination of durability and fatigue resistance, which ensures superior performance and serviceable beam time in hip implant design.

Higher stresses and strains were observed during transient analysis than static analysis, highlighting the importance of considering dynamic conditions in implant design. This analysis finds dynamic and time-varying loads, which are not considered in static analysis.

Future research work will be beneficial for investigating the use of coatings and surface treatments to conduct tribological studies and further evaluate their corrosion resistance. Finally, clinical studies assessing the performance of these materials in real patients will provide crucial data for the continued improvement of hip implant designs.

## 5 REFERENCES

- [1] S. Zengah, A. Mankour, S. Abderahmane, H. Salah, A. Mallek, and M. M. Bouziane, "Numerical analysis of the crack growth path in the cement of hip spacers," *Frat. ed Integrita Strutt.*, vol. 16, no. 61, pp. 266–281, 2022. <https://doi.org/10.3221/IGF-ESIS.61.18>
- [2] G. Bergmann *et al.*, "Hip contact forces and gait patterns from routine activities," *J. Biomech.*, vol. 34, no. 7, pp. 859–871, 2001. [https://doi.org/10.1016/S0021-9290\(01\)00040-9](https://doi.org/10.1016/S0021-9290(01)00040-9)
- [3] M. Fu, H. Zhou, Y. Li, H. Jin, and X. Liu, "Global, regional, and national burdens of hip osteoarthritis from 1990 to 2019: Estimates from the 2019 global burden of disease study," *Arthritis Res. Ther.*, vol. 24, 2022. <https://doi.org/10.1186/s13075-021-02705-6>
- [4] S. Affatato, A. Ruggiero, and M. Merola, "Advanced biomaterials in hip joint arthroplasty. A review on polymer and ceramics composites as alternative bearings," *Compos. Part B Eng.*, vol. 83, pp. 276–283, 2015. <https://doi.org/10.1016/j.compositesb.2015.07.019>

- [5] L. Petrolo, D. Testi, F. Taddei, and M. Viceconti, "Effect of a virtual reality interface on the learning curve and on the accuracy of a surgical planner for total hip replacement," *Comput. Methods Programs Biomed.*, vol. 97, no. 1, pp. 86–91, 2010. <https://doi.org/10.1016/j.cmpb.2009.11.002>
- [6] A. A. Oshkour, N. A. A. Osman, M. Bayat, R. Afshar, and F. Berto, "Three-dimensional finite element analyses of functionally graded femoral prostheses with different geometrical configurations," *Mater. Des.*, vol. 56, pp. 998–1008, 2014. <https://doi.org/10.1016/j.matdes.2013.12.054>
- [7] M. Sloan, A. Premkumar, and N. P. Sheth, "Projected volume of primary total joint arthroplasty in the U.S., 2014 to 2030," *J. Bone Jt. Surg.*, vol. 100, no. 17, pp. 1455–1460, 2018. <https://doi.org/10.2106/JBJS.17.01617>
- [8] 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>
- [9] K. N. Chethan, N. Shyamasunder Bhat, M. Zuber, and B. Satish Shenoy, "Evolution of different designs and wear studies in total hip prosthesis using finite element analysis: A review," *Cogent Eng.*, vol. 9, no. 1, 2022. <https://doi.org/10.1080/23311916.2022.2027081>
- [10] G. Kharmanda, "Integration of multi-objective structural optimization into cementless hip prosthesis design: Improved Austin-Moore model," *Comput. Methods Biomech. Biomed. Engin.*, vol. 19, no. 14, pp. 1557–1566, 2016. <https://doi.org/10.1080/10255842.2016.1170121>
- [11] 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," *Int. J. Model. Simul.*, vol. 43, no. 5, pp. 797–811, 2023. <https://doi.org/10.1080/02286203.2023.2212346>
- [12] S. Savilahti, I. Myllyneva, K. J. J. Pajamäki, and T. S. Lindholm, "Survival of Lubinus straight (IP) and curved (SP) total hip prostheses in 543 patients after 4–13 years," *Arch. Orthop. Trauma Surg.*, vol. 116, pp. 10–13, 1997. <https://doi.org/10.1007/BF00434092>
- [13] N. Shaikh, B. Satish Shenoy, N. Shyamasunder Bhat, S. Shetty, and K. N. Chethan, "Wear estimation at the contact surfaces of oval shaped hip implants using finite element analysis," *Cogent Eng.*, vol. 10, no. 1, 2023. <https://doi.org/10.1080/23311916.2023.2222985>
- [14] B. R. Rawal, A. Yadav, and V. Pare, "Life estimation of knee joint prosthesis by combined effect of fatigue and wear," *Procedia Technol.*, vol. 23, pp. 60–67, 2016. <https://doi.org/10.1016/j.protcy.2016.03.072>
- [15] M. Belwanshi, P. Jayaswal, and A. Aherwar, "Wear and fatigue behaviour investigation of hip implant head-stem interface using finite element analysis," *Mater. Today, Proc.*, vol. 56, pp. 2893–2901, 2022. <https://doi.org/10.1016/j.matpr.2021.10.188>
- [16] C. Gutmann, N. Shaikh, B. Satish Shenoy, N. Shyamasunder Bhat, L. G. Keni, and K. N. Chethan, "Wear estimation of hip implants with varying chamfer geometry at the trunnion junction: A finite element analysis," *Biomed. Phys. Eng. Express*, vol. 9, p. 035004, 2023. <https://doi.org/10.1088/2057-1976/acb710>
- [17] A. T. Alpkaya and S. Mihcin, "Dynamic computational wear model of PEEK-on-XLPE bearing couple in total hip replacements," *Med. Eng. Phys.*, vol. 117, 2023. <https://doi.org/10.1016/j.medengphy.2023.104006>
- [18] M. Merola and S. Affatato, "Materials for hip prostheses: A review of wear and loading considerations," *Materials*, vol. 12, no. 3, p. 495, 2019. <https://doi.org/10.3390/ma12030495>
- [19] N. Sahai *et al.*, "Designing & simulation of a lightweight hip implant stem: An FEM based approach," *Adv. Mater. Process. Technol.*, vol. 8, no. sup3, pp. 1126–1134, 2022. <https://doi.org/10.1080/2374068X.2021.1934646>
- [20] J. Triyono, A. R. Prabowo, and J. M. Sohn, "Dynamic analysis of different fenestration design in artificial hip joint using finite element analysis," in *AIP Conference Proceedings*, 2023, p. 030058. <https://doi.org/10.1063/5.0138985>

- [21] T. Joshi and G. Gupta, "Effect of dynamic loading on hip implant using finite element method," *Materials Today: Proceedings*, vol. 46, pp. 10211–10216, 2021. <https://doi.org/10.1016/j.matpr.2020.11.378>
- [22] 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>
- [23] M. E. Shehata, K. B. Mustapha, and E. M. Shehata, "Finite element and multivariate random forests modelling for stress shield attenuation in customized hip implants," *Forces Mech.*, vol. 10, p. 100151, 2023. <https://doi.org/10.1016/j.finmec.2022.100151>
- [24] M. Guzmán *et al.*, "Finite element assessment of a hybrid proposal for hip stem, from a standardized base and different activities," *Appl. Sci.*, vol. 12, no. 16, 2022. <https://doi.org/10.3390/app12167963>
- [25] Y. K. Cheah, A. H. Azman, and M. Y. Bajuri, "Finite-element analysis of load-bearing hip implant design for additive manufacturing," *J. Fail. Anal. Prev.*, vol. 22, pp. 356–367, 2022. <https://doi.org/10.1007/s11668-021-01304-6>
- [26] 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," *J. Inst. Eng. Ser. D.*, vol. 105, pp. 1–20, 2024. <https://doi.org/10.1007/s40033-023-00447-0>
- [27] 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," in *AIP Conference Proceedings*, 2022, p. 030008, 2022. <https://doi.org/10.1063/5.0110578>
- [28] J. Triyono, A. R. Prabowo, and J. M. Sohn, "Investigation of meshing strategy on mechanical behaviour of hip stem implant design using FEA," *Open Eng.*, vol. 10, no. 1, pp. 769–775, 2020. <https://doi.org/10.1515/eng-2020-0087>
- [29] G. Zhang *et al.*, "Parametric analysis of the effect of impaction load on the stability of head-neck junction in total hip arthroplasty," *Clin. Biomech.*, vol. 94, 2022. <https://doi.org/10.1016/j.clinbiomech.2022.105633>
- [30] K. N. Chethan, M. Zuber, N. Shyamasunder Bhat, B. Satish Shenoy, and C. R. Kini, "Static structural analysis of different stem designs used in total hip arthroplasty using finite element method," *Heliyon*, vol. 5, no. 6, p. e01767, 2019. <https://doi.org/10.1016/j.heliyon.2019.e01767>
- [31] 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, 2022. <https://doi.org/10.1080/23311916.2022.2027080>
- [32] M. I. Ammarullah *et al.*, "Von Mises stress analysis of CoCrMo-on-CoCrMo hip prosthesis based on body mass index: 2D FEM estimation," in *AIP Conference Proceedings*, 2023, p. 030001. <https://doi.org/10.1063/5.0128350>
- [33] J. V. Corda, K. N. Chethan, N. Shyamasunder Bhat, S. Shetty, B. Satish Shenoy, and M. Zuber, "Finite element analysis of elliptical shaped stem profile of hip prosthesis using dynamic loading conditions," *Biomed. Phys. Eng. Express*, vol. 9, no. 6, p. 065028, 2023. <https://doi.org/10.1088/2057-1976/acfe14>
- [34] J. Reginald, M. Kalayarasan, K. N. Chethan, and P. Dhanabal, "Static, dynamic, and fatigue life investigation of a hip prosthesis for walking gait using finite element analysis," *Int. J. Model. Simul.*, vol. 43, no. 5, pp. 797–811, 2023. <https://doi.org/10.1080/02286203.2023.2212346>
- [35] J. Triyono, A. R. Prabowo, and J. M. Sohn, "Dynamic analysis of different fenestration design in artificial hip joint using finite element analysis," in *AIP Conference Proceedings*, 2023, p. 2674. <https://doi.org/10.1063/5.0138985>
- [36] T. Joshi and G. Gupta, "Effect of dynamic loading on hip implant using finite element method," *Materials Today: Proceedings*, vol. 46, pp. 10211–10216, 2021. <https://doi.org/10.1016/j.matpr.2020.11.378>

- [37] 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, pp. 959–980, 2023. <https://doi.org/10.1007/s11012-023-01661-6>
- [38] 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," *J. Appl. Eng. Sci.*, vol. 21, no. 3, pp. 896–907, 2023. <https://doi.org/10.5937/jaes0-44094>
- [39] A. D. Oza, N. Gupta, and R. Singh, "Design and non-linear finite element analysis of titanium-based femoral hip-stem for Indian population," *Int. J. Interact. Des. Manuf.*, vol. 17, pp. 2489–2493, 2023. <https://doi.org/10.1007/s12008-022-01047-0>
- [40] L. Xie *et al.*, "Numerical analysis and experimental validation on residual stress distribution of titanium matrix composite after shot peening treatment," *Mech. Mater.*, vol. 99, pp. 2–8, 2016. <https://doi.org/10.1016/j.mechmat.2016.05.005>
- [41] 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, no. 1, p. 9205930, 2021. <https://doi.org/10.1155/2021/9205930>
- [42] 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, pp. 4390–4397, 2021. <https://doi.org/10.1016/j.matpr.2020.11.852>
- [43] 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>
- [44] 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, 2022. <https://doi.org/10.1080/23311916.2022.2027080>
- [45] M. G. Gok and O. Cihan, "Numerical analysis of the application of different lattice designs and materials for reciprocating engine connecting rods," *Sci. Iran.*, vol. 29, no. 5, pp. 2362–2373, 2022. <https://doi.org/10.24200/sci.2022.59400.6216>
- [46] M. T. Mezher, O. S. Barrak, and N. S. M. Namer, "Modelling and experimental study of dissimilar arc stud welding of AISI 304L to AISI 316L stainless steel," *Int. J. Integr. Eng.*, vol. 14, no. 6, pp. 88–101, 2022. <https://doi.org/10.30880/ijie.2022.14.06.009>
- [47] M. Niinomi, "Fatigue failure of metallic biomaterials," in *Metals for Biomedical Devices*, pp. 153–188, 2019. <https://doi.org/10.1016/B978-0-08-102666-3.00005-5>
- [48] ASTM, "F 2996-20 standard practice for Finite Element Analysis (FEA) of non-modular metallic orthopaedic hip femoral stems," *ASTM Int. Conshohocken, PA*, vol. 13.02, p. 11, 2020. <https://doi.org/10.1520/F2996-20>
- [49] 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.," 2020. [Online]. Available: <https://www.iso.org/standard/42769.html>
- [50] J. Triyono, A. R. Prabowo, and J. M. Sohn, "Investigation of meshing strategy on mechanical behaviour of hip stem implant design using FEA," *Open Eng.*, vol. 10, no. 1, pp. 769–775, 2020. <https://doi.org/10.1515/eng-2020-0087>
- [51] G. Zhang *et al.*, "Parametric analysis of the effect of impaction load on the stability of head-neck junction in total hip arthroplasty," *Clin. Biomech.*, vol. 94, 2022. <https://doi.org/10.1016/j.clinbiomech.2022.105633>
- [52] 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, 2020. <https://doi.org/10.1080/23311916.2020.1719575>

- [53] M. Wu, J. Mu, L. Zhuang, Y. Kong, and X. Zhou, "Fatigue analysis of injector body based on ANSYS workbench," *Vibroengineering Procedia*, vol. 30, pp. 193–198, 2020. <https://doi.org/10.21595/vp.2019.21183>
- [54] 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," *Int. J. online Biomed. Eng.*, vol. 20, no. 2, pp. 16–30, 2024. <https://doi.org/10.3991/ijoe.v20i02.46817>
- [55] K. Nabudda *et al.*, "Identification of flexural modulus and poisson's ratio of fresh femoral bone based on a finite element model," *Int. J. Online Biomed. Eng.*, vol. 18, no. 4, pp. 94–105, 2022. <https://doi.org/10.3991/ijoe.v18i04.28939>
- [56] J. L. Serna-Landivar, D. M. Anticona-Valderrama, and P. A. Flores, "Design and validation through finite elements of a crusher for the plastic industry," in *Proceedings of the LACCEI International Multi-conference for Engineering, Education and Technology*, 2023. [Online]. Available: <https://www.scopus.com/inward/record.uri?eid=2-s2.0-85172289103&partnerID=40&md5=4966b1853c59cc4f58ba9a09d0bb8c33>

## 6 AUTHORS

**Daniela Milagros Anticona-Valderrama**, Degree in physical mathematics, master's degree in education administration, PhD in public management and governance at the Universidad Cesar Vallejo, PhD candidate in didactics of scientific research at the National University of Panama. Experience 6 years as a professor at Universidad Tecnológica del Perú, Universidad Autónoma del PERÚ, Universidad Cesar Vallejo and Universidad Privada del Norte. Research areas: Education, environmental management, mathematics, engineering, and technology (E-mail: [daniela.anticona@upn.pe](mailto:daniela.anticona@upn.pe)).

**José L Serna-Landivar** is MSC Candidate in Mechanical Engineering from Pontificia Universidad Católica del Perú (PUCP) and Eng. in Mechanical Engineering from Universidad Tecnológica de Perú. Experience in computer-aided design in INVENTOR, SOLIDWORKS, STEEL ADVANCE, TEKLA STRUCTURES, and simulation with ANSYS SAP2000 computational tools. Research areas: Applied mechanics, mechanical design, finite element method for numerical solution of structural problems (E-mail: [u18101283@utp.edu.pe](mailto:u18101283@utp.edu.pe)).

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

**Manuel López Miranda**, Dr. in industrial engineering completed at UNI, master's degree in industrial engineering at UNI, industrial engineer at URP, professor at URP Professional with more than 25 years of experience in metal mechanics CAD/CAM/CNC/CAPP (E-mail: [mlopez@urp.edu.pe](mailto:mlopez@urp.edu.pe)).

**Maria Ysabel Garcia-Alvarez**, Civil Engineer graduated from the National University San Luis Gonzaga, PhD candidate in Sciences from the National University Hermilio Valdizán, Doctor in Public Management and Governance and Master's in civil engineering with a mention in Construction Business Management from Universidad César Vallejo. Experience 6 years as a professor at Universidad César Vallejo and the Universidad Tecnológica del Perú. University Professor Researcher (E-mail: [c19447@utp.edu.pe](mailto:c19447@utp.edu.pe)).

**Leslie Katerine Serna Landivar**, Systems and Information Technology Engineer at the UAP, Master's in Public Management at the UCV, professor at the PNP Piura Technical Professional Higher Education School (E-mail: [lksernal@ucvvirtual.edu.pe](mailto:lksernal@ucvvirtual.edu.pe)).