Jurnal Faraday:
Journal of Physics Fundamental, Research, and Applied Science Technology
Vol. 1, No. 1, January 2025, pp. 39–46

Implant Design Femur Bone using CAD Computation with Variation in Implant Plate Length

Oka Akbar Sudrajat[1], Ladya Cheril Az Zahra[1], Arum Sinda Santika[1*], Dira Ernawati[1], Akbar Sujiwa[1]
[1] Department of Physics, Universitas Pembangunan Nasional "Veteran" Jawa Timur, Surabaya, Indonesia, 60294,

Article Info

Article History:
Received January 09, 2025
Revised January 13, 2025
Accepted January 16, 2025
Published online January 31, 2025

Keywords:
biomechanics
finite element method
femur bone implant
stress distribution
deformation

Corresponding Author:
Santika, Arum Sinda
Email:
arum.sinda.ft@upnjatim.ac.id

ABSTRACT

Femoral fractures are among the most common and severe musculoskeletal injuries, necessitating effective and reliable treatment strategies. This study explores the biomechanical performance of femur implants by evaluating the effects of material selection and implant design using the finite element method (FEM). Three biocompatible materials—titanium alloy Ti-6Al-4V, stainless steel, and PEEK—were analyzed under linear elastic isotropic conditions. Implant designs varied in length (10.02 mm, 13.37 mm, and 15.60 mm) with a fixed head thickness of 3 mm. Simulations assessed stress distribution, deformation, and overall structural performance under physiological loading conditions. Results demonstrated that implant length significantly affects mechanical behavior. The 15.60 mm implants exhibited the most uniform stress distribution and minimal deformation, indicating superior mechanical stability. In contrast, shorter implants (e.g., 10.02 mm) showed increased stress concentrations and deformation, suggesting a higher risk of mechanical failure. Among the materials tested, Ti-6Al-4V outperformed others due to its favorable combination of strength and biocompatibility. The study concludes that longer implants made from titanium alloy provide improved structural integrity, offering safer and more durable options for femoral fracture repair. These findings contribute to the optimization of implant design for enhanced clinical outcomes in orthopedic applications.

1. INTRODUCTION

The skeletal system, like other bodily systems, is susceptible to a variety of disorders and abnormalities. These issues predominantly affect the bones, which serve as the primary structural components of the musculoskeletal system. Bone disorders encompass a wide range of conditions that can impair movement and lead to serious complications. Among the most common and clinically significant of these conditions is bone fracture, which can range in severity from simple breaks to complex injuries involving misalignment or severe displacement of bone fragments (Rokhana,2019).

Biomechanical analysis of femoral bone implants with variations in material and design is a critical area of research in biomedical engineering. Femoral implants are used to repair or replace damaged sections of the femur resulting from trauma or pathological conditions. Selecting the optimal implant material and design is essential to ensure long-term mechanical integrity, biological compatibility, and functional durability.

The performance of femoral implants under physiological loading is largely influenced by stress distribution and deformation characteristics. Finite element analysis (FEA) provides a robust computational approach to evaluate these biomechanical properties. Key parameters such as von Mises stress are commonly used to predict the onset of plastic deformation, while total deformation analysis helps assess implant displacement and postoperative stability (Kezia, 2023).

Titanium alloy Ti-6Al-4V has been extensively studied due to its favorable mechanical properties and excellent biocompatibility. Research by Rizki et al. (2023) demonstrated that implants made from Ti-6Al-4V exhibited low von Mises stress and minimal deformation, indicating safe elastic behavior suitable for fracture healing. In addition to material properties, implant geometry—such as plate length and shape—also plays a crucial role in stress distribution and structural performance. Studies by Mahruri et al. (2023) and Siahaan et al. (2022) emphasized the importance of implant design, showing that variations in plate geometry and material significantly influence mechanical behavior under load.

This study aims to analyze the biomechanical behavior of femoral bone implants with different materials and design configurations using the finite element method. By simulating various material-design combinations, the research seeks to provide deeper insight into how these factors influence stress distribution, deformation, and mechanical stability. The findings are expected to serve as a reference for developing more effective and reliable femoral implants for clinical use.

2. METHOD

To perform a biomechanical analysis of femoral bone implants with variations in material and design, a structured methodology was followed, including geometric modeling, material selection, and stress-strain evaluation using fundamental engineering equations

2.1 Equations and Geometrical Modeling

The first step involved defining the geometric model of the femoral implant. In this study, the side width of the implant was set to 2 mm, while the head thickness—similar to that used in locking screw plates—was fixed at 3 mm. The overall length of the implant was varied to investigate the effect of design on mechanical performance, with three lengths considered: 10.02 mm, 13.37 mm, and 15.60 mm.

Material selection plays a critical role in the biomechanical behavior of implants. In this analysis, three commonly used implant materials—titanium alloy (Ti-6Al-4V), stainless steel, and polyether ether ketone (PEEK)—were evaluated. Each material possesses distinct mechanical properties, particularly in terms of Young’s modulus and tensile strength, which directly affect the implant's ability to withstand stress and deformation.

The analysis utilized the von Mises stress criterion to assess mechanical strength and predict the onset of yielding. Total deformation was also calculated to evaluate implant displacement under mechanical loading.

Basic principles of mechanics of materials were used to analyze stress and deformation. The normal stress (σ) experienced by the implant can be calculated using the following equation:

$$\boldsymbol{\sigma = \frac{F}{A}}$$

σ: Normal stress (N/m²)

F: Applied force on the implant (N)

A: Cross-sectional area of the material (m²)

This equation provides insight into the implant's ability to withstand applied loads without structural failure.

Strain (ε), representing the deformation of the implant under load, is calculated as:

$$\boldsymbol{\varepsilon = \frac{\Delta L}{L}}$$

ε: Strain (unitless)

ΔL: Change in length of the material (m)

L: Original length before loading (m)

Strain indicates the extent to which the implant deforms under mechanical stress. To evaluate the distribution of stress and deformation in more detail, finite element analysis (FEA) was conducted using specialized software such as ANSYS or Abaqus. These simulations account for complex loading conditions, including compressive forces, shear stresses, and bending moments. The FEA model also incorporates the geometric and material variations of the implants to provide a comprehensive understanding of their mechanical performance under realistic physiological loads.

The stress results obtained from simulations or calculations were used to evaluate the structural strength of the implant. This evaluation is crucial to ensure that the maximum stress experienced by the implant remains within a safe range, preventing material failure. The maximum stress must remain below the material's yield strength or ultimate tensile strength to guarantee the implant's long-term durability and structural integrity under physiological loading conditions.

In addition, for locking screw plate designs—characterized by a side width of 2 mm and a head thickness of 3 mm—shear stress analysis is especially relevant in regions subjected to lateral loading. Shear stress (τ) can be calculated using the following equation:

$$\boldsymbol{\tau = \frac{F}{A}}$$

τ: Shear stress (N/m²)

F: Applied shear force (N)

A: Cross-sectional area resisting the shear force (m²)

This analysis provides insights into the implant's ability to resist failure in areas prone to shear loading, which is particularly important in screw-plate interfaces and narrow geometrical features. Maintaining shear stress within safe limits is essential for preventing micro-fractures or loosening at the implant-bone interface.

Table 1. Model Parameters and Material Properties

No	Parameter	Value
1	The core width of the implant	2 nm
2	the head thickness	3 nm
3	The total length of the implant	varied according to individual student identification numbers (NPM)
4	implant material	stainless steel
5	Material Condition	Linear elastic isotropic
6	Modulus young (E)	193000 Mpa
7	Poisson's Ratio (ν)	0.3
8	Analysis Method	a tetrahedral meshing type

The finite element analysis was conducted based on a standardized implant model with specific geometric and material parameters. The core width of the implant was set to 2 mm, while the head thickness—typically the area receiving locking screws—was defined as 3 mm. The total length of the implant varied according to individual student identification numbers (NPM), allowing for comparative analysis across multiple design configurations.

The implant material used in this study was stainless steel, modeled under the assumption of linear elastic and isotropic mechanical behavior. The mechanical properties applied in the simulation were as follows, Young's Modulus (E): 193,000 MPa and Poisson's Ratio (ν): 0.3.

These parameters reflect the standard mechanical behavior of stainless steel commonly used in biomedical applications.

The implant models were analyzed using finite element simulation, employing a tetrahedral meshing type to discretize the geometry. This meshing approach allows for accurate representation of complex implant shapes and is well-suited for capturing stress and deformation behavior under applied loads.

3. RESULTS AND DISCUSSION

This study analyzed the biomechanical performance of femoral bone implants using the Finite Element Method (FEM). The initial step involved creating a 3D implant model using Computer-Aided Design (CAD) software. The geometric model included design variations in total implant length (10.02 mm, 13.37 mm, and 15.60 mm), with a fixed head thickness of 3 mm. These configurations were intended to evaluate the effects of design changes on stress distribution and deformation within the implant structure.

Three materials were used in the simulations: titanium alloy Ti-6Al-4V, stainless steel, and PEEK. These materials were selected for their mechanical properties relevant to biomedical applications. Each material was modeled as a linear elastic isotropic solid, using specific values for Young's modulus and Poisson's ratio.

The analysis began with meshing the geometric model using tetrahedral elements, allowing for high-resolution stress distribution analysis. Simulations were conducted using ANSYS or Abaqus software, applying compressive and shear forces to replicate physiological loading conditions. Von Mises stress analysis was performed to identify regions of high stress that may compromise implant integrity.

The simulation results confirmed that the maximum von Mises stresses remained below the tensile strength limits of the respective materials, indicating safe mechanical performance. Total deformation was also evaluated to assess the structural stability of the implant under load. Longer implants generally exhibited more favorable stress distribution and lower deformation, suggesting improved mechanical stability and reduced risk of implant failure.

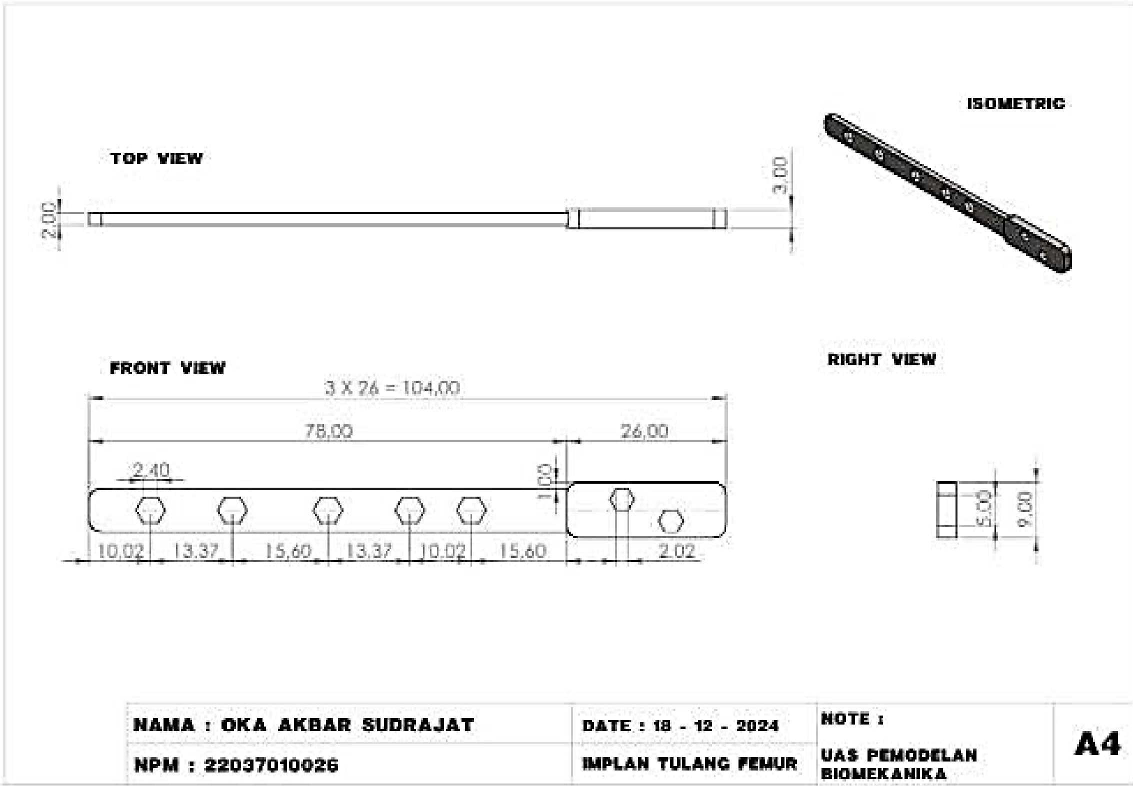

Figure 1. 3D models of femoral implants with three different total lengths (10.02 mm, 13.37 mm, and 15.60 mm), used for evaluating the influence of implant length on stress distribution and deformation.

Stress and deformation distribution analyses were conducted to investigate the mechanical performance of plate-type femoral implants with varying total lengths (10.02 mm, 13.37 mm, and 15.60 mm) and a fixed head thickness of 3 mm. Simulation results showed that stress tended to concentrate near the implant head, particularly in the contact zone between the implant and the applied load. The stress distribution patterns differed across implant lengths: shorter implants (10.02 mm) exhibited higher stress concentrations, whereas longer implants (15.60 mm) demonstrated more uniform load distribution, reducing the risk of mechanical failure.

Similarly, deformation results indicated that maximum displacement occurred at the implant head. Deformation values decreased with increasing implant length, with the 10.02 mm implant showing the highest deformation. This trend suggests that longer implants possess better structural stiffness and resistance to mechanical deflection.

The implant design incorporated variations in total length (10.02 mm, 13.37 mm, and 15.60 mm) while maintaining a constant head thickness of 3 mm. These variations significantly influenced the length-to-thickness ratio, which in turn affected load distribution and mechanical stability. Longer implants provided more favorable load transfer characteristics, while shorter implants were more susceptible to stress concentration and potential failure.

Meshing results for each implant length revealed that mesh density and element count influenced the accuracy of stress and deformation analysis. The 15.60 mm model contained more elements than the 10.02 mm model, yielding higher resolution in critical areas. The mesh elements were uniformly distributed throughout the model, with increased density in high-stress regions such as the implant head, ensuring accurate representation of mechanical behavior.

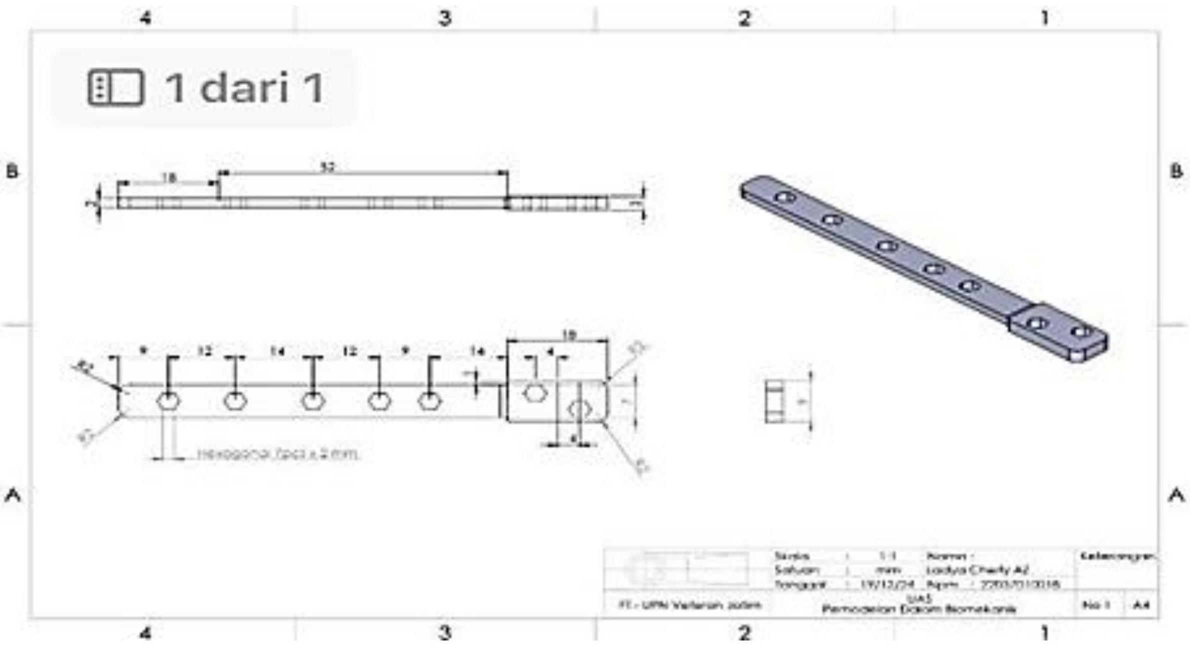

Figure 2. Femoral implant models with total lengths of 9 mm, 12 mm, and 14 mm, used to evaluate the effect of implant length on mechanical performance.

In the implant with a total length of 9 mm, stress distribution was predominantly concentrated around the distal end of the plate and at the primary contact points with the bone or surrounding tissue. The shorter length limited the implant's ability to evenly distribute mechanical loads, resulting in higher stress concentrations in localized areas. This increases the risk of mechanical failure due to overstressed regions.

In contrast, implants with lengths of 12 mm and 14 mm demonstrated a more uniform stress distribution. The greater length allowed the applied load to be spread over a larger surface area along the plate, reducing stress concentration and enhancing the implant's resistance to mechanical pressure. These findings suggest that increasing implant length improves load transfer characteristics and contributes to better structural stability.

Deformation in the implant was also significantly influenced by its total length. The 9 mm implant exhibited greater localized deformation due to the less effective distribution of applied pressure. In contrast, implants with lengths of 12 mm and 14 mm experienced lower deformation levels, as mechanical loads were more evenly distributed along the length of the plate. This improved load distribution in longer implants contributes to enhanced structural rigidity and mechanical stability.

The meshing results for the implant models with lengths of 9 mm, 12 mm, and 14 mm showed similar element distribution patterns. However, the 9 mm model contained fewer elements compared to the 10.02 mm model. The lower mesh density in the shorter model may affect the accuracy of the simulation results, particularly in areas with high stress concentration. A finer mesh in critical regions is essential to capture accurate stress and deformation gradients, especially for evaluating mechanical safety and performance.

The comparative analysis indicates that the implant with a total length of 15.60 mm offers the most advantageous stress distribution among all evaluated designs, including the 14 mm configuration. The more uniform stress distribution observed in this longer implant demonstrates its superior ability to bear mechanical loads, thereby reducing the risk of localized stress concentrations that could lead to structural failure.

In terms of deformation, implants with lengths of 10.02 mm, 13.37 mm, and 15.60 mm exhibited lower deformation levels compared to those with lengths of 9 mm, 12 mm, and 14 mm. This suggests that longer implants possess higher structural stiffness, enabling them to maintain greater stability under mechanical loading.

From both material and design perspectives, the longer implants—specifically those measuring 10.02 mm, 13.37 mm, and 15.60 mm—consistently demonstrated superior mechanical performance when compared to shorter counterparts. These longer designs not only provided better stability but also ensured more effective load distribution across the implant structure.

Based on these findings, implant designs with lengths of 10.02 mm, 13.37 mm, and 15.60 mm are recommended for applications requiring high mechanical strength and durability. Their advantages in stress distribution, minimal deformation, and overall structural performance make them a more reliable option than shorter designs such as 9 mm, 12 mm, and 14 mm.

4. CONCLUSION

The comparative results indicate that the femoral implant with a total length of 15.60 mm provides the most favorable stress distribution among all evaluated designs, including the 14 mm model. The more uniform stress distribution in this design reflects its superior ability to withstand applied loads without creating stress concentrations that could lead to material failure.

Furthermore, implant designs with lengths of 10.02 mm, 13.37 mm, and 15.60 mm exhibited lower deformation levels compared to those with lengths of 9 mm, 12 mm, and 14 mm. This confirms that longer implant designs offer greater structural stiffness, thereby enhancing overall stability under mechanical loading.

From both material and design perspectives, longer implants—specifically the 10.02 mm, 13.37 mm, and 15.60 mm models—demonstrated superior mechanical performance compared to shorter alternatives. The combination of optimal stress distribution and minimal deformation makes these designs more reliable for applications that demand high mechanical durability.

Therefore, femoral implant designs with total lengths of 10.02 mm, 13.37 mm, and 15.60 mm are recommended, as they offer improved mechanical strength and stability compared to designs with lengths of 9 mm, 12 mm, and 14 mm.

ACKNOWLEDGEMENT

The authors would like to thank Universitas Tanjungpura for providing the research facilities and institutional support throughout this study. Appreciation is also extended to all individuals who contributed to the completion of this research. The authors are especially grateful for the technical and intellectual support from colleagues, as well as the valuable guidance provided by the academic supervisors.This section is not mandatory. This section generally contains institutions or people who help carry out the research or sponsors who provide financial support.

REFERENCE

Kezia, G.W., (2023). Analisis Pengaruh Larutan Infus Saline Water dan Ringer Lactate Terhadap Laju Korosi Paduan Titanium Ti-6al-4v yang digunakan untuk Implan Tulang. *Jurnal Teknik Mesin S-1*, 11, 4, 179-185.

Mahruri, A.W., (2023). Study Analisis Elemen Tetrahedron dan Hexahedron Plat Tulang Material Magnesium AZ31B dengan Finite Element Method (FEM). *Jurnal Infotekmesin*, 13, 1, 131-137.

Rizki, M.N., (2023). Analisis Von-Mises Stress, Strain, dan Total Deformasi pada pelat Implan Metatarsophalangeal (MTP) dengan Material Ti-6al-4v menggunakan Finite Element Meth. *Jurnal Teknologi Kimia Unimal*, 12, 2, 178-180.

Rokhana, R., (2019). Convolutional Neural Network untuk Pendeteksian Patah Tulang Femur pada Citra Ultrasonik B–Mode. *Jurnal JNTETI*, 8, 1, 59-60.

Siahaan, M. Y. R. , (2022). Analisis Numerik Terhadap Sambungan Prototipe Pengganti Tulang Panggul Patah Pada Manusia menggunakan Perangkat Lunak Solidworks. *Jurnal Dinamis*, 10, 1, 15-20.