Volume 3, Number 2, 2026

Pages 196-207

Bioscience Insights

ISSN: 3080-3489 (Print)
ISSN: 3080-3497 (Online)

BiosciInsights.com

FINITE ELEMENT ANALYSIS OF FEMORAL FRACTURE FIXATION PLATES FOR OPTIMIZED STRESS DISTRIBUTION AND IMPLANT STABILITY

Kashaf Khanzada[*1], Sehreen Moorat[2], Natasha Mukhtiar[3], Shahida Qayoom[4], Aneeqa Khanzada[5]

[*1,2,3,4,5]*Institute of Biomedical Engineering and Technology, Liaquat University of Medical and Health Sciences, Jamshoro, 76090, Pakistan.*

[*1]khanzadakashaf@gmail.com,[2]natasha.mukhtiar@lumhs.edu.pk,[3]sehreen.moorat@lumhs.edu.pk, [4]shahidaqayoom0@gmail.com, [5]aneeqakhanzada1@gmail.com

Article Information

Article History

Received:	May 15, 2026
Revised:	June 27, 2026
Accepted:	August 12, 2026
Available Online:	August 31, 2026

Keywords:
Femur - Bone Fracture - Fixation plates - Stress analysis - Stress shielding

Abstract

This Research Study analyze the Biomechanical performance (Stress Strain and Displacement) Fixation Plates used for fixing the Femur Fracture. The Femur Anatomy and Fixation Plates were designed in CAD tool and further Biomechanical performance were analyzed using Comsol Multiphysics. In simulation Femur distal end kept Fixed and Applied load on femoral head to analyze the walking condition of human body. Results Finalized that overall deformation of Femur and Fracture plate was very low which means that the system was stable and can hold fractured bone properly. However the highest stress and strain were observed near the fracture region and at screw bone area because of greater portion of applied load concentrated here. Among all the materials (Stainless steel and Cobalt) which were analyzed Titanium is the one which shows best biomechanical performance because it provide good balance between Stiffness and stability. The appropriate material selection and using Finite element analysis made the stress Distribution to the bone plate easy. This study help to reduce Stress Shielding and ensure the successful healing of the fractured bone. It recommends the use of mechanically stable materials such as titanium alloys along with FEM-assisted pre-surgical planning.

INTRODUCTION

In both developed and developing nations, trauma is the leading cause of death and disability. One of the most frequent injuries is a femur bone fracture. Because the human body's femurs are its strongest, longest, and heaviest, femur fractures are a significant area of research in orthopedic trauma. Injury to normal healthy femur is commonly induced by high energy events such as athletic accident, motor vehicle accident or fall [4].

The femur is the leg's most proximal bone in vertebrates that can walk and jump [1][2]. It supports the largest percentage of body weight during routine weight bearing activities and is the longest, largest, and strongest bone in humans when compressed. Its length, which helps determine stature from skeletal remains, is around 26% of an individual's overall height in anthropology [3][4]

The femur forms two primary joints: the hip joint at its proximal end, where the femoral head articulates with the pelvic acetabulum, and the knee joint at its distal end, where the condyles articulate with the tibia and patella. The distal end is made up of the medial and lateral condyles as well as the intercondylar fossa, whereas the higher anatomical area is made up of the femoral neck, greater and lesser trochanters, and proximal condyles. The femoral shaft has a wider distal portion and is long, thin, and almost cylindrical [5]. From a biomechanical standpoint, walking, running, and jumping put the femur under significant compressive and bending forces. It is a crucial bone for assessing implant behavior and fracture-fixation techniques because stress concentrations are especially high in the femoral neck and fracture-prone areas [6].

Medical devices such as implants have been created especially to treat complex fractures. Modern techniques that speed up secondary wound healing have gradually replaced traditional treatments for open fractures. Elderly people are treated with surgery and improved fixation devices to try to improve their mobility. An orthopedic surgeon will simply straighten the fractured bones and hold off on treating the next patient until the fractures have healed. The patient bears no responsibility for the formation of a union as long as the surgeon properly aligns and implants the fractures.

Bone plates are an essential form of internal fixation device for fixing human bone fractures and are often comprised of stainless steel and titanium alloys. Despite their great strength, ductility, and resistance to wear, metals are susceptible to electro-magnetic field sensitivity, enhanced thermal conductivity, and corrosion under physiological conditions. The Young's modulus of Ti-6Al-4V is five times greater than that of human cortical bone. The much higher stiffness of metallic plates for bone fixation with respect to cortical bones leads to the increased transfer of larger stresses through plates rather than through bones [7]. Due to its superior corrosion resistance and biocompatibility, pure titanium metal is also one of the most used materials for bone plates. Due to its hexagonal crystal structure, titanium has less ductility than stainless steel. Compared to stainless steel plates, this makes shaping titanium plates challenging because titanium plates have an elastic module of 68 GPa as opposed to stainless steel 200 GPa, they also provide less stress-shielding to bone (for the same geometries) after healing [17]. They cannot be contoured as easily as SS plates, though.

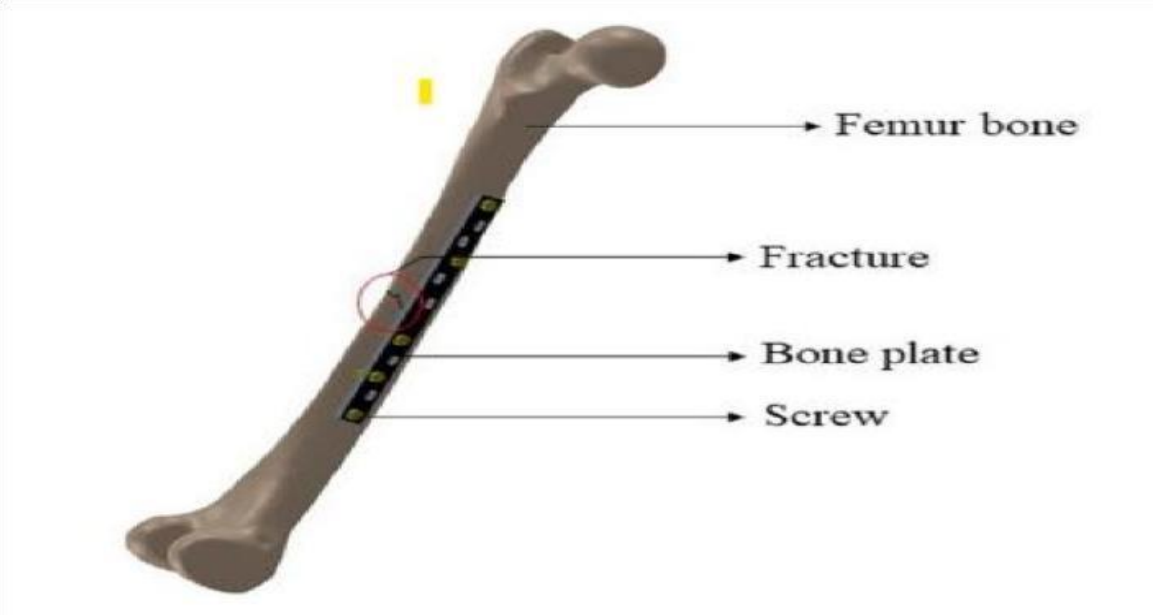


Figure 1: Bone plate attached to Fractured Femur Bone [8]

Stress shielding is also a critical issue in hip endoprostheses, where a large mismatch between the stiffness of the implant and the surrounding bone reduces the physiological load transferred to the femur. If the implant is much stiffer than the surrounding tissue the mechanical load may not be properly transferred to the bone. With passing time this improper distribution of load can cause bone reabsorption and structural weakening. Recent study concluded that designing implant stiffness to more closely match the properties of bone reduce stress shielding allowing bone to experience appropriate mechanical stress and improving the long term performance and stability of implant. Additive manufacturing or 3D printing now a days very effective way to control implant geometry and stiffness, enabling the design of bone plate system that distribute load evenly across the bone while remaining mechanically safe under critical load. Long term studies and numerical analyses have shown that reduced stiffness implant increases bone stress, decrease bone reabsorption, and improve prosthesis function and service life [9].

To better study the mechanical behavior of fixation plates, three dimensional geometry was created in CAD tool SOLIDWORKS [10] with careful control over thickness contour and screw hole configuration. The CAD design were imported into Comsol Multiphysics through suitable file format where finite element analysis was used to analyze biomechanical performance under physiology loading conditions [11][12]. This simulation help to identify the area of stress concentration and regions of inadequate load sharing, which is important for preventing biomechanical performance complications and optimizing long term bone and implant performance [13][7][14][15] [16] [17][18].

MATERIAL AND METHOD:

GEOMETRY:

Design of Bone Plate: The model of screw plate fracture system was designed by computer aided design software SolidWorks. The reference model was considered for the design geometry on software as shown in figure 2. According to figure 2 the First part of Fracture system model of Plate was designed as shown in figure 3, shows the 3D model of the fixation plate. The design includes a long rectangular plate with a series of evenly spaced holes along its length. These holes are designed to accommodate fixation screws that attach the plate firmly to the bone. The plate geometry is designed to match the general anatomical structure of the femoral shaft so that it can lie properly along the bone surface. The plate dimensions are as follows, the length of the plate is 135mm, the width of the plate is 16mm, and the thickness of the plate

is 4.8mm. Screw Diameter of 4.50mm was taken for design. Second Part of Femur Fracture system Screw was modelled as shown in figure 4. During design different SolidWorks commands like Sketch (rectangle), Extruded Base, Hole Wizard, Linear Pattern, Fillet and Extruded cut were used. Figure 5 shows the Fixation of Figure 3 and 4 by using Assembly part of SolidWorks.

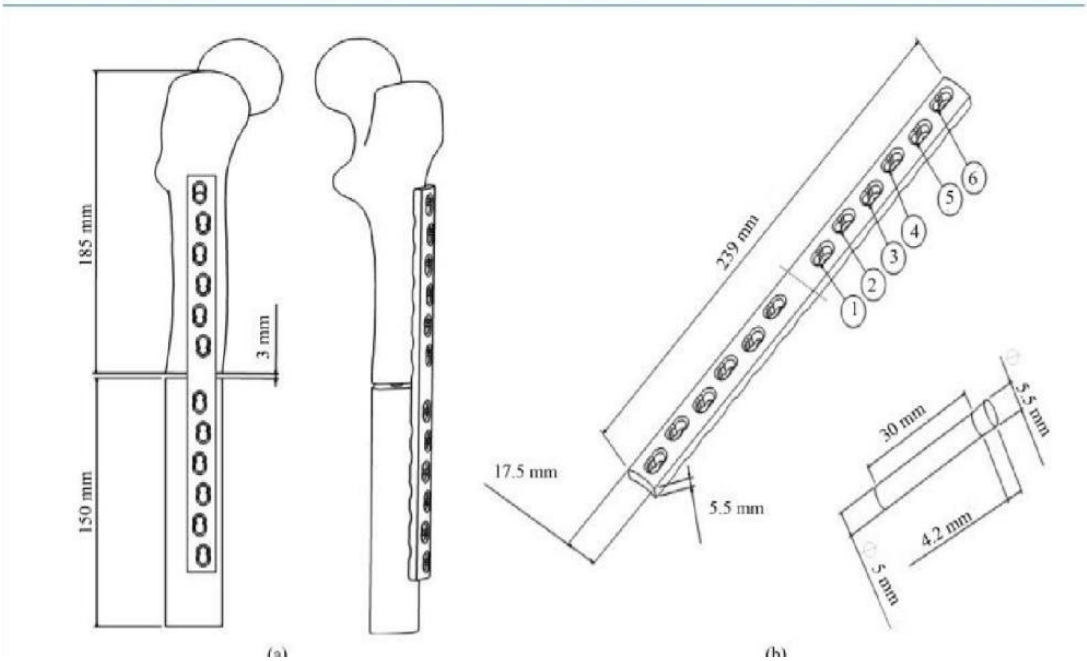


Figure 2: Reference Model for design [2]

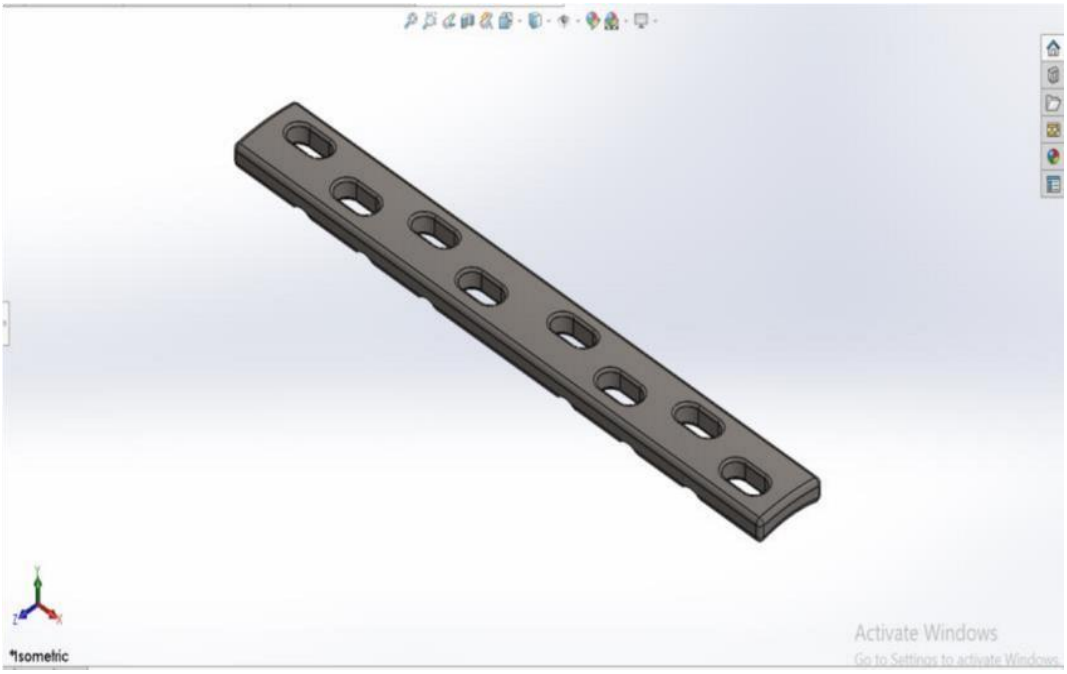


Figure 3: Model of Plate

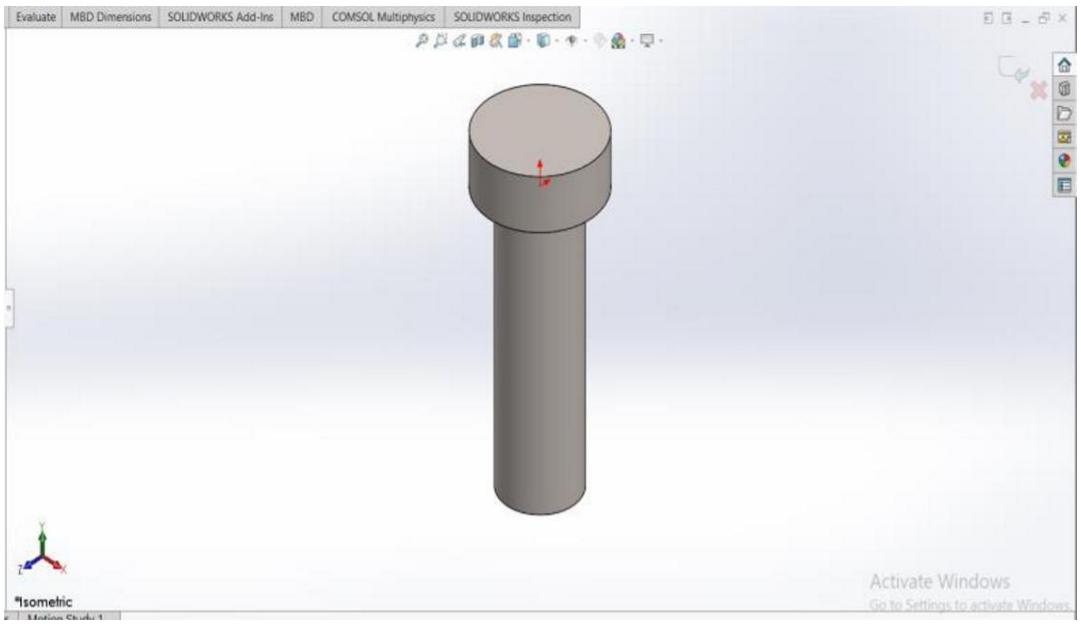


Figure 4: Model of Screw

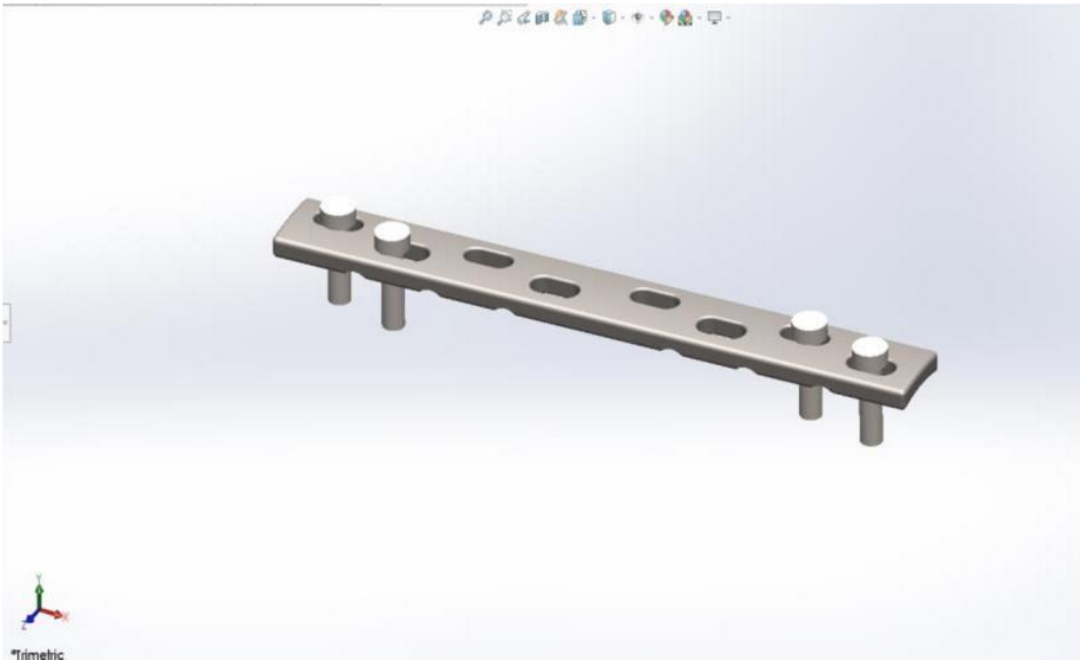


Figure 5: Model of Plate Screw Fracture System

Design of Bone: Femur Bone was also designed using SolidWorks as shown in figure 6. The Bone dimensions are taken as: the diameter of Bone is 50mm, the Length is 480mm and the width is 190mm.

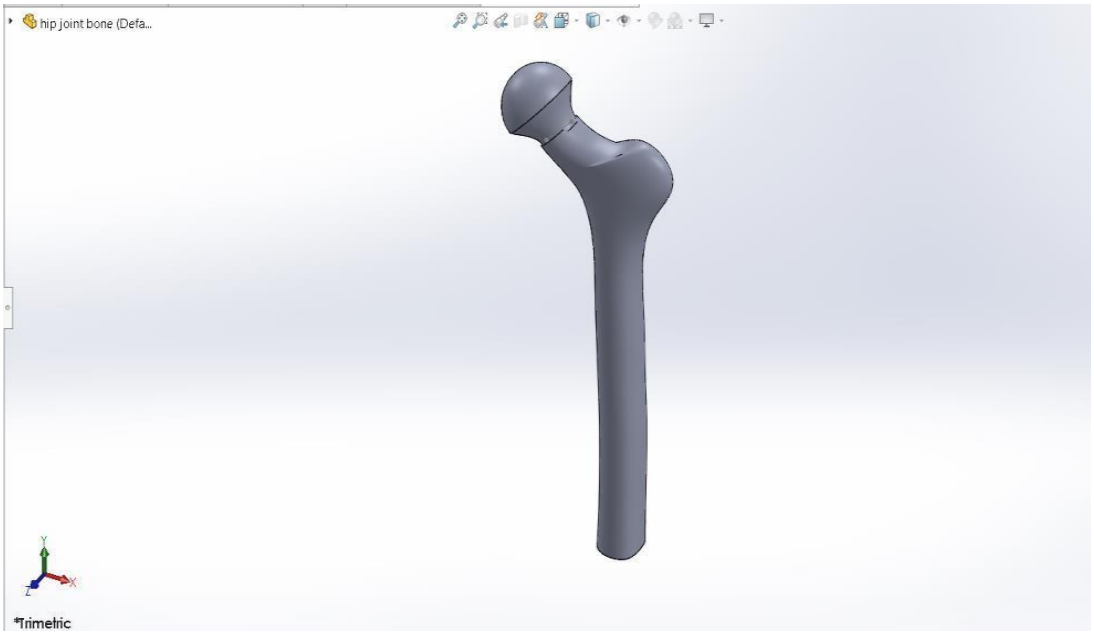


Figure 6: Model of Femur Bone

Figure 7 shows that these components of Fracture System were assembled by Assembly Part of the Software.



Figure 7: Assembly of Plate Screw with Femur Bone

Finite Element Modelling:
Geometry: For analysis the model Polysolid(x_t) file was imported to COMSOL Multiphysics, the imported geometry was shown in figure 8.

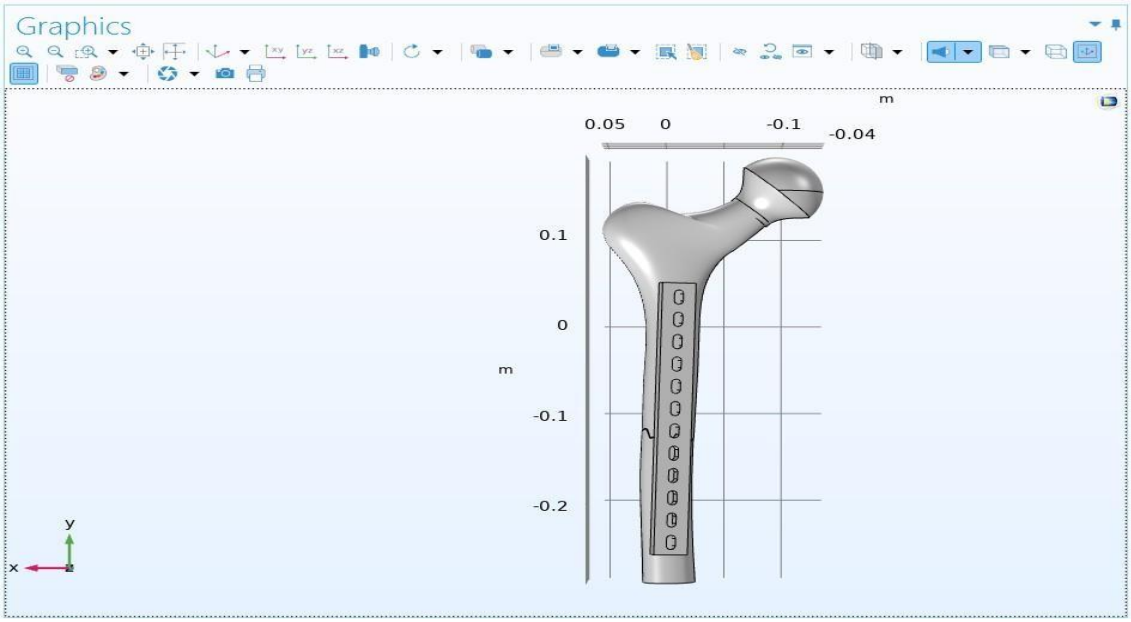


Figure 8: Imported Model

Material: Femur Bone was assigned with Cortical bone material, [4] [19].
Because the outer part of the Femur is cortical bone, which is dense, strong, and load-bearing .Most of the mechanical stress in long bones is carried by this cortical layer. Table 1 shows the material properties used in Comsol. Femur Plate was assigned with Titanium alloy because it is commonly used in orthopedic implants. Titanium alloy has high strength, Excellent, biocompatibility, High Corrosion resistance and good Fatigue resistance for supporting the fractured bone.

Table 1: Material Properties used in Comsol

Material properties		
Property	**Bone**	**Titanium Alloy**
Density	1750	4500
Elastic Modulus	16.7	120
Poisson's Ratio	0.3	0.32

Physics: This section was used to apply load to the Femur bone to replicate the conditions during walking [2] [20].The superior articular surface of the femoral head, the anatomical area where the hip joint typically transfers body-weight stresses to the femur while standing and walking, was subjected to a compressive load of 3000 N. This location was chosen as the most physiologically correct point for load application in finite element models because, during walking, the joint response force travels straight through the femoral head and into the shaft. Consequently, the application of a 3000 N load in the downward (-Z) direction on the femoral head gives a realistic distribution of stress and strain within the bone and fixation system and accurately simulates the compressive pressures that the femur experiences during normal walking activities.

Solid Mechanics was selected to all domains of model, assign 3000 edge loads to femur head and set remaining part as fixed constraint.

Mesh: A fine mesh was applied to entire region of bone plate system to accurately capture stress concentration strain and displacement specially around the Screw holes plates edges and femoral head as shown in figure 9. Using a fine mesh ensures that stress region are precisely analyzed allowing the realistic Study of implant stability and potential screw failure, while maintaining manageable simulator solver performance.

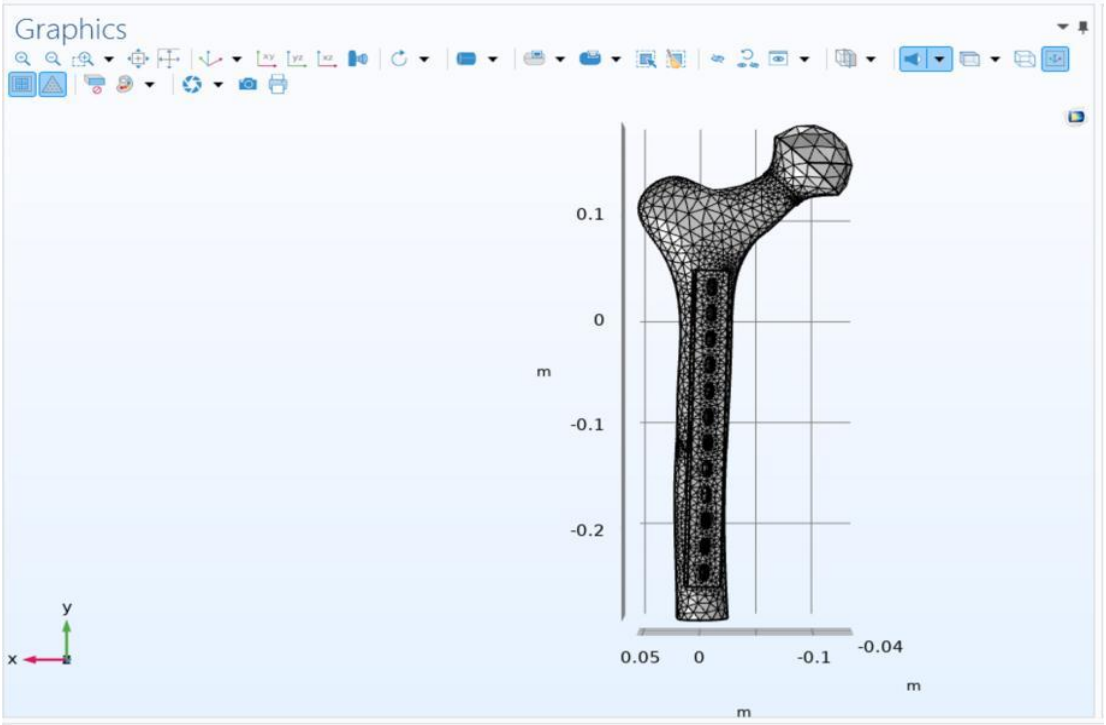


Figure 9: Mesh

Study: For the analysis a stationary study was selected and constant load was applied to the femur. The simulation was then run to compute the final stress, strain and Displacement provide insight study of the mechanical behavior of the bone plate system under loading conditions.

Results and Discussion: This research study used finite element analysis to evaluate the biomechanical behavior of femur and fixation plate. The von mises stress analysis showed that most of the bone experienced low stress while the largest stress concentration occurred near the plate and the screw plate region. These areas carry the majority of load transferred through the implant. The strain was highest in the regions with concentrated stress, specifically around the fracture site and screw holes as shown. This shows that areas bearing the more load also experience greater deformation. The displacement analysis revealed minimal overall movement indicating that the plate and screw effectively stabilized the femur. This ensures that the chosen design of plate screw provides best mechanical support. Proper Planning of Screw Placement ensures more uniform stress distribution, reduced strain and improved implant stability. The figure 10 shows the stress analysis of bone and bone plate. In this model the surface distribution of Von Mises stress was displayed which shows the amount of stress distributed throughout the material when load is applied and how the material close to failure. The colour scale on the right side represents the magnitude of stress while blue indicates low stress and red indicates high stress, as the most of the structure is blue which suggests that under applied load the stress is relatively low and the structure behaves in a stable manner. Figure 11 shows the displacement analysis of bone and bone plate. The upper region of the bone shows higher displacement while the bone plate and lower regions appear blue indicating small movement

hence the distribution suggests that the plate provides structural support successfully. The figure 12 shows the strain analysis of bone and bone plate obtained from simulation. In the results most part of bone and bone plate appears in blue, indicating the strain levels are very low under the applied loads. This suggests that the bone plate system experiences minimal deformation and maintains structural stability.

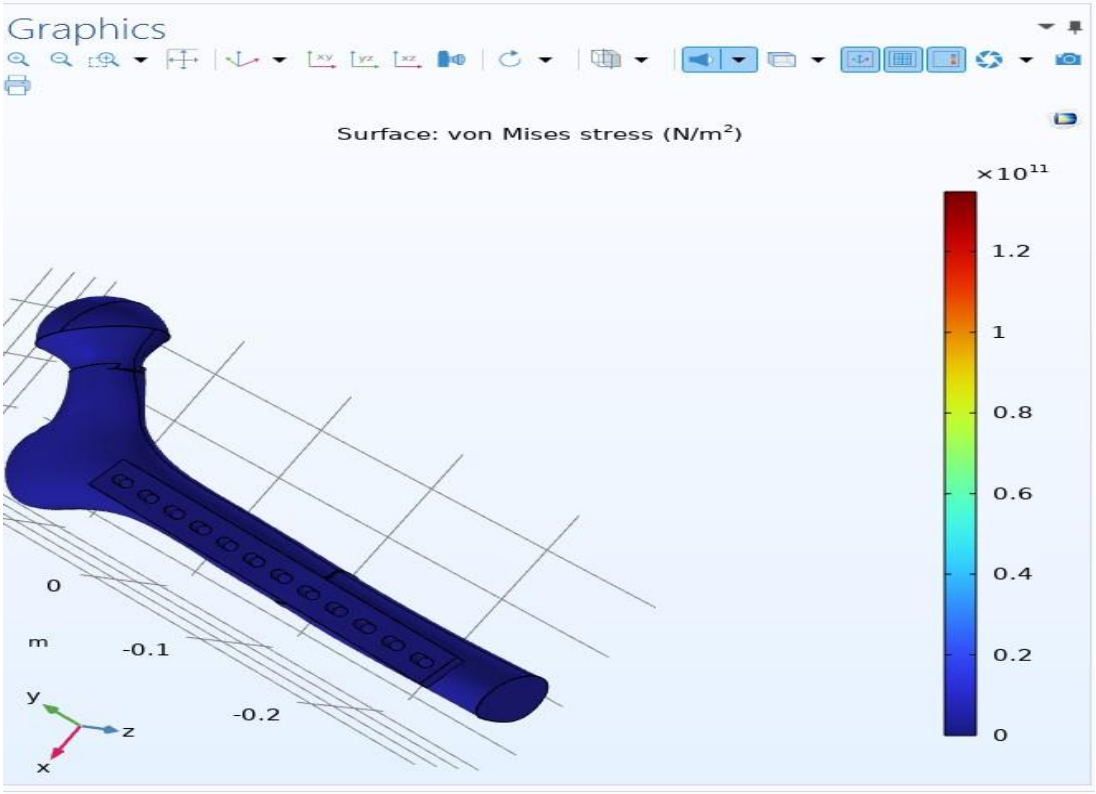


Figure 10 Stress analysis of Bone and Bone Plate

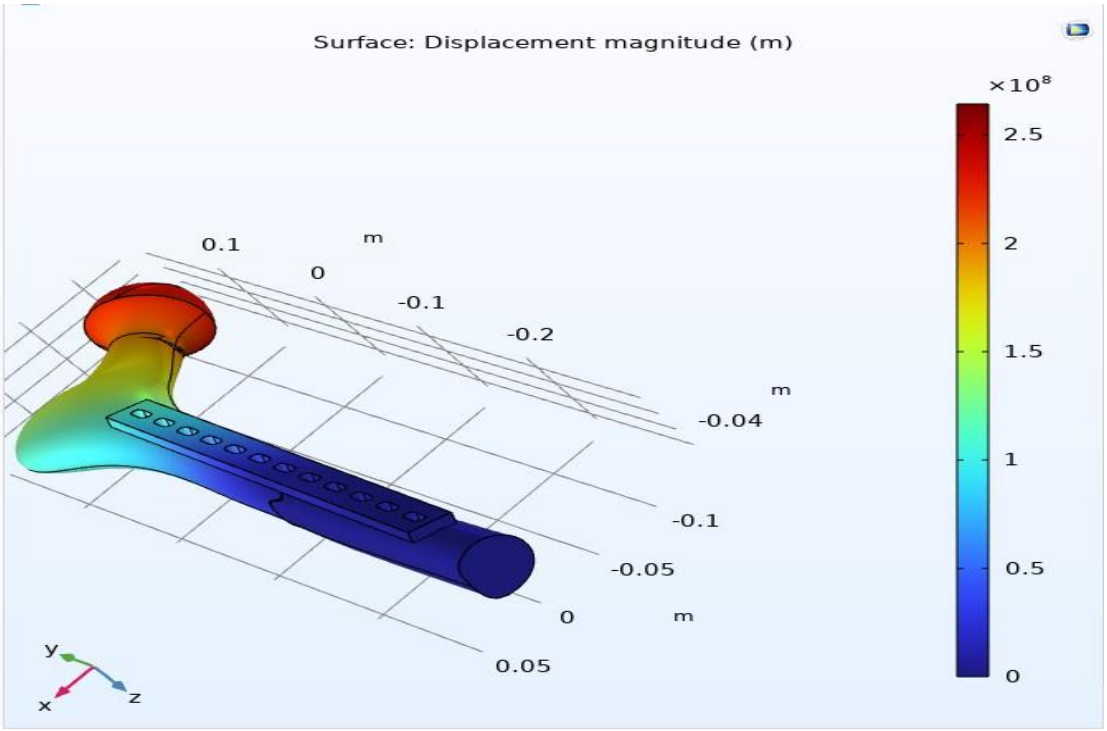


Figure 11: Displacement analysis of Bone and Bone Plate

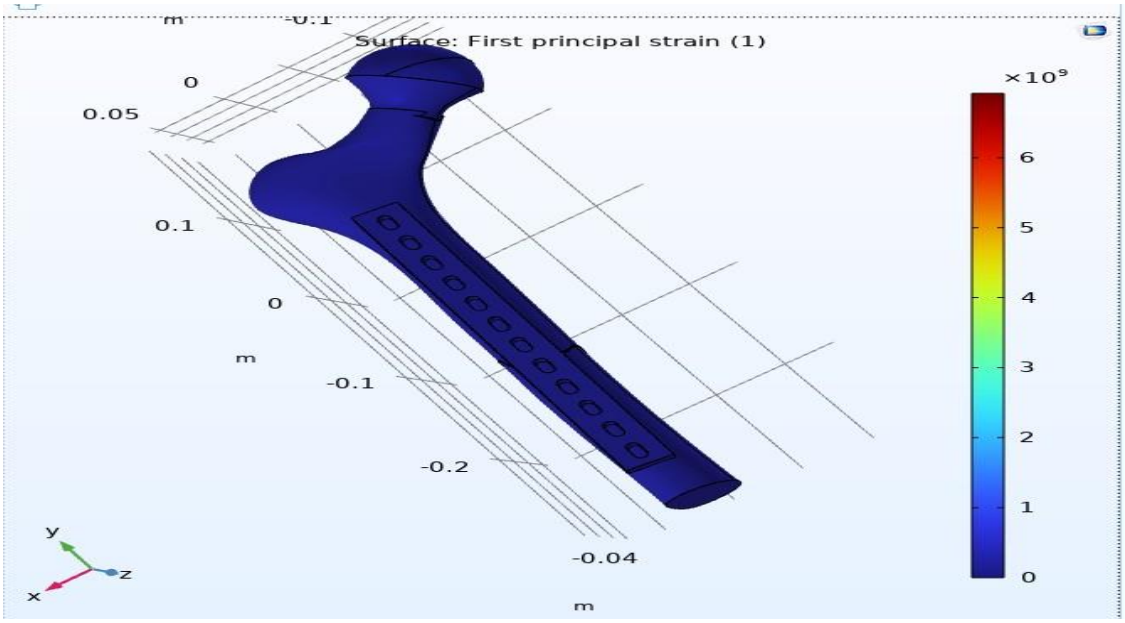


Figure 12: Strain analysis of Bone and Bone Plate

Conclusion: The finite element analysis of femur fracture fixation plates showed that stress and strain are concentrated at the fracture site and around the screw bone region, while overall displacement remains low indicating stable fixation. Among the materials titanium alloy demonstrated superior mechanical performance, reducing stress shielding and providing sufficient strength for safe bone support. These findings concluded that careful selection of implant material, proper design of fixation plates and FEM presurgical analysis are essential to optimize stress distribution maintain implant stability and promote effective fracture healing. In order to further enhance implant design and clinical results, future research

should investigate alternate implant materials, create patient-specific FEM models, and use dynamic simulations to simulate changing loads during everyday activities.

REFERENCES:

S. Solan and R. Kulkarni, "Estimation of total length of femur from its fragments in south indian population," *Journal of Clinical and Diagnostic Research*, vol. 7, no. 10, pp. 2111–2115, Oct. 2013, doi: 10.7860/JCDR/2013/6275.3465.

J. Nourisa, A. Baseri, L. Sudak, and G. Rouhi, "The Effects of Bone Screw Configurations on the Interfragmentary Movement in a Long Bone Fixed by a Limited Contact Locking Compression Plate," 2015, doi: 10.4236/jbise.2015.

S. V Late and H. Keche, "Morphometric Study of Proximal End of the Fully Ossified Human Femur:
A Cross-Sectional Study," *Cureus*, Sep. 2022, doi: 10.7759/cureus.29188.

S. Das and S. K. Sarangi, "Finite Element Analysis of Femur Fracture Fixation Plates," 2014.
[Online]. Available: http://www.krishisanskriti.org/ijbab.html

S. Das and S. K. Sarangi, "Finite Element Analysis of Femur Fracture Fixation Plates," 2014.
[Online]. Available: http://www.krishisanskriti.org/ijbab.html

N. Kaku, T. Hosoyama, Y. Shibuta, M. Kimura, and H. Tsumura, "Influence of femoral bowing on stress distribution of the proximal femur: a three-dimensional finite element analysis," *J Orthop Surg Res*, vol. 18, no. 1, Dec. 2023, doi: 10.1186/s13018-023-03559-1.

M. Ceddia, G. Solarino, M. Tucci, L. Lamberti, and B. Trentadue, "Stress Analysis of Tibial Bone Using Three Different Materials for Bone Fixation Plates," *Journal of Composites Science*, vol. 8, no. 9, Sep. 2024, doi: 10.3390/jcs8090334.

S. M. Kennedy, V. Arunachalam, and A. Kannan, "A novel carbon-flax bioepoxy hybrid composite bone plate with enhanced bio-mechanical performance," *Materwiss Werksttech*, vol. 55, no. 4, pp. 508–517, Apr. 2024, doi: 10.1002/mawe.202300230.

S. A. Naghavi *et al.*, "A novel hybrid design and modelling of a customised graded Ti-6Al-4V porous hip implant to reduce stress-shielding: An experimental and numerical analysis," *Front Bioeng Biotechnol*, vol. 11, Jan. 2023, doi: 10.3389/fbioe.2023.1092361.

M. Palka, P. Miszczyk, M. Jurewicz, and R. Perz, "Finite element method analysis of bone stress for variants of locking plate placement," *Heliyon*, vol. 10, no. 8, Apr. 2024, doi:
10.1016/j.heliyon.2024.e26840.

G. E. Henderson, E. H. Conard, and J. L. Palladino, "Finite Element Modeling of Human Femur Diaphysis."

I. V. Antoniac *et al.*, "Failure analysis of a humeral shaft locking compression plate-surface investigation and simulation by finite element method," *Materials*, vol. 12, no. 7, 2019, doi:
10.3390/ma12071128.

A. A. Al-Tamimi, C. Quental, J. Folgado, C. Peach, and P. Bartolo, "Stress analysis in a bone fracture fixed with topology-optimised plates," *Biomech Model Mechanobiol*, vol. 19, no. 2, pp. 693–699, Apr. 2020, doi: 10.1007/s10237-019-01240-3.

J. Kurniawan, S. Y. Lin, and W. T. Wang, "Development and Finite Element Analysis of a Novel Bent Bone Plate," *Applied Sciences (Switzerland)*, vol. 12, no. 21, Nov. 2022, doi:
10.3390/app122110900.

V. Verma and K. Pal, "A Finite Element Investigation on the Design of Mechanically Compatible Functionally Graded Orthopaedic Plate for Diaphyseal Tibia Transverse Fracture," *Composites Part C: Open Access*, vol. 7, Mar. 2022, doi: 10.1016/j.jcomc.2022.100228.

F. Xiang *et al.*, "Finite element analysis of a novel anatomical locking plate for scapular neck fracture," *J Orthop Surg Res*, vol. 18, no. 1, Dec. 2023, doi: 10.1186/s13018-023-03743-3.

P. S. R. S. Maharaj, R. Maheswaran, and A. Vasanthanathan, "Numerical analysis of fractured femur bone with prosthetic bone plates," in *Procedia Engineering*, Elsevier Ltd, 2013, pp. 1242– 1251. doi: 10.1016/j.proeng.2013.09.204.

G. Narayanaswamy and B. A. Thomas, "Finite element analysis of tibia bone model," *Int J Eng Adv Technol*, vol. 9, no. 1, pp. 4607–4610, Oct. 2019, doi: 10.35940/ijeat.F9305.109119.

N. Tan and R. J. van Arkel, “Topology optimisation for compliant hip implant design and reduced strain shielding,” *Materials*, vol. 14, no. 23, Dec. 2021, doi: 10.3390/ma14237184.

M. Syawal, A. Sulong, A. Syahrom, and Z. Zakaria, “STUDY OF LOCKING COMPRESSION PLATE THROUGH BIODEGRADABLE IMPLANT,” 2022.