



Comparative Biomechanical Study of Healthy and Fractured Femur Bone with Two Implants Using FEA Analysis

Aditya Devhare Dept. of Mechanical Engineering Tulsiramji Gaikwad Patil College of Engineering & Technology Nagpur, India adityadevhare05@gmail.com	Ayush Rokde Dept. of Mechanical Engineering Tulsiramji Gaikwad Patil College of Engineering & Technology Nagpur, India ayushrokde3@gmail.com	Vikki Mate Dept. of Mechanical Engineering Tulsiramji Gaikwad Patil College of Engineering & Technology Nagpur, India Vikkymate420@gmail.com
Saharsh Sawaitul Dept. of Mechanical Engineering Tulsiramji Gaikwad Patil College of Engineering & Technology Nagpur, India Saharshsawaitul2@gmail.com	Ayush Patrange Dept. of Mechanical Engineering Tulsiramji Gaikwad Patil College of Engineering & Technology Nagpur, India ayushpatrange677@gmail.com	Guided by :- Prof. Shrutika Nitnaware Tulsiramji Gaikwad Patil College of Engineering & Technology Nagpur, India assodean.tnp.diploma@tgpcet.com

How to Cite this Article:

Devhare, A., Rokde, A., Mate, V., Sawaitul, S. & Patrange, A. (2026). Comparative Biomechanical Study of Healthy and Fractured Femur Bone with Two Implants Using FEA Analysis. International Journal of Creative and Open Research in Engineering and Management, <i>10</i>(10), 1-9. <https://doi.org/10.55041/ijcope.v2i10.003>

License:

This article is published under the terms of the Creative Commons Attribution 4.0 International License (CC BY 4.0), which permits unrestricted use, distribution, and reproduction in any medium, provided the original author(s) and the source are credited.

© The Author(s). Published by International Journal of Creative and Open Research in Engineering and Management.



<https://doi.org/10.55041/ijcope.v2i10.003>

Abstract—This study presents a comparative biomechanical finite element analysis (FEA) of a human femur bone under three configurations: (1) a healthy, intact femur, (2) a fractured femur stabilized with an intramedullary interlocking nail (Implant 1), and (3) a fractured femur stabilized with a bone fixation plate (Implant 2). A three-dimensional femur geometry, reconstructed from scan data, was modelled in SolidWorks 2026 and imported into ANSYS Workbench (Student 2026 R1) for static structural analysis. The bone was assigned isotropic linear-elastic material properties (Nylon 66) as a simplified surrogate for cortical bone, while the implant components were modelled using Titanium alloy Ti-6Al-7Nb. A static axial load of 700 N, representative of physiological body-weight loading through the hip joint, was applied at the femoral head with the distal (knee) end fully constrained. Equivalent (von-Mises) stress, equivalent elastic strain, and total deformation were evaluated for each configuration. Results show that the healthy femur exhibited a maximum total deformation of 10.195 mm and a maximum equivalent stress of 13.91 MPa, whereas the fractured femur stabilized with the intramedullary nail exhibited a maximum deformation of 1.145 mm, with a maximum equivalent stress of 25.79 MPa. The fractured femur stabilized with the bone plate exhibited a maximum deformation of 2.348 mm, a maximum equivalent elastic

strain of 1.2414×10^{-3} mm/mm, and the largest maximum equivalent stress of 113.31 MPa. These results quantify the different load-transfer responses produced by the two fixation configurations and identify regions of elevated stress and deformation relevant to implant design and placement.

Index Terms—Femur fracture, Finite Element Analysis, Intramedullary nail, Bone plate, Biomechanics, Ti-6Al-7Nb, ANSYS, Orthopedic implant

I. INTRODUCTION

The femur is the longest and strongest bone in the human body, and it bears substantial loads during standing, walking, and other daily activities. On average, femur length accounts for approximately 26.74% of a person's height, a ratio consistent across sexes and most ethnic groups [1]. Femoral fractures, particularly in the shaft (diaphysis) and neck regions, are common consequences of high-impact trauma or, in elderly patients, low-energy falls associated with osteoporosis. Restoring the mechanical integrity of a fractured femur requires internal fixation devices — most commonly intramedullary nails or fixation plates — that transfer load across the fracture site while the bone heals.

Finite Element Analysis (FEA) has become an indispensable tool in orthopedic biomechanics, allowing researchers and implant designers to predict stress and strain distributions, identify regions prone to failure, and compare the mechanical performance of different fixation strategies without the cost and ethical constraints of cadaveric or in-vivo testing [2]–[4]. FEA models of the femur are typically reconstructed from CT or MRI scan data, converted into a 3D solid, meshed, and subjected to physiological boundary conditions representative of standing, walking, or single-leg stance [1], [5].



This study uses FEA to quantitatively compare the mechanical response of (a) a healthy femur, (b) a fractured femur fixed with an intramedullary nail, and (c) a fractured femur fixed with a bone plate, under a representative physiological load. The objective of this work is to evaluate and compare the equivalent stress, equivalent elastic strain, and total deformation across these three configurations, thereby providing insight into how each fixation method redistributes load relative to the intact bone, and to situate these findings within the broader body of published femur FEA literature.

Femoral shaft and neck fractures represent a significant clinical and economic burden worldwide, with proximal femoral fractures alone accounting for approximately 5–10% of all fragility fractures in older adults [6]. The choice between an intramedullary nail and a bone plate for a given fracture pattern has direct clinical consequences: intramedullary devices are generally load-sharing and centrally placed along the bone's mechanical axis, whereas plates are load-bearing and act eccentrically along the lateral cortex, which can result in different stress distributions, displacement behaviour, and long-term risk of implant loosening or secondary fracture. A rigorous, quantitative comparison of these two fixation strategies — of the kind undertaken in this study — is therefore of direct relevance to implant selection in orthopedic trauma surgery, in addition to its value as an engineering design exercise.

II. LITERATURE REVIEW

A substantial body of published work has applied FEA to the femur, both to characterize the intact bone's mechanical behaviour and to evaluate fixation devices and alternative implant materials. This section summarizes representative studies that directly inform the modeling assumptions, material properties, and boundary conditions used in the present work.

Aftab et al. [1] generated a 3D femur model from MRI data and used ANSYS to evaluate the von-Mises stress induced in the natural femur bone versus candidate alternative composite materials (PMMA, Al_2O_3 , and Nylon 6/6) under standing, walking, jumping, and running loads. They reported a stress of 11.966 MPa for the natural femur under standing load (370 N per leg), rising to 45.915 MPa under running load (1463 N), and concluded that PMMA produced stress values closest to natural bone at comparatively low material cost.

Takahashi et al. [4] used FEA together with axial loading tests to compare single-plate and double-plate fixation,

using reversed locking compression distal femoral plates, for Vancouver B1 periprosthetic femoral fractures under a 1,500 N axial load. In the double-plate construct the maximum von Mises stress in the lateral plate was 268 MPa, whereas in single plating the maximum stress occurred at the femoral defect level (1,303 MPa); principal strains fell from 0.63% to 0.058%, a trend that the strain-gauge loading tests supported.

In a review of FEA for fracture fixation, Lewis et al. [2] reported a pragmatic consensus on how geometry, meshing, material properties, interactions, loads, and boundary conditions are formulated, noted that studies most often report implant stress, bone strain around screws, or interfragmentary displacement, and cautioned that most published models are not rigorously validated — a caveat that applies equally to the preliminary model developed here.

Bazyar et al. [7] reviewed the range of isotropic, orthotropic, and anisotropic material property sets reported in the femur FEA literature, noting that the choice of an appropriate density–elasticity relationship for cortical bone significantly affects predicted stress and strain magnitudes, and that simplified isotropic assumptions — while common — introduce quantifiable deviation from anisotropic, subject-specific models.

Kwak et al. [3] simulated 96 finite element models of transverse subtrochanteric fractures fixed with short cephalomedullary nails, varying the fracture level, the fracture gap, and the distal locking-screw configuration in normal and osteoporotic bone. Two distal locking screws gave a wider safe range than one at fracture gaps of 3 mm or less, and peak von Mises stresses rose and the safe range narrowed as the fracture gap increased.

Most recently, Yang et al. [6] compared the Dynamic Hip Screw (DHS), Proximal Femoral Nail Anti-Rotation (PFNA), and InterTAN devices for proximal femoral basicervical fractures using a validated intact-femur FEA model under a 700 N single-leg-stance load — the same loading magnitude adopted in the present study — reporting that cephalomedullary nails provided superior displacement control over the DHS in unstable fracture patterns, while all recorded implant stresses remained below the fatigue limit of medical titanium (850–900 MPa).

Regarding implant materials and stiffness, Ceddia et al. [8] compared Ti-6Al-4V and carbon-fibre-reinforced PEEK intramedullary nails for proximal femoral fractures in young and osteoporotic bone under mid-stance gait



loading that included age-adjusted muscle forces. The titanium nail produced lower bone stresses but marked proximal stress shielding, particularly in osteoporotic bone, whereas the CFR-PEEK nail produced higher and more uniform diaphyseal stresses. In a review of implant stiffness and fracture healing, Mori et al. [9] noted that excessively rigid metallic constructs can lead to complications, that titanium-alloy locking plates have been shown to be biomechanically superior to stainless steel, and that lower-modulus alloys such as TiNbSn are being developed to address stress shielding. Naghavi et al. [10] likewise combined experiments with FEA to show that a graded porous Ti-6Al-4V hip implant reduced stress shielding by about 70% and bone-mass loss by about 60% relative to a fully solid implant.

A number of recent FEA studies have compared newer nail designs for proximal femoral fractures. Wang et al. [11] evaluated a newly designed intramedullary nail against PFNA and InterTAN for intertrochanteric fractures, and Yang et al. [12] compared a proximal femoral bionic nail with three conventional cephalomedullary nails for reverse-obliquity intertrochanteric fractures, verifying their femur model against non-contact optical strain measurements on a femoral specimen; they identified the A3.3 pattern as the most unstable. Zhu et al. [13] examined how different reduction patterns alter stress distribution in intertrochanteric fractures fixed with an intramedullary nail.

For femoral neck fractures, Ding et al. [14] reported that a proximal femoral bionic nail reduced peak von Mises and principal stresses and stress concentration relative to dynamic hip screws and cannulated screws under axial loading. Huang et al. [15] and Zhu et al. [16] used a 2100 N axial load in FEA comparisons of modified intramedullary systems and of cannulated-screw, biplanar double-support-screw, and dynamic-hip-screw fixation, respectively; in the latter, the biplanar double-support screw gave smaller femoral displacement than the other two devices at every reduction quality. Zhang et al. [17] showed that changing the bolt length, bolt position, and axis-bolt angle of the femoral neck system alters construct stability in Pauwels type III fractures.

Collectively, this literature establishes (i) isotropic linear-elastic material modeling as a widely used simplification for preliminary femur FEA studies [2], [7], (ii) titanium alloys as a widely used metallic implant material, reported to be biomechanically favorable relative to stainless steel for locking plates [9] while raising stiffness-related

concerns such as stress shielding [8], [10], and (iii) axial loading at the femoral head with the distal condyle constrained — 700 N in [6] and 1500–2100 N in [4], [15], [16] — as a common single-leg-stance boundary-condition simplification [5], all of which directly informed the methodology adopted below.

Beyond the studies summarized above, the broader femur FEA literature has explored several related directions relevant to the present work. Studies on mini dental and orthopedic screw implants have shown that thread pitch and screw geometry strongly influence peri-implant stress concentration, with higher thread pitch generally producing higher peak stress at the bone-implant interface — a consideration directly applicable to the M7 locking-screw design used for Implant 1 in this study. Similarly, density-elasticity relationships derived from CT Hounsfield-unit data have been shown to introduce mean strain-prediction errors on the order of 10–25% relative to simplified isotropic models, reinforcing the view that the present Nylon 66 surrogate material should be interpreted as a comparative baseline rather than a patient-specific prediction.

A further recurring theme across the reviewed studies is the trade-off between implant stiffness and stress shielding: while a stiffer implant (such as stainless steel) reduces bone displacement more aggressively, it also carries a larger fraction of the applied load away from the surrounding bone, which over time can reduce local bone density due to Wolff's law adaptive remodeling [8]–[10]. Titanium alloys, with an elastic modulus roughly 2–3 times lower than stainless steel, are reported to strike a more favorable balance between mechanical stability and physiological load-sharing [8], [9], which is the primary rationale for the material selection adopted in Section III-B.

III. MATERIALS AND METHODS

A. Geometric Modeling

A three-dimensional femur model was obtained as a triangulated surface mesh and imported into SolidWorks 2026 for dimensional extraction and CAD reconstruction. Key anatomical dimensions were measured directly on the model: an overall bone length of 471.17 mm and a medullary canal diameter of 7 mm at its narrowest (isthmus) section, consistent with the proportions reported for adult femora in [1]. These dimensions were subsequently used to proportion the two implant designs so that each would fit within the intramedullary canal and along the shaft.



B. Material Properties

The femur bone was modeled as a homogeneous, isotropic, linearly elastic solid using Nylon 66 properties as a simplified surrogate for cortical bone tissue, consistent with the isotropic simplification widely adopted in the reviewed literature [1], [2], [6], [7]. Both implant configurations were modeled using Titanium alloy Ti-6Al-7Nb, a biocompatible titanium alloy of the kind widely used in orthopedic trauma implants, where titanium constructs have been reported to be biomechanically favorable relative to stainless steel [8], [9]. The mechanical properties assigned to each material in this study are summarized in Table I.

TABLE I
MATERIAL PROPERTIES USED IN FEA

Property	Nylon 66 (Bone)	Ti-6Al-7Nb (Implant)
Young's Modulus (MPa)	3000	105000
Poisson's Ratio	0.4	0.3
Density (kg/m ³)	1140	4520
Bulk Modulus (MPa)	5000	87500
Shear Modulus (MPa)	1071.4	40385

TABLE I. Isotropic linear-elastic material properties assigned to the bone and implant components.

For comparison, Table II summarizes representative material properties reported for candidate bone and implant materials in the literature [1], illustrating the range of Young's modulus and strength values considered by other researchers when selecting surrogate or alternative femur materials.

TABLE II
REPRESENTATIVE MATERIAL PROPERTIES REPORTED IN LITERATURE [1]

Material	Young's Modulus (GPa)	Density (g/cc)	Poisson's Ratio	Ult. Tensile Strength (MPa)
Human/Natural bone	18	1.8	0.33	150
PMMA	220	1.18	0.2	63
Al ₂ O ₃ (Alumina)	240	8.5	0.31	250
Nylon 6/6	3.0	1.14–3.72*	0.21–0.4*	65–195

TABLE II. *Reported Nylon 6/6 property ranges vary between sources; the lower-bound values (used by [1]) were adopted for the present bone surrogate model.

C. Implant Configurations

Implant 1 (Intramedullary Nail): A cannulated straight titanium nail, approximately 360 mm in length and 6 mm in diameter, was designed to be inserted through the medullary canal, spanning the fracture site, and secured with M7 diameter locking screws through the proximal and distal shaft.

Implant 2 (Bone Plate): A fixation plate positioned along the lateral shaft surface, secured to the bone through the existing screw holes, providing extramedullary fixation across the fracture site, following the plate-and-screw fixation approach reported in [4]. The Implant 2 assembly (plate and screws) was imported into ANSYS from the SpaceClaim file "Plate Final.scdocx"; the plate body has a bounding length of 198.48 mm.

D. Meshing

The healthy femur model was discretized into 12,276 tetrahedral elements with 21,542 nodes. The fractured femur with Implant 1 assembly (comprising the bone fragments, nail, and locking screws) was meshed into 72,615 elements with 123,039 nodes, with bonded contact defined at the bone–screw and bone–fracture interfaces. The fractured femur with Implant 2 assembly (comprising the bone fragments, plate, and screws; 12 bodies in total) was meshed with patch-independent tetrahedral elements into 79,498 elements with 126,112 nodes, with 12 bonded contact regions. Default program-controlled adaptive sizing was used in ANSYS Workbench 2026 R1 for all three models — a mesh density broadly comparable to that used in [1] (29,162 elements) and substantially finer than the coarse validation meshes used in early femur FEA studies.

E. Boundary Conditions and Loading

A fixed support was applied at the distal (knee) end of the femur in all cases, restraining all degrees of freedom to represent the load transfer to the tibia through the knee joint. Although a fixed knee is a commonly used simplification, it is non-physiological and can produce unrealistic femoral biomechanics [5]. A static force of 700 N — representative of body weight transmitted through the hip joint during single-leg stance, and matching the loading condition adopted by Yang et al. [6] for their intact and fractured femur models — was applied at the proximal (femoral head) end in each configuration. Static Structural analysis was performed in ANSYS Mechanical (Solver Target: Mechanical APDL), and Equivalent (von-Mises) Stress, Equivalent Elastic Strain, and Total Deformation were computed as the primary result quantities. In the Implant 2 model, the 700 N force was applied in the positive global X direction, as in the healthy-femur model, with the knee end constrained through a zero-displacement condition on a nodal named selection (3131 nodes).

IV. RESULTS

A. Case 1: Healthy Femur Bone

Under the 700 N axial load, the healthy femur bone exhibited a maximum equivalent stress of 13.91 MPa (average 0.920 MPa), a maximum equivalent elastic strain of 4.638×10^{-3} mm/mm (average 3.205×10^{-4} mm/mm), and a maximum total deformation of 10.195 mm (average 4.092 mm), occurring at the loaded (proximal) end of the bone. These results establish the baseline mechanical response of the unfixed, intact bone against which the two fixed configurations are compared, and are of the same order of magnitude as the standing-condition stress (11.966 MPa) reported for the natural femur by Aftab et al. [1]. The boundary conditions and simulation results are illustrated in Fig. 1–Fig. 4.

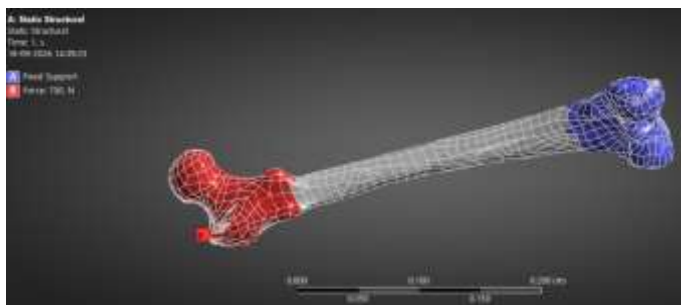


Fig. 1. Boundary conditions for the healthy femur: fixed support (A) at the distal end and 700 N force (B) at the proximal end.

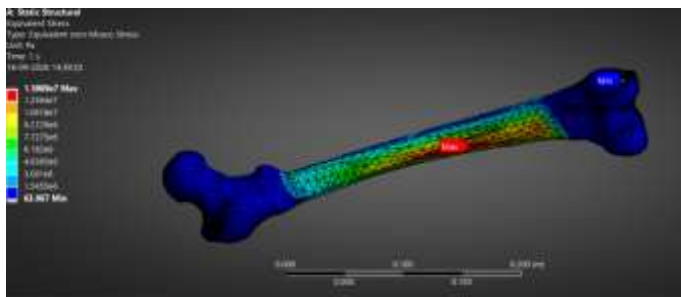


Fig. 2. Equivalent (von-Mises) stress distribution in the healthy femur (Max. 13.91 MPa).

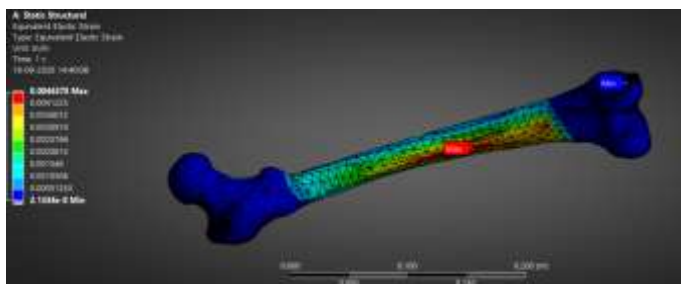


Fig. 3. Equivalent elastic strain distribution in the healthy femur (Max. 4.638×10^{-3} mm/mm).

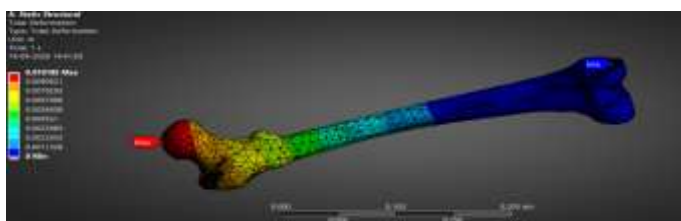


Fig. 4. Total deformation distribution in the healthy femur (Max. 10.195 mm).

B. Case 2: Fractured Femur with Implant 1 (Intramedullary Nail)

With the fractured femur stabilized by the Ti-6Al-7Nb intramedullary nail and locking screws, the maximum equivalent stress rose to 25.79 MPa (average 0.269 MPa), occurring in the bone material adjacent to the screw fixation points rather than in the implant itself, indicating that the bone — being far more compliant than the titanium nail — concentrates stress near the load-transfer interfaces. This pattern is consistent with the screw-junction stress concentration reported for cephalomedullary nails by Yang et al. [6]. The maximum equivalent elastic strain was 2.091×10^{-3} mm/mm (average 5.77×10^{-5} mm/mm). Most notably, the maximum total deformation was reduced to 1.145 mm (average 0.556 mm) — an approximately 89% reduction relative to the healthy, unfixed bone — demonstrating the substantial stabilizing effect of the intramedullary fixation in restricting displacement across the fracture site. This magnitude of displacement is comparable to the 0.95–1.38 mm range reported for simple basicervical fractures with cephalomedullary nails under an identical 700 N load in [6]. The boundary conditions and simulation results are illustrated in Fig. 5–Fig. 8.



Fig. 5. Boundary conditions for the fractured femur with Implant 1: fixed support (A) at the distal end and 700 N force (B) at the proximal end.

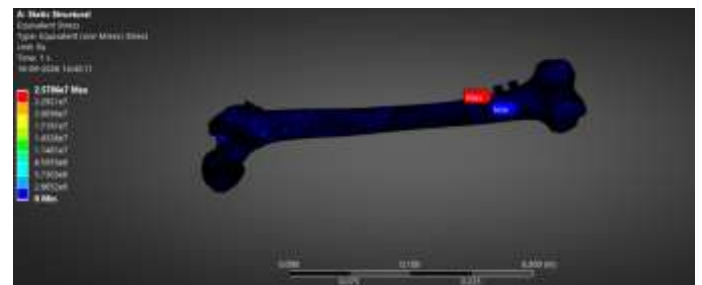


Fig. 6. Equivalent (von-Mises) stress distribution with Implant 1 (Max. 25.79 MPa).



Fig. 7. Equivalent elastic strain distribution with Implant 1 (Max. 2.091×10^{-3} mm/mm).

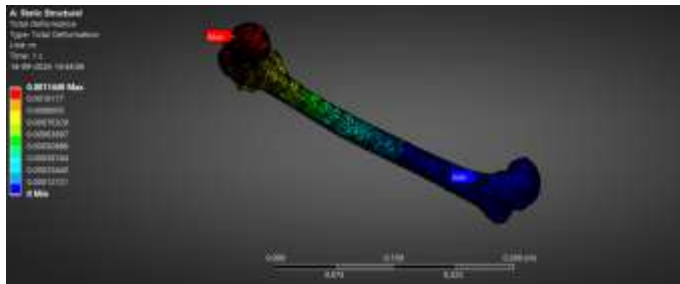


Fig. 8. Total deformation distribution with Implant 1 (Max. 1.145 mm).

C. Case 3: Fractured Femur with Implant 2 (Bone Plate)

With the fractured femur stabilized by the Ti-6Al-7Nb lateral bone plate and screws, the maximum equivalent stress reached 113.31 MPa (average 0.71183 MPa), which was the highest maximum stress among the three analyzed cases. The maximum equivalent elastic strain was 1.2414×10^{-3} mm/mm (average 6.6364×10^{-5} mm/mm), while the maximum total deformation was 2.348 mm (average 1.14 mm). Relative to the healthy femur, the plate configuration reduced maximum deformation by approximately 77.0% and maximum equivalent elastic strain by approximately 73.2%, while the maximum equivalent stress increased by approximately 714.6%. Compared with the intramedullary-nail case, the plate produced approximately 105.1% greater maximum deformation, 40.6% lower maximum equivalent elastic strain, and 339.4% higher maximum equivalent stress. In the contour plots, the maximum-value markers for stress, strain, and deformation appear in the proximal region near the upper end of the plate. Because the ANSYS results were scoped to all bodies, these maxima are global values and are not attributed to the plate, the screws, or the bone individually. The model and simulation results are illustrated in Fig. 9–Fig. 12.

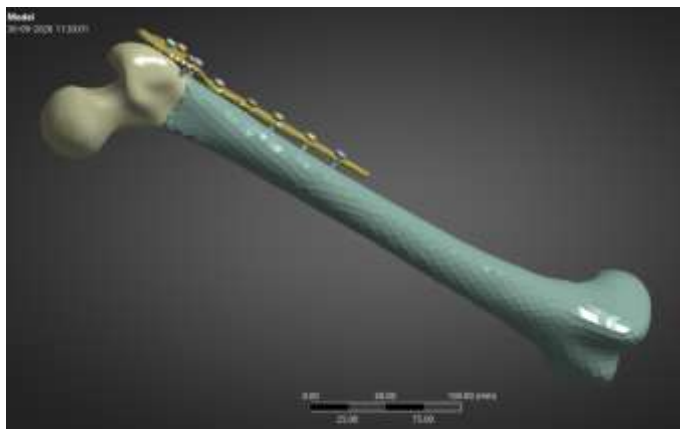


Fig. 9. Assembly model of the fractured femur with Implant 2 (lateral bone plate and screws).

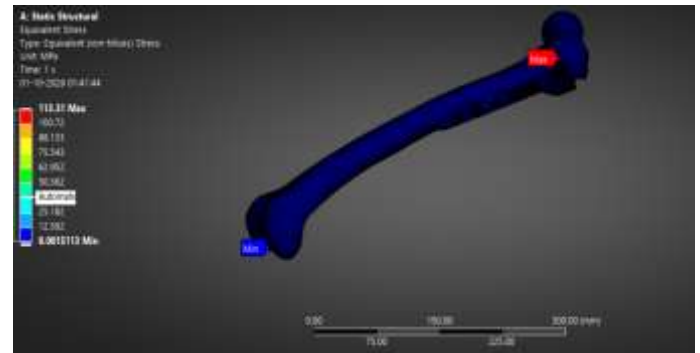


Fig. 10. Equivalent (von-Mises) stress distribution with Implant 2 (Max. 113.31 MPa).

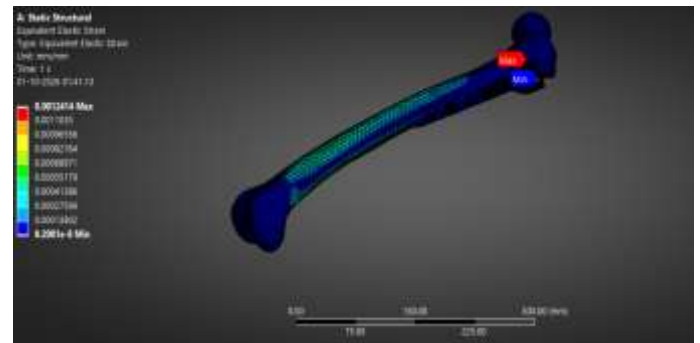


Fig. 11. Equivalent elastic strain distribution with Implant 2 (Max. 1.2414×10^{-3} mm/mm).

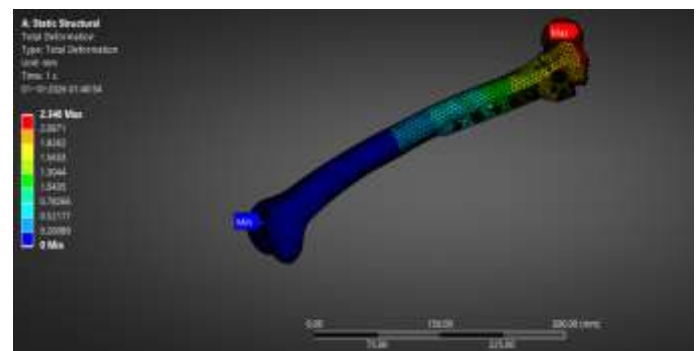


Fig. 12. Total deformation distribution with Implant 2 (Max. 2.348 mm).

D. Comparative Summary

TABLE III
COMPARATIVE FEA RESULTS SUMMARY (700 N LOAD)

Configuration	Max. Eq. Stress (MPa)	Max. Eq. Elastic Strain	Max. Total Deformation (mm)
Healthy Femur	13.91	4.638×10^{-3}	10.195
Fractured + Implant 1 (Nail)	25.79	2.091×10^{-3}	1.145
Fractured + Implant 2 (Plate)	113.31	1.2414×10^{-3}	2.348

TABLE III. Comparison of maximum equivalent stress, equivalent elastic strain, and total deformation across the three simulated configurations.

V. DISCUSSION

The healthy femur, lacking any internal fixation and modeled with compliant surrogate material properties, exhibited the largest total deformation of the three cases, consistent with its lower effective bending stiffness under the applied end-loading condition. Introduction of the



titanium intramedullary nail substantially reduced total deformation (from 10.195 mm to 1.145 mm), confirming that the high-stiffness titanium component ($E = 105$ GPa, roughly 35 times that of the surrogate bone material) carries a disproportionate share of the applied load and restores much of the structural rigidity lost at the fracture site. This is consistent with the general finding across the reviewed literature [4], [6] that a stiff metallic fixation construct restricts displacement and strain across the fracture site.

Interestingly, the maximum equivalent stress increased from 13.91 MPa to 25.79 MPa after fixation. This is attributed to stress concentration in the bone material immediately surrounding the screw holes and fracture interface — regions where load transfer between the stiff implant and the more compliant bone is locally abrupt. This is a well-documented phenomenon in orthopedic biomechanics [3], [4], [6] and underscores the clinical importance of screw placement and hole geometry in minimizing peri-implant stress risers, which can contribute to stress-shielding or secondary fracture. Notably, Yang et al. [6] similarly found that stress concentrated at the screw–device junction rather than diffusely through the bone, and that the inferior screw position reduced both displacement and stress relative to a central position — a refinement not yet incorporated into the present nail design but identified here as a direction for future work.

The bone-plate configuration (Implant 2) produced a maximum equivalent elastic strain of 1.2414×10^{-3} mm/mm, lower than both the healthy femur and the intramedullary-nail case, while its maximum equivalent stress of 113.31 MPa was the highest of the three cases. Its maximum total deformation was 2.348 mm, which was higher than the 1.145 mm obtained for the intramedullary nail but substantially lower than the 10.195 mm of the healthy femur. Relative to the healthy femur, the plate reduced maximum deformation by approximately 77.0% and strain by approximately 73.2%, while stress increased by approximately 714.6%. Relative to the nail configuration, the plate increased maximum deformation by about 105.1%, reduced maximum strain by about 40.6%, and increased maximum stress by about 339.4%. These results indicate different load-transfer responses for the two fixation arrangements rather than a single performance measure being lower for one configuration in every category. Three caveats apply to this comparison. First, the Implant 2 model applies the 700 N force in the positive global X direction, as in the healthy-femur model, whereas the Implant 1 model applies it in the negative global Z direction, so the comparison with Implant 1 is subject to a coordinate-system check. Second, the maxima are global values over all bodies and cannot be assigned to the plate,

the screws, or the bone. Third, all contact regions in the Implant 2 model are bonded and no mesh-convergence study was performed, so the values are not claimed to be mesh-independent.

Compared against the wider literature, the present healthy-femur stress value (13.91 MPa at 700 N) is broadly consistent with the standing-condition stress of 11.966 MPa reported by Aftab et al. [1] at a comparable per-leg load (370 N, roughly half of the present 700 N load, applied to a different bone geometry and boundary condition), and the maximum total deformation of 1.145 mm for the nailed configuration falls within the 0.95–1.38 mm range reported for cephalomedullary nail fixation of simple fractures under an identical 700 N load by Yang et al. [6], lending reasonable external validity to the present model despite its simplified material assumptions.

A key limitation of the present study is the use of isotropic, homogeneous Nylon 66 properties as a surrogate for cortical bone, rather than the anisotropic, heterogeneous properties of real bone tissue (typical cortical bone Young's modulus ≈ 17 – 18 GPa [1], [6], [7]). While this simplification is common in preliminary computational studies and does not affect the qualitative comparison between configurations, quantitative stress and deformation values reported here should be interpreted as relative trends rather than absolute physiological predictions, consistent with the caveats raised by Bazzyar et al. [7] regarding material-property selection in femur FEA. Future work should, where possible, incorporate anisotropic cortical/cancellous bone material models validated against experimental strain measurements or literature data, as demonstrated by Yang et al. [12] and as called for by Lewis et al. [2].

A second limitation is the exclusion of dynamic and cyclic loading effects. The present analysis considers only a single static load case representative of single-leg stance, whereas in-vivo loading during walking, stair-climbing, and stumbling involves cyclic and impact loading that can induce fatigue failure at stress levels well below the static yield strength of the implant material. As noted by Yang et al. [6] and Ceddia et al. [8], who respectively compare implant stresses with the fatigue limit of titanium and identify dynamic loading as the next step beyond static analysis, comprehensive implant evaluation ultimately requires fatigue-life analysis in addition to static structural FEA, particularly for younger, more active patients where the implant may be subjected to millions of loading cycles before bone union is achieved.



Third, muscle forces acting across the hip and knee joints — which in reality partially unload the bone during standing and gait — were omitted from the present boundary conditions, consistent with the simplification adopted in [6]. Aftab et al. [1] noted that incorporating muscle force effects can reduce the induced bone stress by as much as 30% relative to a bone-only loading model, suggesting that the absolute stress values reported here may represent a conservative (upper-bound) estimate relative to true physiological conditions. Studies that derive femoral loading from muscle-inclusive models, such as [5] and [8], show how much the loading and constraint assumptions can matter. In the same vein, the fixed distal support used here is a common but non-physiological constraint: Babil et al. [5] found that conventional constraints, including a fixed knee, produce unrealistic femoral biomechanics compared with a physiological constraint method, which gave femoral head deflections below 1 mm. The absolute displacements reported here, particularly for the compliant Nylon 66 healthy-femur model, should therefore be read as relative indicators only. Despite these simplifications, the relative comparison among the healthy, nail-fixed, and plate-fixed configurations — which is the primary objective of this study — is expected to remain robust, as the same boundary-condition assumptions were applied across all cases.

VI. CONCLUSION

A comparative finite element analysis was performed on a healthy femur, a fractured femur stabilized with an intramedullary interlocking nail (Implant 1), and a fractured femur stabilized with a lateral bone plate (Implant 2), under a representative 700 N physiological load. The nail reduced maximum total deformation from 10.195 mm to 1.145 mm, approximately an 88.8% reduction relative to the healthy case. The plate reduced maximum deformation to 2.348 mm, approximately a 77.0% reduction relative to the healthy case. The maximum equivalent stress was 25.79 MPa for the nail and 113.31 MPa for the plate, compared with 13.91 MPa for the healthy femur. The corresponding maximum equivalent elastic strains were 2.091×10^{-3} mm/mm for the nail and 1.2414×10^{-3} mm/mm for the plate. Thus, the plate case showed the higher peak stress but lower peak strain than the nail case, while the nail case showed lower maximum deformation. These results should be interpreted with the documented differences in load direction, global all-body result scoping, bonded contacts, and absence of mesh-convergence verification.

REFERENCES

- [1] S.G. Aftab, A. Faisal, H. Hussain, B. Sreedhara, N. Ramesh Babu, and B.A. Praveen, "Structural analysis of human femur bone to select an alternative composite material," *Materials Today: Proceedings*, 2021. doi: 10.1016/j.matpr.2021.08.197.
- [2] G.S. Lewis et al., "Finite element analysis of fracture fixation," *Current Osteoporosis Reports*, vol. 19, no. 4, pp. 403–416, 2021. doi: 10.1007/s11914-021-00690-y.
- [3] D.-K. Kwak, S.-H. Bang, W.-H. Kim, S.-J. Lee, S. Lee, and J.-H. Yoo, "Biomechanics of subtrochanteric fracture fixation using short cephalomedullary nails: A finite element analysis," *PLOS ONE*, vol. 16, art. no. e0253862, 2021. doi: 10.1371/journal.pone.0253862.
- [4] D. Takahashi et al., "Finite element analysis of double-plate fixation using reversed locking compression-distal femoral plates for Vancouver B1 periprosthetic femoral fractures," *BMC Musculoskeletal Disorders*, vol. 22, art. no. 276, 2021. doi: 10.1186/s12891-021-04152-5.
- [5] A.Y. Babil et al., "Effect of different constraining boundary conditions on simulated femoral stresses and strains during gait," *Scientific Reports*, vol. 14, art. no. 10808, 2024. doi: 10.1038/s41598-024-61305-x.
- [6] T. Yang, F. Noraddin, B. Liu, Z. Zhang, and H.-L. Gu, "Finite element analysis of implant selection and screw positioning in proximal femoral basicervical fractures," *Scientific Reports*, vol. 15, art. no. 35360, 2025. doi: 10.1038/s41598-025-19260-8.
- [7] P. Bazyar, A. Baumgart, H. Altenbach, and A. Usbeck, "An overview of selected material properties in finite element modeling of the human femur," *Biomechanics*, vol. 3, no. 1, pp. 124–135, 2023. doi: 10.3390/biomechanics3010012.
- [8] M. Ceddia, E. Pesare, G. Solarino, L. Lamberti, and B. Trentadue, "Biomechanical comparison of titanium and CFR-PEEK intramedullary nails using finite element analysis," *Journal of Composites Science*, vol. 9, no. 11, art. no. 576, 2025. doi: 10.3390/jcs9110576.
- [9] Y. Mori et al., "A review of the impacts of implant stiffness on fracture healing," *Applied Sciences*, vol. 14, no. 6, art. no. 2259, 2024. doi: 10.3390/app14062259.
- [10] 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," *Frontiers in Bioengineering and Biotechnology*, vol. 11, art. no. 1092361, 2023. doi: 10.3389/fbioe.2023.1092361.
- [11] C. Wang et al., "Biomechanical evaluation of a new intramedullary nail compared with proximal femoral nail antirotation and InterTAN for the management of femoral intertrochanteric fractures," *Frontiers in Bioengineering and Biotechnology*, vol. 12, art. no. 1353677, 2024. doi: 10.3389/fbioe.2024.1353677.
- [12] Y. Yang et al., "Comparative study of a novel proximal femoral bionic nail and three conventional cephalomedullary nails for reverse obliquity intertrochanteric fractures: a finite element analysis," *Frontiers in Bioengineering and Biotechnology*, vol. 12, art. no. 1393154, 2024. doi: 10.3389/fbioe.2024.1393154.
- [13] J. Zhu et al., "Effects of different reduction patterns on stress distribution in patients with intertrochanteric fractures with intramedullary nail fixation: a finite element analysis," *Frontiers in Bioengineering and Biotechnology*, vol. 13, art. no. 1507774, 2025. doi: 10.3389/fbioe.2025.1507774.
- [14] K. Ding et al., "Proximal femoral bionic nail—a novel internal fixation system for the treatment of femoral neck fractures: a finite element analysis," *Frontiers in*



Bioengineering and Biotechnology, vol. 11, art. no. 1297507, 2023. doi: 10.3389/fbioe.2023.1297507.

[15] Q. Huang et al., “Biomechanical evaluation of two modified intramedullary fixation system for treating unstable femoral neck fractures: A finite element analysis,” *Frontiers in Bioengineering and Biotechnology*, vol. 11, art. no. 1116976, 2023. doi: 10.3389/fbioe.2023.1116976.

[16] Z. Zhu et al., “Biomechanical study of different internal fixation devices for femoral neck fractures: finite element analysis based on different reduction qualities,” *Frontiers in Bioengineering and Biotechnology*, vol. 13, art. no. 1678294, 2025. doi: 10.3389/fbioe.2025.1678294.

[17] X. Zhang, S. Zhang, Z. Zhong, W. Zhang, and Z. Xiong, “Computational evaluation of the biomechanical effects of position changes in the femoral neck system on Pauwels type III femoral neck fractures: an in silico study,” *Frontiers in Bioengineering and Biotechnology*, vol. 13, art. no. 1493555, 2025. doi: 10.3389/fbioe.2025.1493555.