International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Finite Element Analysis of Femur Bone Exploring Different Loading Conditions and Modelling Fracture Scenarios

Prasanna Raut [a*], ***Dr Mahesh M Kawade*** [b], ***Pratik Waghmare*** [c]

[a*] *Research Scholar, Veermata Jijabai Technological Institute, Mumbai, India*
[b] *Associate Professor, Department of Mechanical Engineering, PES's Modern College of Engineering, Savitribai Phule Pune University, India.*
[c] *Department of Mechanical Engineering, Savitribai Phule Pune University, Pune, India*

DOI: https://doi.org/10.55248/gengpi.5.0124.0223

ABSTRACT

Early in life, bones develop quickly because they contain critical elements like calcium and phosphorus. They also have particular physical properties, including linear elasticity, isotropy, and consistency. They provide support for the softer parts of the body and are an essential component of the human skeleton. Trauma affects both developed and developing nations and is a major global cause of mortality and disability. In a prior prediction, the World Health Organisation stated that by 2020, trauma will overtake other causes as the major cause of years of life lost in urban and developing countries. The fabrication of bone joints utilising different biomaterials, such as lightweight, high-strength aluminium alloy (Al) and silicon carbide, is one method of repairing bone fractures, which are usually caused by trauma. In order to determine the best material for fracture fixing, the main goal of this study is to evaluate different bone joints in terms of their material qualities and capacity to increase strength. The femur bone is modelled for the investigation, and CREO and ANSYS are used to analyse its characteristics. In addition, modelling and analysis of fixation joints for broken bones are part of the project.

Keywords: CREO, Fracture, Strength, Tissues and ANSYS.

Introduction

The usage of metal dental implants dates back thousands of years in the history of biomaterials, according to archaeological research. However, following World War, considerable developments in biomaterials took place. Biomaterials are now understood to be "artificial or natural substances employed in the construction of structures to replace lost or damaged biological tissue, thereby restoring its shape and function" [1-5]. Cells and biomaterials can interact in ways that cause particular reactions. Two important criteria are used to assess their performance: biofunctionality and biocompatibility [5–9]. The finite element method is a method for tackling integral and differential equations using numerical methods to solve boundary value issues. A useful application of this methodology is finite element analysis (FEA), which is used in materials mechanics to analyse both new product designs and old designs [10-14]. The FEA method is used in this work to examine the human femur, or thigh bone. The femur, which makes up around 26% of a person's height [15-18], is the longest, heaviest, and one of the strongest bones in the human body. In anthropology, this ratio is particularly useful since it provides a foundation for determining a person's height from partial skeletal remains. Numerous engineering applications, including orthopedic biomechanics for evaluating stresses in human bones, employ the widely used technique of finite element analysis (FEA) to examine structural stresses [19-23]. FEA, as demonstrated, aids to the design of implants by identifying high-stress locations as shown in figure1.

Fig. 1 - Artificial Hip Joint and Femur Interaction

It is well acknowledged that the Finite Element Method is an important tool in biomechanics, particularly for examining where to place implants in femur fractures. These studies not only aid in our understanding of the connection between structure and function, but they also give medical professionals essential knowledge for their work [24-27]. A large portion of recent progress has been on system-level device deployment and refinement, with major effects on human interaction, clinical care, prosthetics, orthotics, and orthopaedic surgery. Understanding failure causes and providing direction for femur replacement design and procedures depend on the methodology of finite element analysis (FEA) and finite element modelling of the femur bone under physiological settings. Using software like PRO/Engineer, Solid Edge, MIMICS, etc., and continuous anatomical data derived from radiographs or CT/MRI scans, one may create a computer-aided design (CAD) model [28-32]. Finite Element Analysis is used to examine difficulties and deformations experienced throughout different activities. Due to the nonlinear behaviour and great heterogeneity of human bones, it is difficult to assign material attributes to every direction of the bone model [33-34]. Materials can be assigned in two different ways in biomechanical studies: either inside Mimics or within the finite element model. The workflow for femur bone analysis using FEA is shown in Figure 2.

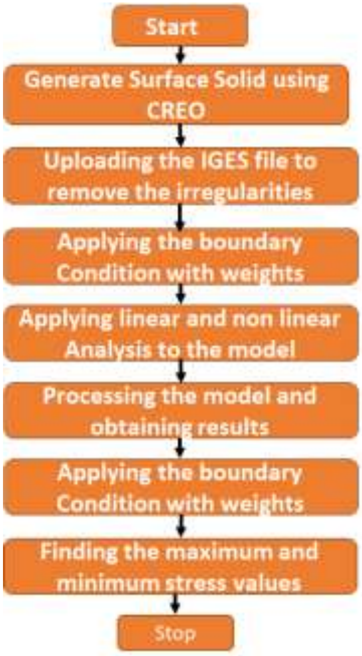

Fig. 2 - Flow Chart for Femur Analysis Method

The ability of ANSYS's comprehensive capabilities to address challenging engineering problems is well known. You can successfully build the model required for analysis with careful planning and the use of various methodologies. To build your model, however, you could find it more advantageous to use Design Modeller in the ANSYS Workbench environment or hire a CAD modeller. An ANSYS model can be generated using either the direct generation technique with solid modelling or the solid modelling methodology. In the solid modelling technique, you declare the required forms, set controls over component sizes, and define the model's geometric borders. The ANSYS system is then told to automatically create all the nodes and elements as necessary [35-37]. In contrast, the direct generation approach entails locating each node and figuring out each component's size, shape, and connection before adding them to the ANSYS model.

Material Selection

The biomaterials chosen for this study include:

- Stainless steel
- Aluminium alloy and silicon carbide composition

The dimensions of the human femur bone were reproduced in CREO Software using information from Journal [7]. In order to analyse the human bone, ANSYS was then used. Figure 3 shows a visual depiction of the CREO-modeled femur bone. The actions taken throughout the analysis process are described in the following sections.

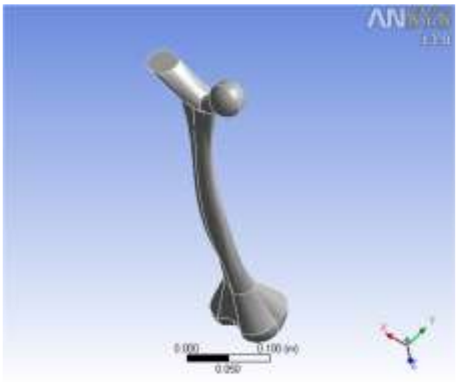

Fig. 3 - Modeling of a femur bone

Since the structure of human bones is very heterogeneous and nonlinear, it is difficult to allocate material qualities to every direction inside the bone model. There are two places where material attributes can be set: either in the Mimics module or in the Finite Element module. In this instance, ANSYS allows direct specification of the material characteristics. Density of 2800 kg/m3, Young's modulus of 1.181 GPa, and Poisson's ratio of 0.28 have all been used in the research.

Pre-processing is the process of creating a component or assembly (model), material characteristics, and pertinent boundary conditions in order to describe a physical structure using finite element analysis (FEA). It entails solving the system to provide a numerical representation of it, followed by post-processing to analyse the output of that representation. The majority of real-world components and assemblies are too complicated to be precisely, much alone fast, analysed without the aid of a computer and the proper analysis tools, despite the fact that basic forms and straightforward issues may frequently be addressed manually.

After the model has been built, the femur bone model's surface mesh must be generated using Finite Element Analysis (FEA). In this instance, a small mesh size was used. There were 25,251 elements and 45,247 nodes produced as a consequence of this procedure. A visual example of the mesh-based femur bone model in ANSYS is shown in Figure 4.

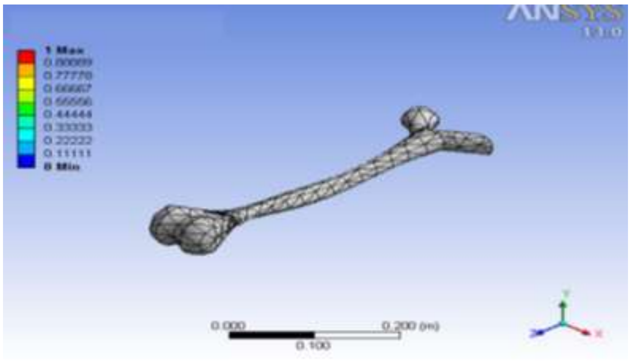

Fig. 4 - Meshed model

Table 1 - Lists the femur bone's compositional characteristics.

Properties of Material	Value of the Bone
Youngs Modulus	1.20
Poisson’s Ratio	0.28
Density Kg/m^3	2800

Results and Discussion

In this study, a femur bone exposed to 910 pascals of pressure in the usual position for men is examined for stress distribution, total deformation, and fatigue failure. The largest overall deformation measured when an eccentric load was applied was 0.00003547 metres. The results show that the femur's head has the most distortion, whereas its lower end experiences the least deformation. The central region of the femur had the highest primary stress, with the maximal equivalent stress of 1.5e5 pascals. The Von Mises equivalent stress was 1.51e5 pascals.

The study also investigates the stress distribution, total deformation, and fatigue failure of a femur for men in the usual position under 900 pascals of pressure. Both steel and aluminium alloy with silicon carbide composition showed a maximum total deformation of 0.00003325 metres and 0.000054127 metres, respectively, when an eccentric load was applied. Similar to the prior case, the femur's head showed larger distortion, while its lower end showed the lowest deformation. The central portion of the femur generated the most primary stress in both materials, with the greatest equivalent stress in both materials reaching 1.5e5 pascals. Refer to Figures 5 and 6 for graphical depictions of the equivalent stress (Von Mises), which was 1.51e5 pascals.

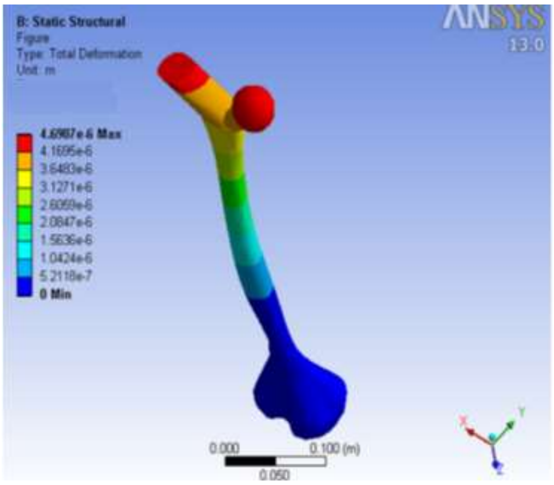

Fig. 5 - Deformation Assessment of Femur Bone using Aluminum-Silicon Carbide Composite

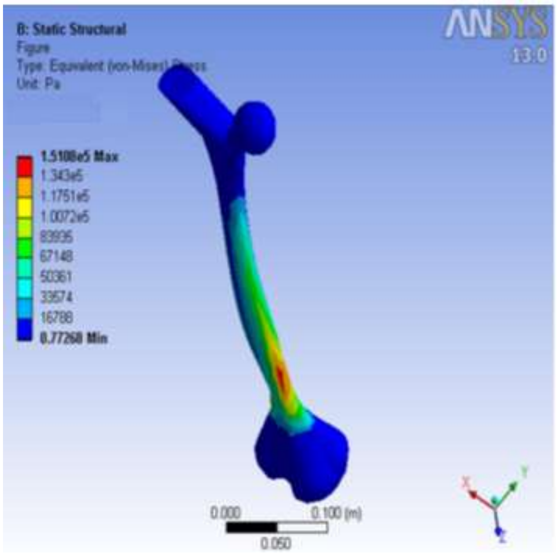

Fig. 6 - Equivalent Stress Analysis of Femur Bone with Steel Material

With different loads, a femur bone model underwent Finite Element Analysis (FEA). The highest stresses discovered by this study are shown in Table 2.

Table 1 - Results of Axial and Bending Load Testing and Observations of Failures.

Loading Input Parameter in kg	**Out Parameter**	
	Axial Stress in Mpa	Bending Stress in Mpa
25	6.21	36.21
40	11.01	51.45
70	26.78	102.13*
160	37.41	220.14
270	61.45	370.14
450	100.012*	605.23
600	130.14	750.14

The results show that a force of 450 kg applied axially and a load of 70 kg applied bending causes the femur bone to break. These findings unequivocally show that the femur bone is stronger in the axial direction than in the bending direction. Figure 7 depicts the increased stress levels.

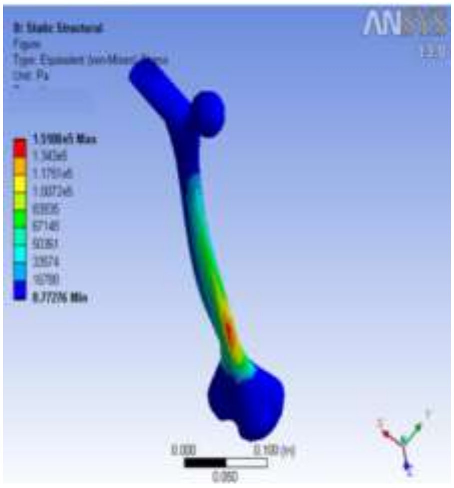

Fig. 7 - Equivalent Stress Analysis of Femur Bone with Aluminum-Silicon Carbide Composite

Conclusion

The following inferences may be made from the Finite Element Analysis (FEA) of the human femur bone. The femur bone's higher resistance to axial stresses is highlighted by the fact that its axial strength is around six times larger than its bending strength. The femur bone in humans exhibits an outstanding capability for load bearing, being able to sustain stresses up to 10 times its own body weight. Given the special mechanical characteristics of real bone, the examination of the data clearly suggests the difficulties in discovering replacement materials for bones. A model that had been put together to for FEA analysis to determine which materials would experience the least amount of stress during loading. Al + Sic composition showed the lowest equivalent stress and total deformation among the investigated materials, indicating its potential as a viable biomaterial. Further research is conducted to acquire the most suitable material by taking into account factors such as stiffness, corrosion resistance, wear resistance, and the Finite Element Method (FEM) results. Stainless steel is identified as the optimal material, exhibiting a deformation of 2.80e-6 and an equivalent stress of 1.47e5 Pa. Additionally, for Al+Si, the total deformation is 5.01e-6, and the equivalent stress remains at 1.47e5.

References

1. Huiskes, R.; Chao, E. Y. S. A Survey of Finite- Element Analysis in Orthopedic Biomechanics – The 1st Decade. Journal of Biomechanics 1983, 16 (6), 385.

2. P.S.R. Senthil Maharaj et al., Numerical analysis of fracture femur bone with prosthetic bone plate Procedia Engineering 64 (2013) 1242 – 1251

3. A Kalaiyarasan, K Sankar, S Sundaram Finite element analysis and modeling of fractured femur bone, Materials Today: Proceedings, 2019, Volume 22, Part 3, 2020, Pages 649-653

4. Khilare, Umesh A., and S. B. Sollapur. "Investigation of Residual Stresses and Its Effect on Mechanical Behaviour of AISI310."

5. Shinde, Tarang, et al. "Fatigue analysis of alloy wheel using cornering fatigue test and its weight optimization." Materials Today: Proceedings 62 (2022): 1470-1474.

6. Shrishail Sollapur, Saravanan, D., et al. "Tribological properties of filler and green filler reinforced polymer composites." Materials Today: Proceedings 50 (2022): 2065-2072. https://doi.org/10.1016/j.matpr.2021.09.414

7. Raut, P.P., Rao, A.S., Sollapur, S. et al. Investigation on the development and building of a voice coil actuator-driven XY micro-motion stage with dual-range capabilities. Int J Interact Des Manuf (2023). https://doi.org/10.1007/s12008-023-01665-2

8. Shrishail, B. Sollapur, and P. Deshmukh Suhas. "XY scanning mechanism: a dynamic approach." International Journal of Mechanical Engineering and Robotics Research 3.4 (2014): 140.

9. Swapnil S. Shinde, Shrishail B Sollapur, "Effect of Residual Stress on the Mechanical Behavior of AISI 304, For TIG Welding", International Journal of Scientific & Engineering Research(IJSER), Volume 8, Issue 10, October-2017.

10. Toradmal, Kuldeep P., Pratik M. Waghmare, and Shrishail B. Sollapur. "Three-point bending analysis of honeycomb sandwich panels: experimental approach." International Journal of Engineering and Techniques 3.5 (2017).

11. S B Sollapur et. al "Mechanical Properties of Bamboo Fiber Reinforced Plastics", IJSART - Volume 3 Issue 9 –SEPTEMBER 2017 ISSN [ONLINE]: 2395-1052, Page | 365-368

12. Khilare, Umesh A., and S. B. Sollapur. "Investigation of Residual Stresses and Its Effect on Mechanical Behaviour of AISI310." Journal for Research | Volume 02 | Issue 05 | July 2016 ISSN: 2395-7549, PP 42-46.

13. Waghmare, Pratik M., Pankaj G. Bedmutha, and Shrishail B. Sollapur. "Investigation of effect of hybridization and layering patterns on mechanical properties of banana and kenaf fibers reinforced epoxy biocomposite." Materials Today: Proceedings 46 (2021): 3220-3224.

14. S B Sollapur et. al “Mechanical Properties of Bamboo Fiber Reinforced Plastics”, IJSART - Volume 3 Issue 9 –SEPTEMBER 2017 ISSN [ONLINE]: 2395-1052, Page | 365-368

15. B.A. Szabo, Babuska, Finite element analysis, Wiley, 1991. ISBN: 978-0-471-50273-9 March 1991 384

16. O.C. Zienkiewicz, et al., The finite element method: Its basis and fundamentals, Elsevier Science, 2005.

17. A.Kalaiyarasan S.Sundaram Finite element analysis of beam with different loading condition Volume 43, Part 2, 2021, Pages 2189-2193

18. Taner Zuylan, Khalil Awadh Murshid, 2001. An Analysis of Anatolian Human Femur Anthropometry. Turk J Med Sci 32 (2002) 231–235 T.BUTAK.

19. Gireesh, Belawagi, B. Sollapur Shrishail, and V. N. Satwik. "Finite element & experimental investigation of composite torsion shaft." Int. J. Engg. Research and Applications 3.2 (2013): 1510-1517.

20. Deore, Om Bipin, Shrishail Sollapur. "Design and Analysis of Complaint Mechanism using FEA." development 7.10 (2020).

21. Baviskar D.D., Rao A.S., Sollapur S. et al. Development and testing of XY stage compliant mechanism. Int J Interact Des Manuf (2023). https://doi.org/10.1007/s12008-023-01612-1

22. Shrishail Sollapur, Shraddha Gunjawate, “Structural Analysis and Topology Optimization of Leaf Spring Bracket”, International Journal of Engineering Research & Technology (IJERT) Vol. 9 Issue 07, July-2020. pp. 1448-1494

23. Sollapur, Shrishail, M. S. Patil, and S. P. Deshmukh. "Position Estimator Algorithm Implementation on Precision Applications." Materials Today: Proceedings 24 (2020): 333-342.

24. S B Sollapur et. al “Experimental Investigation of High Precision XY Mechanism”, International Journal of Mechanical Engineering and Technology,9(5),2018,pp.43–50.

25. Sollapur, Shrishail B., M. S. Patil, and S. P. Deshmukh. "Design and development aspects of flexure mechanism for high precision application." AIP Conference Proceedings. Vol. 1943. No. 1. AIP Publishing, 2018

26. A.H. Bustein et al., the ultimate properties of bone tissue. The effect of yielding, J. Biomech. 5 (1) (1972) 35–44.

27. Umesh K, “Identifying Design Alternative For Piping System Upon Assessment Of Composite Material For Suitability To The Engineering Application”, In International Journal of Advanced Engineering Research and Studies EISSN2249– 8974 III/IV/July-Sept.,2014.

28. S B Sollapur, “Evaluation of Stiffness and Parametric Modeling of XY Flexure Mechanism for Precision Applications”, Journal of Modeling and Simulation of Materials, vol. 1, no. 1, pp. 8-15, May 2018. doi: 10.21467/jmsm.1.1.8-15

29. Shrishail Sollapur, M S Patil, S P Deshmukh, “Advancement and Experimental Investigation of Voice Coil Actuator utilizing Flexural Bearing”, Journal of Mechatronics and Automation, STM Journals, Vol 5, Issue 2, 2018. pp. 40-45

30. Shrishail Sollapur, Pratik Waghmare, “Design and Experimental Investigation of XY Compliant Mechanism for Precision Applications”, ECS Transactions, 2022/4/24, Volume 107 Issue 1 Pages 4967

31. P M Waghmare et al, "A Review Paper on Flexure", International Journal for Science and Advance Research In Technology, Vol 3, Issue 10, 2017.

32. Pratik Waghmare, “Development and Performance Investigation of Solar Concrete Collector at Different Climatic Conditions”, Indian Journal of Engineering and Materials Sciences, https://doi.org/10.56042/ijems.v30i2.1384

33. Waghmare, Pratik M., Shrishail B. Sollapur, and Shweta M. Wange. "Concrete Solar Collector." Advances in Smart Grid and Renewable Energy: Proceedings of ETAEERE-2016. Springer Singapore, 2018.

34. Sollapur, S.B., Sharath, P.C., Waghmare, P. (2024). Applications of Additive Manufacturing in Biomedical and Sports Industry. In: Rajendrachari, S. (eds) Practical Implementations of Additive Manufacturing Technologies. Materials Horizons: From Nature to Nanomaterials. Springer, Singapore. https://doi.org/10.1007/978-981-99-5949-5_13

35. Chate, Ganesh R., et al. "Ceramic material coatings: emerging future applications." Advanced Ceramic Coatings for Emerging Applications. Elsevier, 2023. 3-17.

36. T. Stolarski, Y. Nakasone, et al. Engineering analysis with ANSYS software, Elsevier Science, 2014.

37. Vinod, M., Kumar, C.A., Sollapur, S.B. et al. Study on Fabrication and Mechanical Performance of Flax Fibre-Reinforced Aluminium 6082 Laminates. J. Inst. Eng. India Ser. D (2023). https://doi.org/10.1007/s40033-023-00605-4