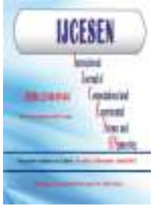
International Journal of Computational and Experimental Science and ENgineering
(IJCESEN)

Vol. 10-No.4 (2024) pp. 621-629
http://www.ijcesen.com

ISSN: 2149-9144

Research Article



Modal and Harmonic Analysis of Femur using Different Boundary Conditions by Finite Element Analysis

S. Mathu Kumar[1]*, V.A. Nagarajan[2], A Radhakrishnan[3]

[1]Faculty of Mechanical Engineering, Ponjesly College of Engineering, Nagercoil, Tamilnadu, India-629 003
* **Corresponding Author Email:** smathukumar@gmail.com - **ORCID:** 0000-0003-4147-9620

[2]Faculty of Mechanical Engineering, University College of Engineering Nagercoil, Tamilnadu, India – 629 004.
Email: nagarajanvava@gmail.com - **ORCID:** 0000-0002-5610-8796

[3]Faculty of Information Technology, University College of Engineering Nagercoil, Tamilnadu, India – 629 004.
Email: radhakrishnan.a@auttvl.ac.in - **ORCID:** 0000-0001-6218-7913

Article Info:

DOI: 10.22399/ijcesen.364
Received : 30 June 2024
Accepted : 08 October 2024

Keywords :

Modal Analysis
Harmonic Analysis
Finite Element Analysis

Abstract:

The longest and largest bone in the human body is the femur. Pelvic bone sustains the weight of the body to which the femur bone is connected. Many researches have been carried out to identify the behaviour of the femur bone. The study aimed to explore the natural frequencies and mode shapes of vibrating devices to gain insights into the dynamic behavior of the femur bone during various physical activities. Examining the impact of patient-specific bone shape and density on bone natural frequencies is crucial. The primary goals of femur bone analysis involve employing computer simulations for fracture detection and employing Finite Element (FE) models to determine natural frequencies and vibration modes. To obtain the natural frequency of the femur bone, different boundary conditions such as free-fixed and fixed-fixed are applied. Avoiding the coincidence of the natural frequency with external excitation frequencies is crucial to prevent femur bone fractures. Also, for different magnitude of loads, femur bone is involved in harmonic analysis is to identify the amplitude and stress against acceleration. The study measured amplitude and stress values throughout a frequency range of 1 Hz to 1500 Hz while altering the acceleration and load. As a result, identifying a resonance frequency combined with excessive stress will have an effect on the femur bone.

1. Introduction

The principles of mechanical engineering are applied to biological systems. This involves dynamics of mechanism, stress, strain, vibration patterns as well as analysis of fluid flow and kinetics present in human body. Finite Element Analysis (FEA) is a very popular tool for performing static and dynamic loading of a human organ.

In contemporary times, bone fractures, particularly resulting from accidents, have become a significant health concern. The femur bone, constituting a substantial portion of the body's weight, is among the most commonly fractured bones in the human body. Consequently, it is imperative to exercise heightened caution and adopt special precautions when using assistive, sporting, or defensive equipment to minimize the risk of resonance and exposure to external excitation frequencies. Modal analysis proves invaluable for discerning vibrational characteristics, encompassing mode shapes and natural frequencies [1].

To anticipate the safe operational limits of human activities, a comprehensive analysis was conducted, encompassing both vibrational characterization and stress analysis across various modes. It is crucial to gain insights into how the specific bone shape and density distribution unique to each patient impact the natural frequencies of bones [2-3].

The mechanical behaviour of biological objects such as bones are dealt with the biomechanical study of bones [4]. The analysis of complicated systems has been made easier and less time consuming [5] in the recent development in the field of computations. The femur bone, being the longest and most robust support structure in the body, plays a pivotal role in transferring the entire weight of the body to the leg [6]. Hence it is essential to understand the behaviour of this bone and through FEA, it need to find the feasible implant materials [7]. Generally, femur bone fractures occur due to the trauma caused by accidents [8]. To analyse the frequencies for different materials, model analysis is one of the criteria to predict whether model frequency exceeds natural frequency [9,10]. To perform femur bone implants, stress analysis and model analysis play an important role [11.12]. In order to design the implants properly it is necessary to find the critical places where the fracture occurs [13]. In the analysis of the femur bone, the choice of implant material is a crucial consideration. Frequency studies are conducted using a model to assess the effects under specified boundary conditions [14].

Any object's physical properties directly influence its vibration characteristics. Therefore, changing the physical properties will change the natural frequencies of that object. A fracture in a bone will change its physical features, which will affect its natural frequencies and mode shape. [15]. The stability of fixing of hip stem in vitro and vivo is analysed using vibration analysis [16]. Bone density is a critical factor as it aids in predicting the likelihood of experiencing a fracture. To assess bone density, a BMD (Bone Mineral Density) test is employed, which measures the quantity of calcium present in specific bone regions. This test provides an estimation of bone strength, and it is typically conducted using computed tomography (CT) or dual-energy X-ray absorptiometry (DEXA) techniques [17].

Certainly, vibration analysis, a non-destructive testing method, is widely favored among various available techniques for assessing factors like fractures, postures, osteoporosis, and the stability of artificial hip joints. This method involves mechanically stimulating the bone and analyzing its response. For a comprehensive analysis, FEA proves to be a powerful tool capable of conducting both static and dynamic loading assessments on bones. FEA is a dependable tool for analyzing structures with complex geometries and non- homogeneous materials. Given that the femur bears the majority of the human body's weight, it indeed carries the highest risk of experiencing failure or fractures.

To analyse structures of complicated geometry and inhomogeneous material properties, FEA a computer-based method is used. [18]. It is used as an alternative tool for biomechanics modelling [19] which has heterogeneous material properties and complicated geometrical shapes.

FEA plays a crucial role in predicting femur fractures. This approach involves the utilization of computational models and biomechanical stress analysis within the human femur to evaluate fracture risks [20-23]. The process entails creating a three-dimensional FE model of the femur, incorporating both its geometrical and mechanical properties, derived from computed tomography (CT) images. Vibration analysis techniques are then employed to validate the model based on CT images, ensuring its accuracy in representing the real femur. The study also investigates how variations in mechanical properties impact the model's parameters.

Furthermore, vibration analysis is conducted, and FEA is used to extract the natural frequencies of the femur. Modal and harmonic analyses are employed to accurately predict the natural frequency and resonance frequency of vibrations within the human body [24,25]. This methodology helps in understanding and potentially preventing femur fractures.

2. Materials and Methods

2.1 Materials

Utilizing a 3D FE model, researchers developed a representation of the left femur bone based on a volumetric reconstruction of coronal CT images. For this study of the femur bone, material characteristics such as a Young's modulus of 14,200 MPa, a Poisson's ratio of 0.35, and a bone density of 2,080 kg/m³ were employed for investigative purposes [26]. The CT image obtained was imported into 3D Doctor Medical Image processing software. Femur reconstructed and meshed femur model is shown in Fig. 1 & Fig. 2.

Furthermore, it is assumed that the femur is composed of an isotropic and homogeneous material. Femur was included mesh for the crystallization of the femur model. It is converted into small elements. Here, a tetrahedron-shaped solid element is used as a mesh in the femur model.

2.2 Vibrational Analysis

The equation employed for the analysis is provided as follows:

$$[M]\{x(t)\} + [C]\{x(t)\} + [K]\{x(t)\} = \{F(t)\} \qquad (1)$$

Where,

[M] = Mass Matrix, [K] = Stiffness matrix, [C] = Damping Coefficient matrix

In the context of undamped free vibration analysis, where both the external excitation force and damping are set to zero ([F] = 0, [c] = 0), Equation (2) can be expressed as follows:

Therefore, the solution to the aforementioned equation can be expressed as:

$$[M]\{x(t)\} + [K]\{x(t)\} = 0 \qquad (2)$$

The solution to the equation provided above can be expressed as follows:

$$\{x\} = \{X\}ei\omega t \qquad (3)$$

In the given solution, {X} represents the respective amplitudes of masses, and the variable ω signifies the corresponding frequency of each Eigenvector. As a result, Equation (2) simplifies to the following: Now, the governing equation can be expressed as:

$$[[K]-\omega^2[M]]\{X\}=0 \tag{4}$$

The problem described above can be treated as linear by substituting ω^2 with λ, allowing it to be solved as a linear problem in matrix algebra [27]. Consequently, Equation (3) transforms into a linear problem in matrix algebra.

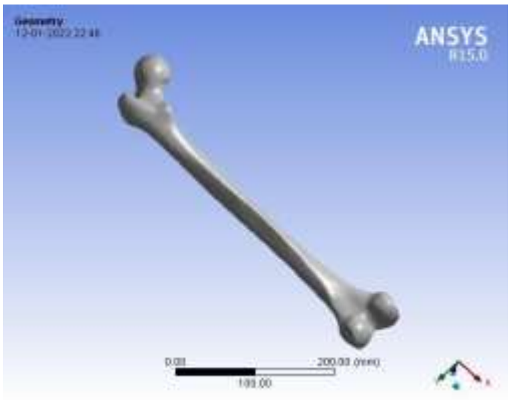


***Figure 1.** Reconstructed femur model*

***Figure 2.** Meshed femur model*

2.3 Modal Analysis

During modal analysis, the geometrical model is subjected to two different boundary conditions. The first boundary condition applied to the femur is a free-fixed configuration, as illustrated in Fig.3.

Various results have been taken with distal end fixed and femoral head free boundary conditions. The second boundary condition is a fixed-fixed configuration. In this fixed-fixed boundary condition, both the femoral head and the distal end are constrained from moving, as depicted in Fig.4. Natural frequency and deformation of six modes in the femur of fixed-free boundary condition are shown in the Table 1. All of these scenarios are utilized to determine the natural frequencies of the femur. The analysis specifically considers the first six vibration mode shapes by applying a free-fixed boundary condition.

***Table 1.** Natural frequency and deformation for the fixed-free boundary condition.*

Mode	Frequency [Hz]	Deformation(mm)
1	54.111	63.019
2	56.587	61.912
3	343.22	87.951
4	376.64	64.999
5	505.23	93.804
6	957.95	70.883

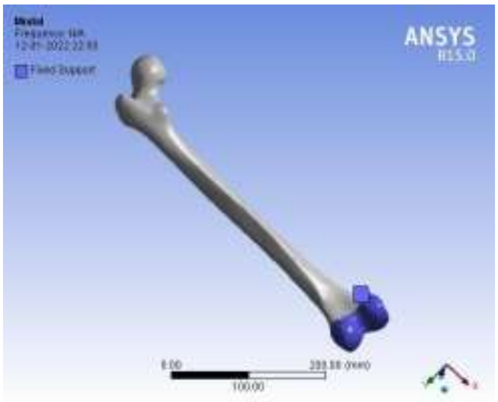


***Figure 3.** Free-fixed boundary conditions*

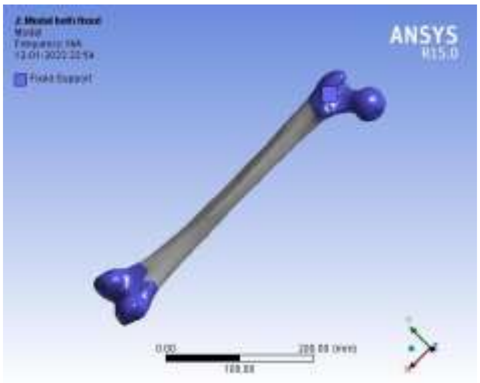


***Figure 4.** Fixed-fixed boundary conditions*

These six mode shapes are visually presented in Fig.5.

To determine the natural frequencies under fixed-fixed boundary conditions, the analysis considered the first six vibration modes. Six mode shapes are shown in the Fig.6.

To determine the natural frequencies under fixed-fixed boundary conditions, the analysis considered the first six vibration modes. Six mode shapes are shown in the Fig.6. The Table 2 displays the natural frequencies and deformation characteristics of the femur under fixed-fixed boundary conditions for six different vibration modes.

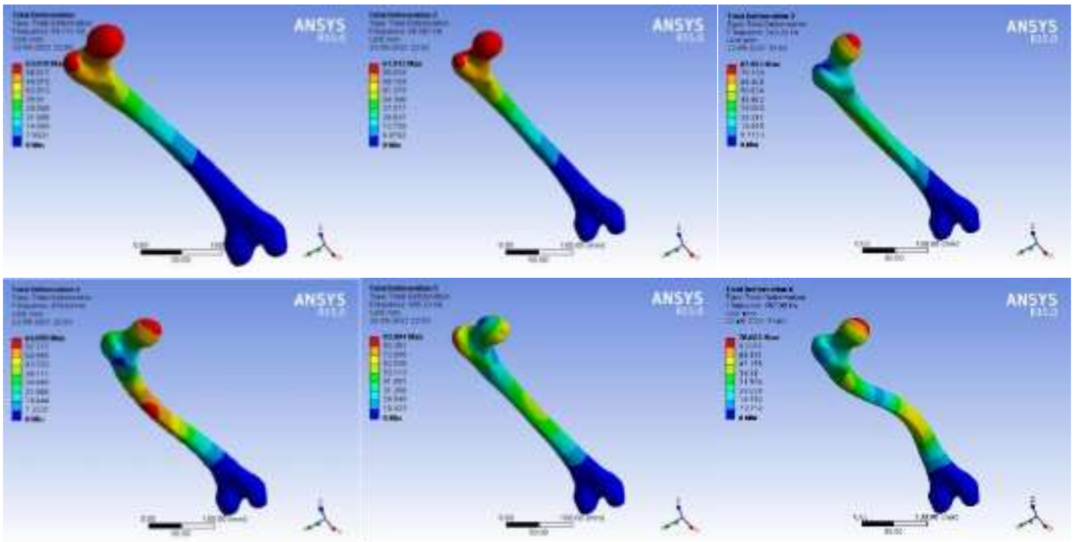


Figure 5. *Mode shapes for Free-Fixed boundary condition*

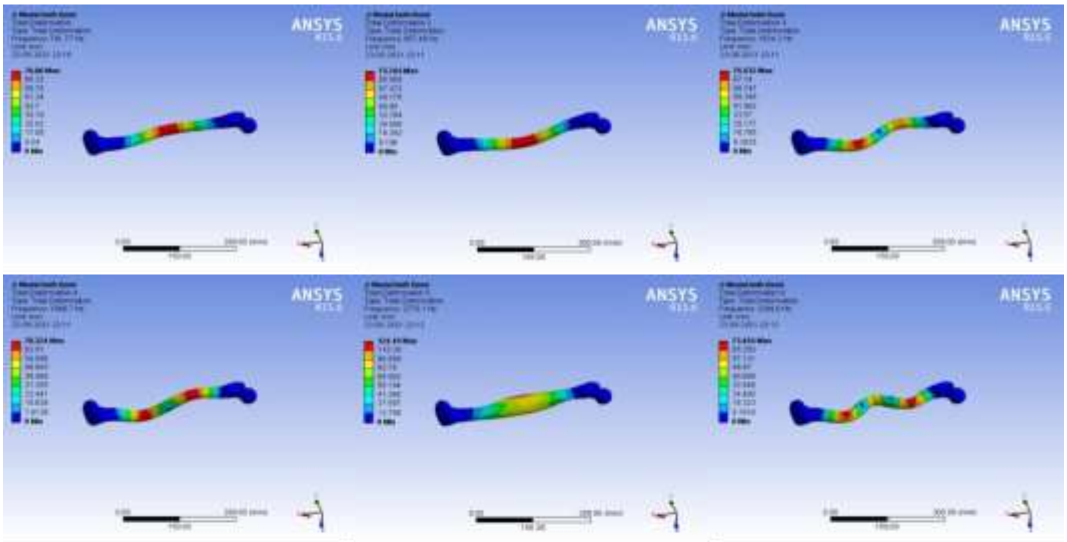


Figure 6. *Mode shapes for Fixed-Fixed boundary condition*

Table 2. *Natural frequency and deformation for the fixed-fixed boundary condition*

Mode	Frequency [Hz]	Deformation (mm)
1	741.77	76.86
2	807.49	73.76
3	1834.2	75.53
4	1966.7	70.32
5	2770.1	124.19
6	3299.6	73.45

2.4 Harmonic Analysis

In this analysis, the human body model will experience a range of acceleration magnitudes across specific frequencies. When the structure or body is exposed to a constant sinusoidal excitation, harmonic analysis proves effective in understanding the dynamic behavior of the body. Because this analysis operates in the frequency domain, it is referred to as frequency response analysis [28].

The results of this analysis yield information regarding the deformation and stresses experienced by the structure along its x, y, and z axes. In this approach, the system is subjected to varying frequencies as input, and the corresponding outcomes are graphed in terms of stress versus frequency and deformation versus frequency. Through the application of harmonic analysis, we can pinpoint the resonant frequency at which the human femur bone model exhibits its most pronounced vibrations. When the frequency of external excitation aligns with the system's natural frequency, resonance can occur, potentially resulting in structural fractures or damage. This understanding can guide us in modifying the structure's geometry to prevent resonance. The study utilized ANSYS software to conduct the harmonic analysis, considering the previously mentioned material properties and applying appropriate boundary conditions to replicate the behavior of the femur bone. Harmonic analysis, also referred to as frequency response analysis, is a form of linear dynamic analysis used to assess a mechanism's response to excitation at precise frequencies. Before conducting a harmonic response analysis, a modal analysis is typically necessary, as the input frequencies for the harmonic response analysis are derived from the outputs of the modal analysis. This type of analysis is frequently employed to gauge the stresses induced by continuous harmonic loading. It allows us to determine the steady-state response of a linear elastic system when subjected to a sequence of harmonic loads with specified frequencies and amplitudes. Harmonic response analysis is conducted to investigate the dynamic behavior of the femur bone when subjected to specific external forces influenced by the fixed-free boundary condition. During this analysis, frequency response functions are generated at different frequencies, and the peaks in amplitude provide insights into critical conditions associated with resonance [29]. In order to simulate harmonic analysis, the material properties, geometry, meshing, and boundary conditions mirror those used in the previously described modal analysis. For the harmonic analysis, the frequency range selected typically spans from 1 to 1500 Hz. The objective is to establish the connection between the frequency response amplitude and frequency for various loading conditions.Then, during the harmonic analysis geometrical model is subjected to four various axial loads at femoral head of the femur. They are 490N, 540N, 588N and 640N [30]. These loads are applied on the femoral head of the femur and distal end is fixed [31]. Boundary conditions applied for harmonic analysis are shown in Fig.7. The harmonic analysis was conducted using three acceleration values: 1 m/s², 2 m/s², and 3 m/s², applied in the tri-axial direction. This analysis identified resonant frequencies across all directions. Furthermore, the harmonic analysis revealed the maximum deformations and highest stress levels occurring when the human femur model vibrates in all directions.

3. Result and discussion

The most recent analysis of the static structural femur bone, considering fixed boundary conditions, has revealed natural frequencies falling within the range of 741.77 Hz to 3299.6 Hz. Additionally; the analysis has indicated that the maximum total deformation recorded during this analysis reaches 124.19 mm.

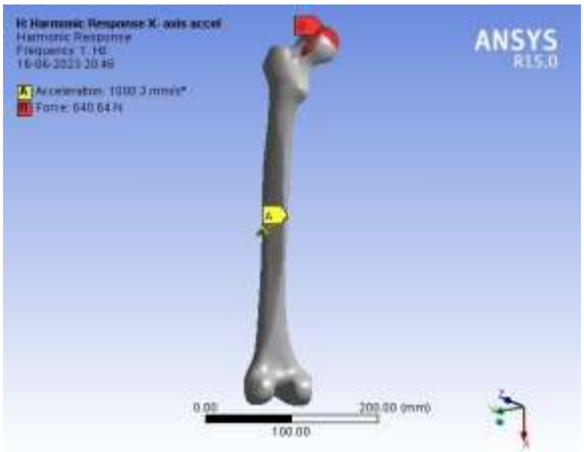


***Figure 7**. Boundary conditions applied for harmonic analysis*

Under the fixed-free boundary conditions, the analysis indicates that the natural frequencies span from 54.11 Hz to 951.95 Hz, resulting in a maximum total amplitude of 93.804 mm. Fig.8 and Fig.9 represents the relationship between mode shapes, frequencies, and corresponding amplitude values.

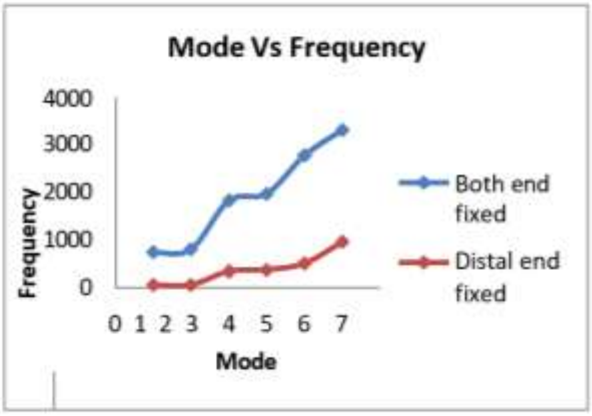


***Figure 8.** Mode Vs. Frequency for modal analysis*

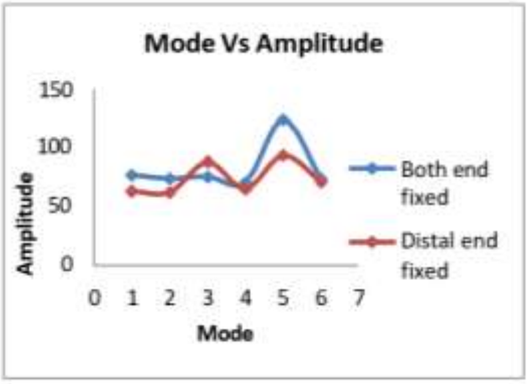


***Figure 9.** Mode Vs. Amplitude for modal analysis*

Harmonic analysis calculates the response of a structure that accelerates over a frequency range and the response is plotted on amplitude vs. frequency and frequency vs. stress graph. Fig.10 shows the frequency vs. amplitude and stress values for 1 m/s2, 2 m/s2 and 3 m/s2 acceleration values at 490 N at X, Y & Z axis.

For 1 m/s2 acceleration, values at 490 N, it is observed that the highest amplitude and highest stress developed at 50.96 Hz at X and Y axis are with an amplitude of 1.0395 mm and 20.711 mm and stress obtained are 4.305 and 0.1858 MPa respectively. The highest amplitude and stress recorded at 60.96 Hz at Z axis with an amplitude of 2.0346 mm and stress is developed at 50.96 Hz with the stress of 4.305 and 0.1858 MPa. A high stress was also recorded at 340.71 Hz was 0.0951 MPa. Similarly for 2 m/s2 and 3 m/s2 acceleration the results obtained are slightly greater than 1 m/s2 acceleration at 490 N.

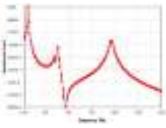
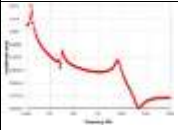
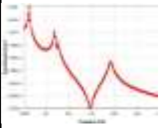
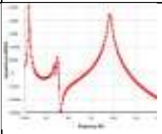
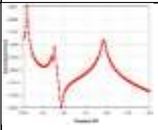
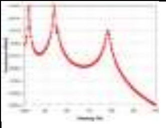
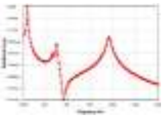
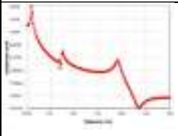
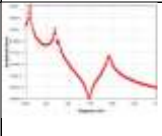
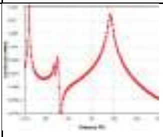
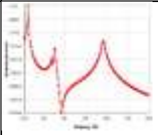
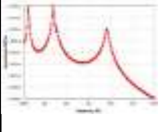
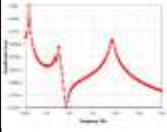
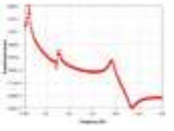
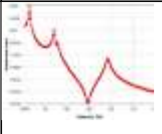
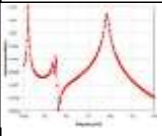
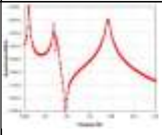
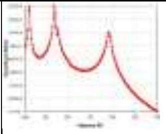
Acc.	Frequency Vs. Amplitude (490 N)			Frequency Vs. Stress. (490 N)		
	X-Axis	Y-Axis	Z-Axis	Y-Axis	Y-Axis	Y-Axis
1 m/s^2						
2 m/s^2						
3 m/s^2						

Figure 10. *Frequency Vs. Amplitude and stress of 490 N load for all axial direction acceleration*

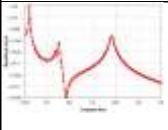
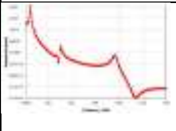
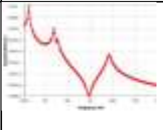
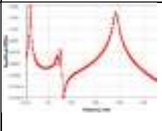
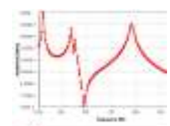
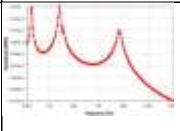
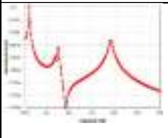
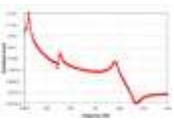
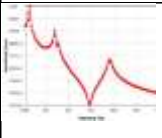
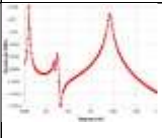
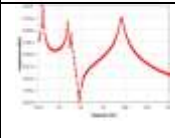
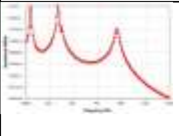
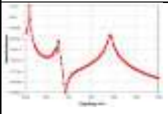
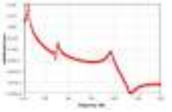
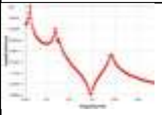
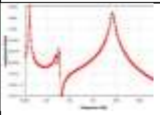
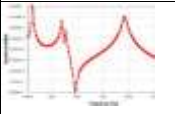
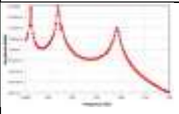
Acc.	Frequency Vs. Amplitude (540 N)			Frequency Vs. Stress (540 N)		
	X-Axis	Y-Axis	Z-Axis	X-Axis	Y-Axis	Z-Axis
1m/s^2						
2m/s^2						
3m/s^2						

Figure 11. *Frequency Vs. Amplitude and stress of 540 N load for all axial direction acceleration*

Fig.11 shows the results of frequency vs amplitude for 1m/s2, 2m/s2 and m/s2 acceleration values at 540 N at X, Y and Z axis respectively. The results obtained for 540 N, at 1 m/s2 acceleration recorded at X and Y axis for the amplitude and stress at 50.96 Hz with 1.1445 mm & 22.801 mm amplitude and stress obtained are 4.7404 & 0.2046 MPa respectively. The highest amplitude and stress at Z axis is noted as 60.96 Hz with an amplitude of 2.2399 mm and stress recorded at 340.77 Hz with value of 0.1047 MPa. Similarly, the results obtained for 2 m/s2 and 3m/s2 have some slight variations.

The results shown in Fig.12 for frequency vs. amplitude and stress for the same acceleration 1 m/s2, 2 m/s2 and 3 m/s2. For 1 m/s2 values at 588 N at X, Y & Z axis results 50.96 Hz with an amplitude of 1.2465 mm & 24.832 mm at the X & Y axis and stress recorded at 50.96 Hz are 5.1629 & 0.2228 MPa. At Z axis the result of amplitude recorded at 60. 96 Hz with 2.4371 mm and stress developed at 340.77 Hz was recorded as 0.1141 MPa. For 2 m/s2 and 3m/s2, the result tends to increase in stress and amplitude gradually. From Fig.13, the amplitude and stress obtained at X, Y and Z axis at 640 N are given below. The amplitude was recorded as 1.3576 mm & 27.042 mm for 50.96 Hz and stress was obtained as 5.6227 & 0.2426 MPa for the same 1 m/s2 acceleration at the X & Y axis respectively.

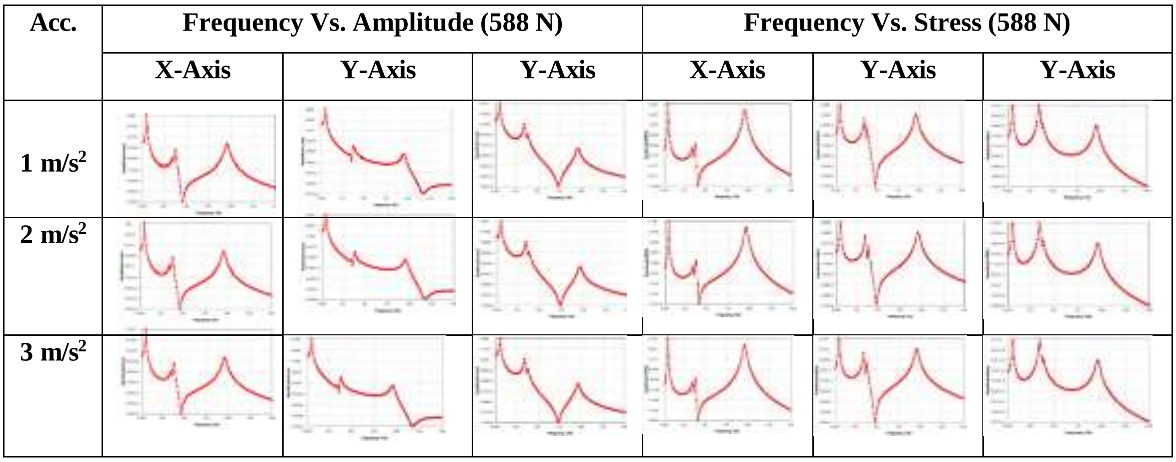


Figure 12. *Frequency Vs. Amplitude and Stress of 588 N load for all axial direction acceleration*

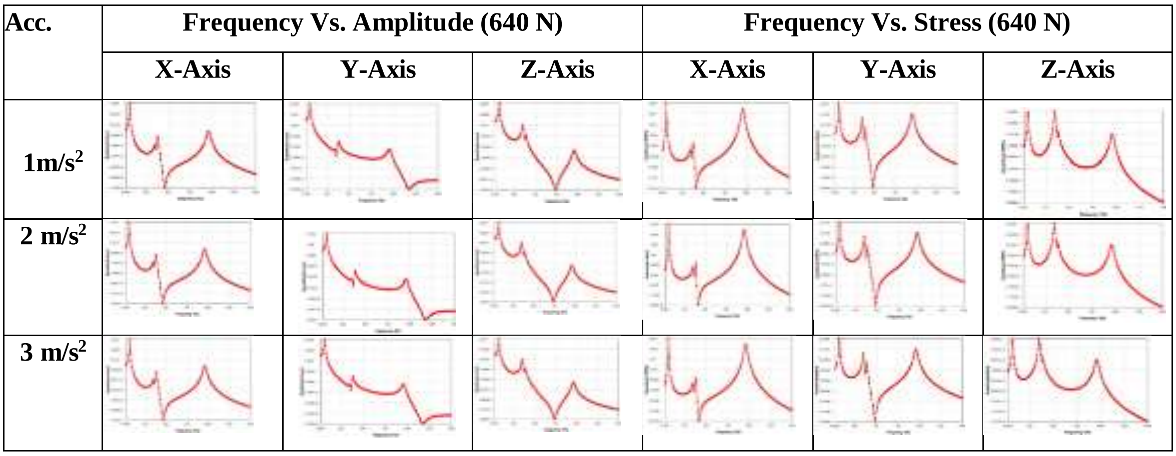


Figure 13. *Frequency Vs. Amplitude and Stress of 640 N load for all axial direction acceleration*

Similarly for Z axis the amplitude and stress was recorded as 2.6529 mm at 60. 96 Hz and 0.1242 MPa at 340.77 Hz respectively. For 2 m/s2 and 3m/s2 acceleration, the results of amplitude and stress at X, Y and Z axis increased gradually as increase in acceleration.

4. Conclusion

The natural frequencies of the first six modes of the femur bone were determined under both free and fixed boundary conditions. This information serves as a guide to mitigate the risk of femur bone fractures. The research focuses on analyzing the vibration characteristics of the femur bone utilizing the FEA method. FEA allows for the determination of natural frequencies, particularly in areas prone to fractures, under external loading conditions. The study proves that vibration analysis helps to detect bone strength and health. Also, it has been proved that it highly depends on its shape.

The research demonstrates that when the excitation frequency aligns with the natural frequency, it can result in bone damage, a phenomenon known as resonance. In such cases, fractures can occur either at the neck region or the bone shaft region of the femur. Harmonic analysis of human femur bone is also done in this work. Furthermore, the study observed amplitude and stress values across a frequency range from 1 Hz to 1500 Hz while varying the acceleration and loads.

Therefore, detecting a resonance frequency and high stress will affect the femur bone. This work also concludes that FEA proves better result than experimental work for cost and time.

Author Statements:

- **Conflict of interest:** The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper
- **Acknowledgement:** The authors declare that they have nobody or no-company to acknowledge.
- **Author contributions:** The authors declare that they have equal right on this paper.
- **Funding information:** The authors declare that there is no funding to be acknowledged.

- **Data availability statement:** The data that support the findings of this study are available on request from the corresponding author. The data are not publicly available due to privacy or ethical restrictions.

References

[1] R. Vijayakumar and M. Madheswaran, (2017). Modal analysis of femur bone using finite element method for healthcare system. *Conference on Emerging Devices and Smart Systems (ICEDSS), Mallasamudram, India,* 2017, pp. 224-228.

[2] Priyadarshi Biplab Kumar, Dayal R. Parhi. (2019). Vibrational Characteristics and Stress Analysis in a Human Femur Bone, *Materials Today: Proceedings*. 4(9):10084-10087. https://doi.org/10.1016/j.matpr.2017.06.325

[3] Balaji D. Kshirsagar, S. Chakradhar Goud, Subim N. Khan. (2020). Vibration analysis of femur bone by using consistent mass matrices and fast fourier transform analyzer, *Materials Today: Proceedings.* 26 (2): 2254-2259.

[4] Majid Mirzaei, Maziyar Keshavarzian, Vahid Naeini. (2020). Analysis of Strength and Failure Pattern of Human Proximal Femur Using Quantitative Computed Tomography (Qct)-Based Finite Element Method, *Bone.* 64:108–114.

[5] Newcombe L, Dewar M, Blunn GW, Fromme P. (2013). Effect of Amputation Level on The Stress Transferred to The Femur by an artificial Limb Directly Attached to the Bone. *Medical Engineering and Physics.*35:1744– 1753.

[6] Sandeep Kumar Parashara, Jai Kumar Sharma. (2016). A Review on Application of Finite Element Modeling in Bone Biomechanics, *Perspectives in Science,* 9: 696-698.

[7] Pelker RR, Saha S. (1983). Stress wave propagation in bone. *J Biomech.* 16(7):481-9.

[8] A.T.M. Phillips. (2009) The Femur as A Musculo-Skeletal Construct: A Free Boundary Condition Modelling Approach, *Medical Engineering & Physics,* 31:673–680.

[9] Pise UV, Bhatt AD, Srivastava RK, Warkedkar R.(2009). A B-spline based heterogeneous modeling and analysis of proximal femur with graded element. *J Biomech.* 42(12):1981-8.

[10] Yuichi Watanabe, Naoto Shiba, Shigeaki Matsuo, Fujio Higuchi, Yoshihiko Tagawa, Akio Inoue. (2000). Biomechanical study of the resurfacing hip arthroplasty: Finite element analysis of the femoral component, *The Journal of Arthroplasty,*15(4):505-511.

[11] Shahar R, Banks-Sills L, Eliasy R. Stress and strain distribution in the intact canine femur: finite element analysis. *(2003). Med Eng Phys.*25(5):387-95.

[12] Sutter EG, Mears SC, Belkoff SM. (2010). A biomechanical evaluation of femoroplasty under simulated fall conditions. *J Orthop Trauma.* 24(2):95-9.

[13] Trabelsi N, Yosibash Z, Wutte C, Augat P, Eberle S. (2011). Patient-specific finite element analysis of the human femur--a double-blinded biomechanical validation. *J Biomech.*44(9):1666-72.

[14] Vaishali Chaudhry, Anubhav Kumar, Shilpa N, Shwetank Avikal, Nithin Kumar K.C. (2019). Modal Analysis of Femur Bone to Find out the Modal Frequencies of Different Bone Implant Materials, *International Journal of Engineering and Advanced Technology*. 8(4): 65-69.

[15] Azra Alizad, Matthew Walch & James F. Greenleaf. (2006).Vibrational Characteristics of Bone Fracture and Fracture Repair: Application to Exercised Rat Femur, *Journal of Biomechanical Engineering,*128 (3):300-308.

[16] V. Pattijna, C. Van Lierdeb, G. Van der Perrea, I. Naertc, J. Vander Slotena, (2006). The resonance frequencies and mode shapes of dental implants: Rigid body behaviour versus bending behaviour. A numerical approach, *Journal of Biomechanics,*39: 939–947.

[17] Mujtaba Almudhaffar, Ameen A. Nassar and Hani A. H. A. Kareem. (2014). Vibration of bones: a case study on human femur, *Journal for Engineering Sciences,*14 (2):229-239.

[18] R. Huiskes and E. Y. S. Chao. (1983). A survey of finite element analysis in orthopedic biomechanics: the first decade, *J. Biomech.,* 16 (6): 385–409.

[19] V. C. Mow, G. A. Ateshian, and R. L. Spilker. (1993). Biomechanics of diarthrodial joints: a review of twenty years of progress. *J. Biomech. Eng.* 115 (4): 460.

[20] Thomas AM, Luo DZ, Dunn JW. (1991). Response of human femur to mechanical vibration. *J Biomed Eng,* 13(1):58–60.

[21] Grassi L, Schileo E, Taddei F, Zani L, Juszczyk M, Cristofolini L, Viceconti M. (2012). Accuracy of finite element predictions in sideways load configurations for the proximal human femur. *J Biomech* ,45(2): 394– 399.

[22] Keyak JH, Rossi SA, Jones KA, Les CM. (2001). Skinner HB, Prediction of fracture location in the proximal femur using finite element models. *Med Eng Phys*, 23(9): 657–664.

[23] Haider IT, Speirs AD, Frei H. (2013). Effect of boundary conditions, impact loading and hydraulic stiffening on femoral fracture strength. *J Biomech,* 46(13): 2115–2121.

[24] Kauraw, V., Chaupal, P. & Rajendran, P. (2023). Vibration analysis of human body under seating posture: an automobile application. *J Braz. Soc. Mech. Sci. Eng.* 45: 200.

[25] Krishna K, Hegde S, G T M, Shenoy B S. (2024). Whole body vibration and rider comfort determination of an electric two-wheeler test rig. *F1000Res.* 12:559.

[26] Mathukumar S, Nagarajan V, Radhakrishnan A. (2019). Analysis and validation of femur bone data using finite element method under static load condition. Proceedings of the Institution of Mechanical Engineers, Part C: *Journal of Mechanical Engineering Science.* 233(16): 5547-5555.

[27] Ashwani Kumar, Himanshu jaiswal, Tarun Garg, Pravin P. Patil. (2014). Free Vibration Modes

Analysis of Femur Bone Fracture Using Varying Boundary Conditions based on FEA, *Procedia Materials Science,* 6: 1593-1599.

[28] P.K. Bharadwaj, R. Prakash. (2021). Modal and harmonic analyses of the Indian male human body subject under semi-supine posture, *Materials Today: Proceedings,* 46(19): 10085-10092.

[29] Rohit Kshirsagar, R Prakash. (2021). Prediction of corrosion-based damages in turbine blades using modal and harmonic analyses, *Materials Today: Proceedings,* 46(19): 10093-10101.

[30] Maheshkumar V. Jadhav, V. R. Gambhire, G. S. Kamble. (2015). Experimental analysis of stresses in real (preserved) intact proximal human femur (thigh) bone under static load. *International Journal of Advanced Technology in Engineering and Science*, 03(1): 1179-1187.

[31] Avi Raj Manral, Narendra Gariya, K.C. Nithin Kumar. (2020). Material optimization for femur bone implants based on vibration analysis. *Materials Today: Proceedings*, 28(4): 2393-2399.