E-ISSN 2581 – 7957
P-ISSN 2277 – 3916

CVR Journal of Science and Technology, Volume 26, June 2024
DOI: 10.32377/cvrjst2620

Biomechanical Analysis of a Femur Bone

C. Sai Kiran[1], M. Siva Sai[2] and P. Surya Teja[3]

[1]Asst. Professor, CVR College of Engineering/Mechanical Engg. Department, Hyderabad, India
Email: csaikiran001@gmail.com
[2] UG Scholar, CVR College of Engineering/Mechanical Engg. Department, Hyderabad, India
Email: avisias1971@gmail.com
[3]UG Scholar, CVR College of Engineering/Mechanical Engg. Department, Hyderabad, India
Email: suryateja43623@gmail.com

Abstract: **In this paper, a CAD model of human femur bone was generated and was analysed which is useful for medical industry for implementing the correct size and correct plate that can withstand the stresses developed by the user. The purpose of the work is to find a suitable material which can withstand the maximum stress made by the user. In this paper internal fixation is analysed. The image of femur bone is generated by using the 3D slicer where CT scan data in form of DICOM files is sent to the software to generate 3D model and the Bone plate, screws are modelled by using SolidWorks, the dimensions of the plate and screws are taken according to the generated femur bone. The function of the bone was analyzed for four different materials i.e. Nylon, PMMA, SS316L, TI-6Al-4V by using Ansys software and when the body is physically active. Total deformation and von-misses stress analysis help to choose the material which can withstand the max stress under different activities.**

Index Terms: **Femur, 3D Slicer, SolidWorks, Ansys, Von-mises stress.**

I. INTRODUCTION

The femur bone, which is commonly known as the thigh bone is the longest and strongest bone in the human body. Femur plays a crucial role in supporting body weight and facilitating movement of the human body.

S. Kirthana et al. [1]. In this paper, biomechanical analysis is performed for the right femur bone, also known as the thigh bone. Under static structural analysis, the authors conducted equivalent (von-Mises) stress, total deformation and factor of safety analysis were conducted. The boundary conditions applied to the femur bone included considering the bottom of the femur bone as a fixed support, while the other end is free in all directions, with force applied on the circumferential bone. They utilized five different materials for the analysis. From their analysis, they concluded that titanium is the most suitable material among the others. Additionally, they considered the cost factor, suggesting PMMA and PEEK as potential choices for the plate and screw material. *P.S.R. Senthil Maharaj* et al. [2]. In this paper, numerical analysis is performed on fractured femur bone with prosthetic plates. The prosthetic bone plate was modelled using SolidWorks software and the femur bone is modelled using mimics software. Then the prosthetic plate was attached to the femur using SolidWorks software. Equivalent stress and directional deformation are performed for five different materials. From the results, titanium was found to have the lowest stress out of the five materials. *K.C. Nithin Kumar* et al. [3]. In this paper, biomechanical analysis of femur bone using Finite Element technique is conducted. The CAD model of femur bone was modelled from MRI/CT scan data using ITK-snap software and pre-processing & post-processing operations are performed using HYPERWORKS software, whereas the solver is NASTRAN 10.0 software. The analysis is conducted using three materials - natural bone material, AZ31, CP Ti. In this study maximum stress and maximum displacement is done for jumping & walking conditions and compared the AZ31, CP Ti with natural bone. From the results it is found out that AZ31 is best suited material for bone implants because of its low stress & displacement compared to natural bone and CP Ti. *S.G. Aftab* et al. [4]. In this paper, biomechanical analysis of femur bone using FEA method is conducted. The 3D model of femur bone was created by using MRI scanning technique and then the model was converted into IGES format for analysis in ANSYS software. Equivalent stress is applied to natural material of femur bone & other three materials (PMMA, Al2O3, Nylon 66) understanding, walking, jumping, and running conditions. After the analysis, the material with low stress compared to natural femur bone for different conditions is selected and this process is considered for the other conditions. *S. Mathukumar, VA Nagarajan* et al. [5]. In this paper, the researchers used the modelling technique to get the 3D model of femur bone from the CT scan data and with the help of computational method i.e. FEA method. Stress and strain analysis is performed on the femur bone under the loads of 490N, 540N, 588N and 640N and the strain gauges were positioned at designated locations labelled 1, 2, 3 and 4. The bottom of the femur bone is considered as fixed support and the opposite end of the femur bone remained fixed in all degrees of freedom. Higher stress values were observed at the neck inferior, and neck superior positions compared to the shaft lateral and shaft medial positions, additionally analysis highlighted this increased weight led to a greater total displacement, with the neck side of the femur exhibiting the highest strain. The research findings propose that conducting experiments directly on femur bones may become unnecessary, given the higher accuracy and reliability of results obtained through finite element (FE) analysis. This suggests that FE analysis offers a possible alternative that could potentially replace or reduce the need for physical experimentation on real femur bones. *P Kishore* et al. [6]. In this paper the researchers conducted an analysis of the femur bone, focusing on evaluating normal stress and total deformation. The CAD model of femur bone

Received on 10-03-2024, Revised on 09-04-2024, Accepted on 09-05-2024.

is modelled by using the CT scan data, which comprises DICOM files consisting of thousands of bone images. This are further imported to 3D slicer to get 3D model of the femur bone and this model is saved in the form of .stl format, it is imported into blender where if the model has any unwanted parts here, they are removed and saved and this .stl format is imported to CATIA V5 where it is converted into .stp format for analysis. The analysis is performed under three different loads 270N, 400N and 650N. The results of their analysis indicate varying levels of normal stress and total deformation experienced by the femur bone under these loads. Additionally, they observed that the femur neck exhibited maximum stress under the applied loads, with higher loads potentially leading to femur neck fractures and associated patient discomfort. *Pravat Kumar Satapathy* et al. [7]. In this paper the authors performed an FE analysis on the femur bone. The bone modelled by using the data from the CT scan and the material properties are assigned using Mimics software. The implant plate materials they considered are titanium, FG material is a homogenous material which consist of different types of materials. Results of the analysis revealed the behaviour of the femur bone under various loading conditions. Deformation and stress distribution were evaluated for both the titanium and FG implant materials. It was observed that the FG material exhibited different mechanical responses compared to titanium, suggesting potential advantages in terms of strength and stability. The results states that the plate material with FG demonstrated good performance compared to the other materials. *Albert E. Yousif and Mustafa Y. Aziz* et al. [8]. has performed biomechanical analysis on femur bone while sitting and going upstairs. The femur bone model by using CT scan data is imported to mimics software in the form of DICOM files. The authors applied fixed constraints at both the distal and proximal ends of the femur this constraint aids the calculation of normal stresses during activities such as going upstairs and sitting down. This study highlights the significance of estimated stress profiles in injury prevention, prosthesis development and the design of more durable implants. Moreover, these stress values have implications for sports medicine, fitness-related research and various biomechanical applications. *Yegireddi Shireesha* et al. [9]. In this paper the authors conducted an FEA analysis on femur bone under equivalents stress and total deformation. The femur bone model is generated by using CT scan data. Further, this model is imported to Ansys software for equivalent stress and total deformation to the femur bone. The analysis is conducted under static loading conditions using ANSYS software, evaluated stresses formed in different femur implant materials, including structural steel and Ti-6Al-4V. These analyses are performed under three different loads 550kg, 650kg & 750kg. They Compared the performance of the two implant materials, here the Ti-6Al-4V demonstrated less deformation under static load conditions. This material's low density and excellent biocompatible make this material as an ideal choice for surgical implants. *Raji Nareliya And Veerendra Kumar.* [10]. In this paper FEA analysis is applied on the femur bone. The femur bone model is generated by importing CT scan data into mimics software from there it is imported to workbench in form of .stp file. They considered the bottom of the femur bone in medical field this bottom (medial and lateral condyle) of femur is known as condyle as the fixed support and applied a load of 750N to the head of the femur bone. Results indicated that higher weight led to increased total displacement, with maximum total deformation observed at the femur's head and minimum at the lower end. Maximum principal stress occurred at the middle section of the femur, accompanied by an equivalent (Von Misses) stress and fatigue life was estimated, and a constant factor of safety of 15 was maintained throughout the femur. Ultimately this research contributes valuable insights into the biomechanical behaviour of the femur bone.

From the literature we concluded that researchers generally suggest titanium stands out as the most biocompatible material. However regarding material choices and loading conditions make it challenging for drawing definitive conclusions, while some studies briefly mention cost considerations in material selection, there is a lack of comprehensive analysis regarding the cost effectiveness of different materials, as FEA shows promise in predicting bone behaviour, the lack of uniformity in how models are constructed and boundaries are defined can lead to contradictions in results and hinder the comparability of studies.

From the studies we explored a wide range of materials for femur bone analysis, including titanium, PMMA, SS316l, Nylon 66, and composite materials.

- Titanium emerged as a favourable material due to its low stress levels and favourable mechanical properties.
- More research needed to assess materials for durability, compatibility with cost-effectiveness in clinical use.

II. METHODOLOGY

A. Generating CAD Model of Human Femur Bone

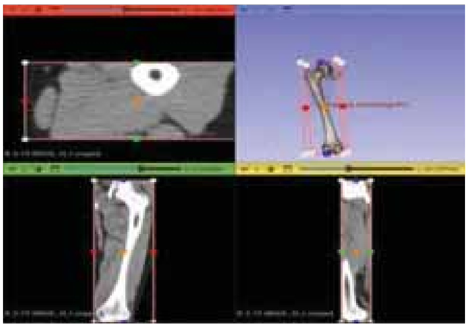


Figure 1. CAD Model Generated Using 3D Slicer Software

The CAD model of bone is generated by importing the DICOM files which are generated from patients CT or MRI scan data and this DICOM files are imported to generate the CAD model of the bone as shown in Fig. 1.

The modelling of plate and screws are done using the SolidWorks software. The dimension of the plate is taken by considering the height of the femur bone and its width. The dimensions of the screw are also taken by considering the CAD model of bone.

B. Refinement of Femur Bone Model Using Meshmixer

Here, after creating a CAD model we import it to mesh mixer Fig. 2 shows the CAD model of bone after trimming of extra parts and refinement of bone model.

The dimensions of Bone:

Figure 2. CAD Model Generated Using 3D Slicer Software

Height of femur bone = 426.3mm

Width of femur bone (shaft) = 30mm (Approx)

Width of condyle (bottom of bone) = 77.58mm

C. Modelling of Fixation Plate

The dimensions of the fixation plate, determined based on the height width of the bone shaft, were modelled using solid works. The dimensions are as follows:

Length of plate: 5mm

Breadth of plate: 16mm

Height of plate: 140mm

D. Modelling of Screw

The shape of screw or threads can significantly impact the stability of plate, ensuring secure fixation to the bone surface. Therefore, careful consideration should be given to the design of screws, modelled using SolidWorks. In this research paper, the dimensions were selected deliberately rather than randomly.

Shape of drive = Hex

Total screw length = 27.7mm

Shank of screw = 5mm

E. Assemble of Fixation Plate to Femur Bone

The internal fixation is assembled to the femur bone using SolidWorks. Fig. 3 shows the assembled model of femur bone.

F. Materials for The Bone, Plate and Screw

The plate and screw materials must be biocompatible to minimize the risk of harmful reactions or rejection by the body. The plate and screw materials should possess adequate strength and durability to withstand the mechanical forces exerted on them during daily activities. For analysis Nylon, PMMA (polymethyl methacrylate), SS316L (stainless steel alloy with a low carbon) and Ti-6Al-4V has been selected. The material properties are referred from Table I.

Nylon 6/6:

Figure 3. Assembled Model of Femur Bone

- Advantages: High strength-to-weight ratio, corrosion resistance.
- Limitations: Chemical sensitivity, creep resistance and moisture absorption.

PMMA (Polymethyl Methacrylate):

- Advantages: Transparency and biocompatibility.
- Limitations: Brittleness and scratch sensitivity

SS316L (Stainless Steel 316L):

- Advantages: Corrosion resistance, high strength and biocompatibility.
- Limitations: Cost, weight and machinability.

Ti-6Al-4V (Titanium Alloy):

- Advantages: Biocompatibility, corrosion resistance and high strength to weight ratio.
- Limitations: Cost, difficulty in machining and low modulus

TABLE I.
MECHANICAL PROPERTIES OF BIOMATERIALS FOR FEMUR, PLATE AND SCREW

Bone & prosthetic plate materials	Young's modulus (GPa)	Density (gm/cc)	Poisson Ratio (γ)
Femur Bone	1.8	18	0.33
Nylon 6/6	3.72	300	0.21
PMMA	1.18	220	0.2
SS316L	7.75	193	0.31
Ti-6Al-4V	120	4.5	0.32

These materials were chosen based on their properties, biocompatibility and their utility in various medical fields.

G. Load Calculations

These load values are calculated for four activities:

For standing activities:

Total weight of human (m) = 75 kg

Weight of one leg (assuming equal distribution) = $\frac{75\,kg}{2}$

$= 37.5\,kg$

Using the formula,

$$f = m \times g. \tag{1}$$

$where, g = 9.81 \frac{m}{s^2}$

$f = 37.5 \ \times 9.81$

$= \ 367.8\,N$

$\approx 370\,N$

For jumping activity:

Step 1: Calculate the new acceleration (a_{jump}) using the formula for vertical displacement:

Jump (h) = 0.1m

Jump duration (t) = 0.1sec

$$h = \frac{1}{2} \times a_{\text{jump}} \times t^2 \tag{2}$$

Submitting the values:

$$0.1 = \frac{1}{2} \times a_{\text{jump}} \times (0.1)^2$$

$$0.1 = \frac{1}{2} \times a_{\text{jump}} \times 0.01$$

$$10 = 0.005 \times a_{\text{jump}}$$

$$a_{\text{jump}} = 20\,\text{m/s}^2$$

Step 2: Calculating the force exerted by one leg F :

$$F = m \times a_{\text{jump}} \tag{3}$$

$m \ = \ 75\,kg$

Submitting the values:

$= 75 \times 20 = 1500\,\text{N}$

$$F = \frac{1500}{2}$$

$= 750\,N$

Therefore, the force exerted by one leg during the jump is 750 N.

For Walking Activity:

An average person exerts 3 times of their body weight. so,

$= 3 \ \times \ 37.5$

$= 1103.62\,N$

$\approx 1100\,N$

For Running Condition:

An average person exerts three and half times of his body weight based on this if a person weighs 75 kg then,
75 x 3.5 x 9.81 = 2571.98N

for one leg = $\frac{2572}{2}$

=1286N

≈ 1300

III. FINITE ELEMENT ANALYSIS

The 3D CAD model is saved in .stp format for the assembly of the plate and screws with femur bone, as well as for analysis purposes. The assembly process is performed in SolidWorks software as shown in Fig. 2, while the analysis is performed in ANSYS Workbench 2022 R1 software.

A. Meshing

Figure 5. Meshing of a Femur Bone

Meshing is performed to divide the geometry into smaller and simpler elements, allowing for the approximation of complex shapes and the numerical solution of equations governing the behaviour of the system. Fig. 4, Shows the generated mesh for the femur bone.

B. Applying Loads.

Different loads are taken for different activities, such as standing, jumping, walking and running.

Figure 4. Applying loads

The load values are taken respectively 370 N, 750 N, 1100 N and 1300 N, these loads are applied on the head of the bone as shown in Fig. 5.

As shown in Fig. 6, at the bottom of the femur bone, fixed support is applied to withstand the pressure acting on the femur bone replicating the real-life activity.

Figure 6. Applying Fixed Support

C. Applying Moment

In biomechanical analysis, moment represent rotational forces on structures like the femur bone. During standing, a small non-zero moment, like 0.01 N-mm, simulates stability. Dynamic activities like walking or running require higher moments up to 1000 N-mm to mimic the forces experienced by the femur. Fig. 7. Shows about the moment applied for jumping, walking and running activity and Fig. 8. Shows about moment applied for standing activity.

Figure 7. Moment For Standing Activity

Figure 8. Moment For Jumping, Walking and Running Activity

IV. RESULTS AND DISCUSSIONS

After applying fixed support and pressure on the bone, structural analysis is performed for four different materials (Nylon, PMMA, SS316l and Ti-6Al-4V) in Ansys workbench software to determine the equivalent (von-mises) stress and the total deformation for four activities/conditions (i.e. standing, jumping, walking and running). The following results were observed in the analysis.

A. Total Deformation of Femur Bone

Total deformation is applied for four materials i.e. Nylon, PMMA, SS316l and Ti-6Al-4V under different activities like standing, jumping, walking and running conditions.

1. Total deformation of Nylon material for four activities

Under the standing condition with a pressure of 370 N, the nylon material exhibits a maximum deformation of 1.0489 mm as shown in Fig. 9.

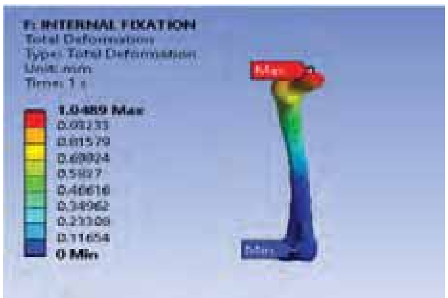


Figure 9. Standing Activity/Condition

As shown in Fig. 9, the total deformation is applied to other activities and the results are given in the below Table II.

TABLE II.
TOTAL DEFORMATION FOR NYLON MATERIAL

Activity/Conditions	Total Deformation [mm]	
	Max	Min
Standing (370N)	1.0489	0
Jumping (750N)	2.1261	0
Walking (1100N)	3.1183	0
Running (1300N)	3.6852	0

2. Total deformation of PMMA material for four activities

Now, the total deformation for the PMMA material for four activities is given in Table III.

TABLE III.
TOTAL DEFORMATION FOR PMMA MATERIAL

Activity/Conditions	Total Deformation [mm]	
	Max	Min
Standing (370N)	1.0827	0
Jumping (750N)	2.1902	0
Walking (1100N)	3.2137	0
Running (1300N)	3.7989	0

3. Total deformation of SS316l material for four activities

Now, the total deformation for the SS316L material for four activities is given in Table IV.

TABLE IV.
TOTAL DEFORMATION FOR SS316L MATERIAL

Activity/Conditions	Total Deformation [mm]	
	Max	Min
Standing (370N)	1.0968	0
Jumping (750N)	2.2187	0
Walking (1100N)	3.2555	0
Running (1300N)	3.8483	0

4. Total deformation of Ti-6Al-4V material for four activities

Now, the total deformation for the Ti-6Al-4V material for four activities is given in Table V.

TABLE V.
TOTAL DEFORMATION FOR TI-6AL-4V MATERIAL

Activity/Conditions	Total Deformation [mm]	
	Max	Min
Standing (370N)	1.4388	0
Jumping (750N)	2.9119	0
Walking (1100N)	4.2729	0
Running (1300N)	5.0506	0

B. Equivalent Stress of Femur Bone

After performing total deformation, the equivalent stress is performed on the femur bone under different activities i.e. standing, jumping, walking and running conditions.

1. Equivalent stress of Nylon material for four activities

Under the standing condition with a pressure of 370 N, the nylon material exhibits a maximum stress of 1538 MPa as shown in Fig. 10.

Figure 10. Standing Activity/Condition

As shown in Fig. 10, Von-mises stress is applied to other activities and the results are given in the below Table VI.

TABLE VI.
EQUIVALENT STRESS FOR NYLON MATERIAL

Activity/Conditions	Equivalent stress [MPa]	
	Max	Min
Standing (370N)	1538	2.39E-05
Jumping (750N)	3117.7	4.843E-05
Walking (1100N)	4572.6	7.103E-05
Running (1300N)	5404	8.42E-05

2. Von-mises stress of PMMA material for four activities

Now, the total deformation for the PMMA material for four activities is given in Table VII.

TABLE VII.
EQUIVALENT STRESS FOR PMMA MATERIAL

Activity/Conditions	Equivalent stress [MPa]	
	Max	Min
Standing (370N)	1188.5	1.53E-05
Jumping (750N)	2411	2.93E-05
Walking (1100N)	3533.5	4.37E-05
Running (1300N)	4175.3	5.19E-05

3. Von-mises stress of SS316l material for four activities

Now, the total deformation for the SS316L material for four activities is given in Table VIII.

TABLE VIII.
EQUIVALENT STRESS FOR SS316L MATERIAL

Activity/Conditions	Equivalent stress [MPa]	
	Max	Min
Standing (370N)	1006.7	1.02E-05
Jumping (750N)	2040	2.68E-05
Walking (1100N)	2990.9	4.04E-05
Running (1300N)	3534.5	4.56E-05

4. Von-mises stress of Ti-6Al-4V material for four activities

Now, the total deformation for the Ti-6Al-4V material for four activities is given in Table IX.

TABLE IX.
EQUIVALENT STRESS FOR TI-6AL-4V MATERIAL

Activity/Conditions	Equivalent stress [MPa]	
	Max	Min
Standing (370N)	9.2911	8.00E-08
Jumping (750N)	18.836	1.59E-07
Walking (1100N)	27.624	2.35E-07
Running (1300N)	32.646	2.78E-07

The graph of total deformation for four different materials is observed in above Fig. 11.

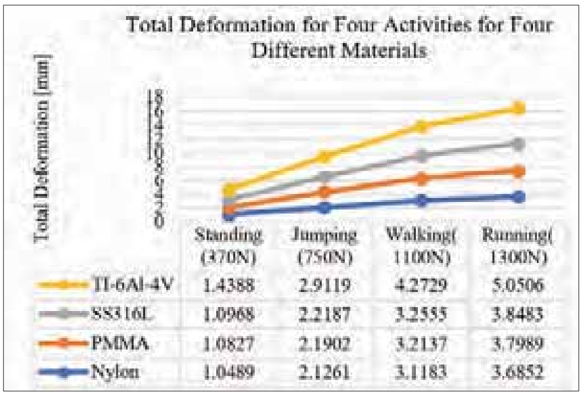
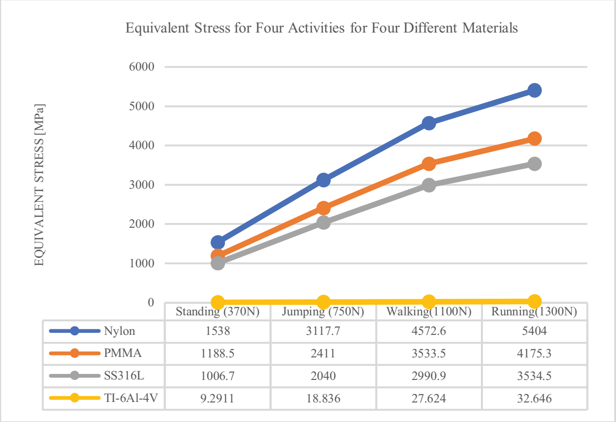


Figure 11. Total Deformation Graph for Different Materials

The graph of Equivalent stress for four different materials is observed in below Fig. 12.



	Standing (370N)	Jumping (750N)	Walking(1100N)	Running(1300N)
Nylon	1538	3117.7	4572.6	5404
PMMA	1188.5	2411	3533.5	4175.3
SS316L	1006.7	2040	2990.9	3534.5
TI-6Al-4V	9.2911	18.836	27.624	32.646

Figure 12. Equivalent stress Graph for Different Materials

V. CONCLUSION

In this paper, the femur model is generated by importing DICOM files into 3D slicer and the analysis is done by using ANSYS workbench 2022 R1 software. The analysis is made for four different materials under different activities i.e. Standing (370 N), Jumping(750 N), Walking(1100N), Running (1300N).

From the results it is observed that the Ti-6Al-4V material demonstrates the lowest equivalent stress values, ranging from 9.2911 MPa for standing to 32.646 MPa for running, indicating its effectiveness in withstanding various loading conditions. After the Ti-6Al-4V, SS316L has the better results compared to PMMA and Nylon. While all four materials are biocompatible the choice of material may vary depending upon specific requirements and if we're considering cost factor into account, TI-6Al-4V is priced approximately Rs 1600 to 2500 per kg, while PMMA price around Rs 135 per kg.

REFERNCES

[1] S. Kirthana, M. Bindhu Supraja , A.S.N Vishwa, Mahalakshmi, "Static structural analysis on femur bone using different plate material," *Materials Today: Proceedings*, Volume 22, Part 4, pp. 2324-2333, 2020.

[2] P.S.R. Senthil Maharaj, R. Maheswaran, A. Vasanthanathan, "Numerical analysis of fractured femur bone with prosthetic bone plates," *Procedia Engineering*, Volume 64, pp. 1242-1251, 2013.

[3] K.C. Nithin Kumar, Narendra Griya, Amir Shaikh, Vaishali Chaudhry, Subhash Chavadaki, "Structural analysis of femur bone to predict the suitable alternative material," *Procedia Manufacturing*, Volume 26, Part 2, pp. 364-368, 2020.

[4] S.G. Aftab, Ahmed Faisal, Hamid Hussain, B. Sreedhara, N. Ramesh Babu, B.A. Praveen, "Structural analysis of human femur bone to select an alternative composite material," *Procedia Manufacturing*, Volume 48, pp. 82-89, 2020

[5] S. Mathukumar, VA Nagarajan and A. Radhakrishnan, "Analysis and validation of femur bone data using finite element method under static load condition," *Proc IMechE art C: J Mechanical Engineering Science 2019*, Volume 233(16), pp. 5547–5555.

[6] P. Kishore, Anjan Kumar Dash, K Dhilip Kumar, "Modelling and analysis of femur bone from CT scan," *International Conference on Advances in Materials Processing and Characterization",* volume 764-012003 (2020).

[7] Pravat Kumar Satapathy, Bamadev Sahoo, Panda L N and Das S, "Finite element analysis of functionally graded bone plate at femur bone fracture site," *IOP Conference Series: Materials Science and Engineering*, Volume 330 (2018) -012027.

[8] Albert E. Yousif and Mustafa Y. Aziz, "Biomechanical analysis of the human femur bone during going upstairs and sitting down," *IOSR Journal of Engineering,* ISSN: 2250-3021 Volume 2, Issue 8 (August 2012), pp.13-19.

[9] Yegireddi Shireesha, Dr S. V. Ramana and P. Govinda Rao, "Modelling and static analysis of femur bone by using different implant materials," *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*, e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 7, Issue 4 (Jul. - Aug. 2013), pp. 82-91.

[10] Raji Nareliya and Veerendra Kumar, "Biomechanical Analysis of Human Femur Bone," *IOSR Journal of Engineering (IOSRJEN)*, ISSN: 2250-3021 Volume 2, Issue 8 (August 2012), pp. 13-19.