CFD ANALYSIS OF FEMUR BONE PLATES

Amitesh Anand[1], Dr. Sohail Bux[2]
[1]MTech Scholar, [2]Professor
[1]Department of Mechanical Engineering, Agnos College of Technology, Bhopal, India
[2]Department of Mechanical Engineering, Agnos College of Technology, Bhopal, India
mtshrj@gmail.com[1]buxsohail@gmail.com[2]

* Corresponding Author: Amitesh Anand

Abstract: *The structural contact and modal analysis of the femur bone is done with a single crack in middle of the shaft; this fractured bone is coupled with the prosthetic plate and screw of same materials. Assembled structured is analyzed by finite element method using transient structural analysis. In the present work a three dimensional CAD model of Femur bone with plate and locking screw is created with the help of CATIA software then imported in ANSYS workbench for further analysis. For contact analysis the contact status is sticking, frictional stress 1798.8 Mpa, contact pressure 948.27 Mpa Penetration 0.00035 mm and the minimum gap due to deformation is 0.013 mm. for the modal analysis the natural frequency range from 5.32Hz to 3973.7Hz.It has been observe from the above conclusion that Ti6Al4V gives the best suitable results and also suggested by doctors and also recommended best suitable biomaterial for femur bone implant.*

Keywords:*Finite Element Analysis, Computerized Tomography, Functionally Graded Material, Selective Laser Melting.*

I. INTRODUCTION

In vertebrates that can walk and leap, the femur bone is the most distal bone of the leg. The femur is the longest and biggest bone in the human body, however it is only the strongest under compression. During typical weight-bearing exercises, the femur bears the majority of the body's weight. Its length is equal to 26% of the individual's height. The femur's body is long, thin, and almost cylindrical in shape. One of the frequent injuries is a femur bone fracture. Because femurs are the strongest, longest, and heaviest bones in the human body, they are a crucial area of research in orthopaedic trauma.

Unlike the majority of the body's tissues, bone tissue has the amazing capacity to regenerate. A shattered bone can renew tissue and restore much of its previous strength if it can be kept together. To retain the bone in place for serious fractures, bone plates are surgically placed. Design, material choice, and biocompatibility are the three key factors to take into account while creating bone plates. The bone plate needs to be sturdy enough to withstand the weight that would ordinarily be placed on the bone while it heals. Additionally, the plate needs to be as rigid as the bone to which it is connected. The implant must not be harmful to the body and cannot set off an inflammatory reaction.

The stiffness of the bone plate is important because the stress shielding will increase with the difference in stiffness. Stress shielding is the phenomenon in which the implant bears most of the load normally placed on the bone. Although this is favorable while the bone is weak, as the bone heals and regains strength, if the bone plate does not allow the bone to carry an increasing load, there will be a reduction of bone mass and final regained strength. From the beginning of their use, material selection was the limiting factor to their success. As technology advanced so did the materials.

Because a difference in stiffness will result in increased stress shielding, the bone plate's stiffness is crucial. The phenomenon known as "stress shielding" occurs when the implant carries the majority of the force typically applied to the bone. Although this is advantageous while the bone is weak, if the bone plate does not let the bone to take an increasing load as the bone heals and regains strength, there will be a loss in bone mass and eventual restored strength. The choice of material has been the key to their failure since they were first used. Materials evolved together with technology.

II. LITERATURE REVIEW

Ahirwar et al.[1] To assess the interface deformation, stress, and strain created at the bone-bioimplant contact, finite element analysis was used. The FEA results showed that when the naturally anisotropic condition of the human femur was well taken into account, the interface deformation and stress for a bone-bioimplant assembly were dramatically reduced. Fu et al. [2] proposed Computational fluid dynamics to examine fluid flow in the scaffold. The simulation findings demonstrated that when the scaffold's pore size increases, its permeability rises and its FSS falls. Additionally, the FSS was dispersed in stages across the cell surface. Li et al.[3] Investigate a modelling technique for biomimetic porous bone scaffolds for biological three-dimensional printing that is practical and efficient and is based on a replication of the histomorphological features of human vertebral cancellous bone. The bionic modelling design approach put forward in this study takes into account the biological properties of the cancellous tissue in the vertebrae and performs imitation and design of images of two-dimensional slices that resemble vertebrae [4]. The method of computational fluid simulation was used to characterise the flow through porous materials. **Arjunan et al.[5]**aims to conduct a comprehensive analysis into the stiffness, strength, permeability, and stress concentration of six scaffold topologies with a porosity range of 68.46–90.98%. The FE model was verified and further developed to investigate the impact of stress concentration on the scaffolds' stiffness and strength. The findings shown that the pore shape can affect the Ti6Al4V bone scaffolds' permeability, stiffness, strength, and stress concentration factor. **Wang et al.[6]** The study's findings on the performance of four groups of various honeycomb designs demonstrate the viability and potential of using honeycomb structures in the creation of biomimetic bone scaffolds. The yield strength (88-146 MPa) was much higher than that of the femoral neck, and the elastic modulus (1.6-3 GPa) of the four scaffolds matched the elastic modulus of human cancellous bone (0.1-4.5 GPa) in terms of static characteristics (0.56–3.71 MPa). With the exception of the SU scaffold's high permeability, the permeability of the other three sets of scaffolds (1.5*108-4.8*108 m2) is comparable to that of cancellous bone (0.5*108-5.0*108 m2). The four groups of scaffolds' wall shear stresses, which ranged from 2.8 to 42.8 MPa, might encourage cell deposition of mineralized extracellular matrix (0.55–30 MPa) in the 3D scaffold, which was helpful for bone tissue regeneration. **Pravat Kumar Satapathy et. al. [7]**This research examines a fragmented Femur bone with a functionally graded bone plate. The fracture fixation plate in this case is made of Functionally Graded Material [8]–[11]. The functionally graded bone plate is thought to be made up of many layers of homogenous elements. **Koris J. et. al. [12]**Scenario 1 had a gap between the end of the femur and the implant collar, scenario 2 had no gap but no bone attachment into the collar, and scenario 3 had no gap but no bone attachment into the collar. Scenario 1 had a gap between the end of the femur and the implant collar, scenario 2 involved no gap but no bone attachment into the collar, and scenario 3 involved bone development down the length of the collar with attachment. In scenario 1, the greatest stress observed in the implant was 3104.2Mpa, compared to 1054.4Mpa in scenario 2 and 321.2Mpa .S. Karuppudaiyan et. al. [13] The biomechanical structural behaviour of tibia bone was investigated using a 3D model created utilising a reverse engineering method. The primary goal of this research is to create a subject-specific finite element model of the tibia bone using the Reverse Engineering approach, as well as to investigate the biomechanical structural behaviour and fracture risk of the tibia bone under physiological loading circumstances [14]–[18]. These mechanical characteristics are employed as an input for finite element analysis since the properties of bone vary depending on the mix of mineral content and protein content. The findings of finite element analysis were compared to the literature on FEM models built using CT scans. **Feifei Jiang et. al. [19]**In response to knee loading, FEA indicated that the maxima of von Mises stress, a predictor of fracture yielding, and the third major compressive stress would be greater in the placebo-treated femur than the drug-treated femur. This FE investigation lends credence to the hypothesis that mechanical weakening of the femur was detected in tumor-invaded trabecular bone, and chemical compounds such as PD407824 may possibly aid in avoiding bone loss and bone fracture. **Evandro Pereira Palacio et. al. [20]** Synthetic adhesives are employed in a variety of medical specialities, including surgery; however, data on their application in orthopaedic treatment are limited. Biomechanical and histomorphometric assessments were done at three separate times , as well as a clinical examination conducted weekly by measuring the animals' body mass. The bone callus area on the humerus in both groups was similar . There were no significant differences in inflammatory cells , osteoblasts , or osteoclasts in either group.

There are following objective of the present work.

1. To perform transient structural analysis of femur bone at different material (like SS316L, Ti6A14V).
2. To perform transient structural analysis on New Design of bone plate.
3. To perform contact analysis to find out contact stress, frictional stress, penetration and gap between screw bone and bone plate.
4. To perform modal analysis to find out natural frequencies.

III. Methodology

Flowchart for Finite element analysis

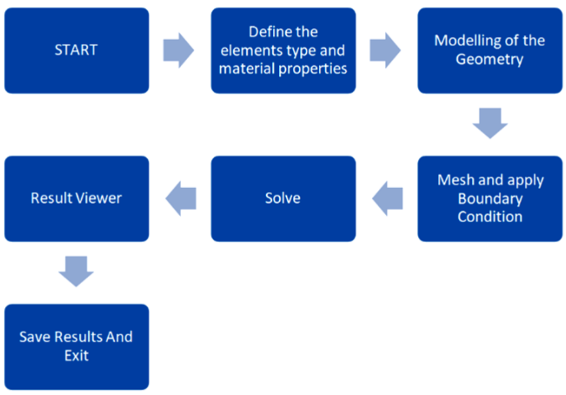

Figure 1: Flowchart for Finite element analysis

Transient structural analysis
CAD Geometry with SS316L:
In the present work a three dimensional CAD model of Femur bone with plate and locking screw is created with the help of CATIA software then imported in ANSYS workbench for further analysis. A three dimensional view of Femur bone with plate and locking screw is shown in figure 2.

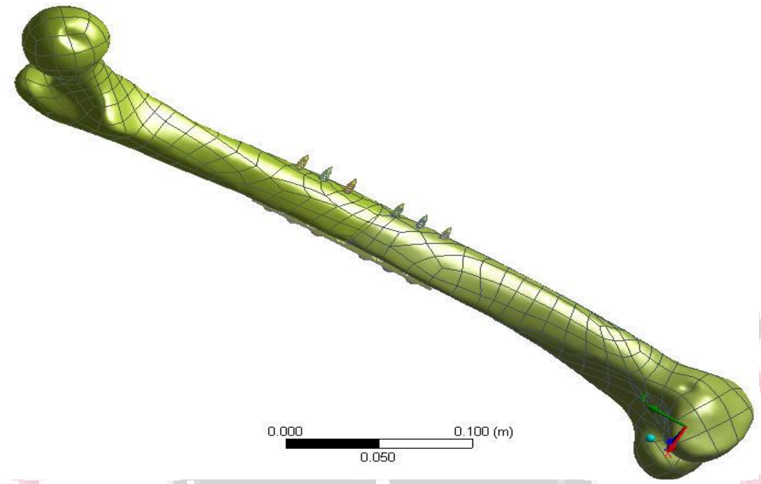

Figure 2: Three dimensional CAD model For with SS316L

Meshing:

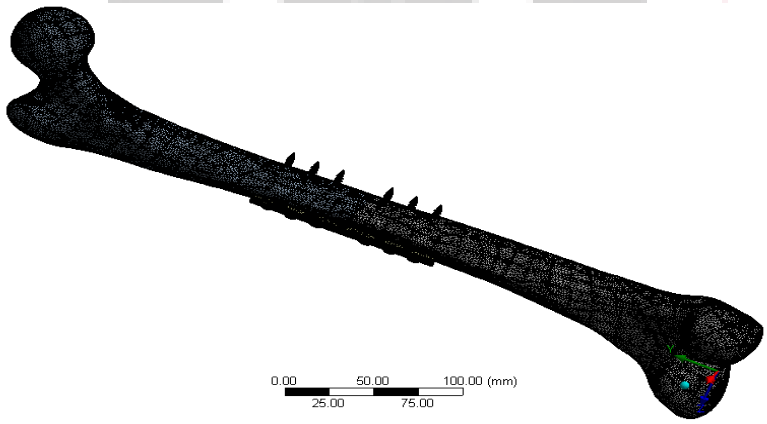

Figure 3: Meshing: Total No. of Nodes: 937238 & Total No. elements: 568664

After completing the CAD geometry of Femur bone with plate and locking screw is imported in ANSYS workbench for further Transient structural analysis and the next step is meshing. Meshing is a critical operation in finite element analysis in this process the CAD geometry is divided into large numbers of small pieces. The small pieces are called mesh. The mesh created for Femur bone with plate and locking screw in this work is shown in figure 3. The total Node is generated 937238 and total No. of Elements is 568664.

Table 1: Material property

Bone & Prosthetic plate materials	Density Kg/m^3	Young's Modulus E (Gpa)	Poission Ration	Ultimate Tensile Strength (Mpa)	Ultimate Compressive Strength (Mpa)
Femur cortical bone	1750	16.7	0.3	43.44+_ 3.62	115.29+_12.94
SS316L	7750	193	0.31	485	570
Titanium alloy	4500	121	0.37	1250	1100

Boundary condition for Structural Analysis:
1. Fixed support is provided at the lower part of bone where it is constrained in all degree of freedom.
2. A vertical force of 750N is applied on the upper part of bone.
3. For the transient structural analysis total time of analysis is of one second.
4. Auto time stepping set as off and the time step is divided in 0.1 second.
5. Since it is a rigid structure and does have the property of large deformation the large deflection option set off.
6. Nonlinear controls leave as default or program control and in the output controls stress & strain is expected as results.
7. APDL solver is used to solve this transient structural analysis.

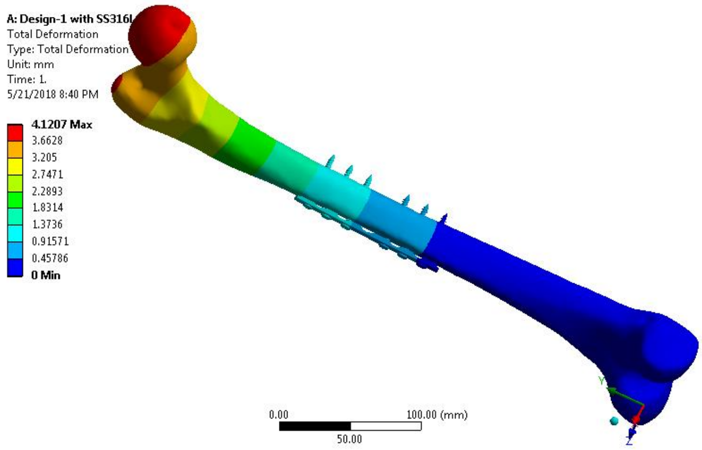

Figure 4: Total Deformation for SS316L

After performing the transient structural analysis the value of total deformation is observe as 4.1207 mm and shown in figure 4 with different color contours.

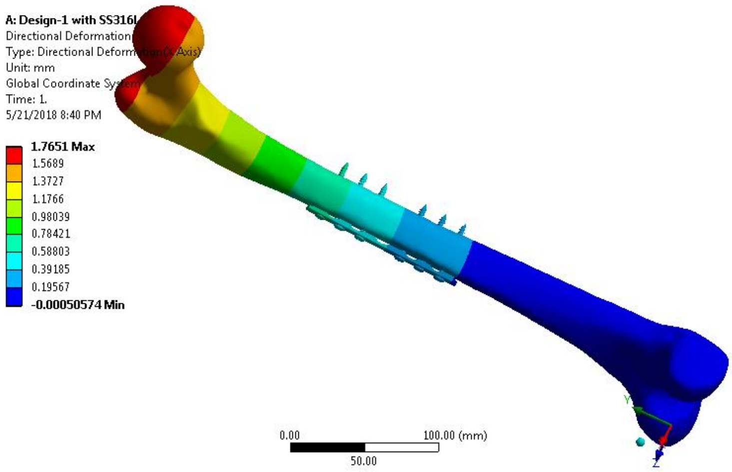

Figure 5: Directional Deformation for SS316L

After performing the transient structural analysis the value of Directional deformation is observe as 1.7651 mm and shown in figure 5 with different color contours.

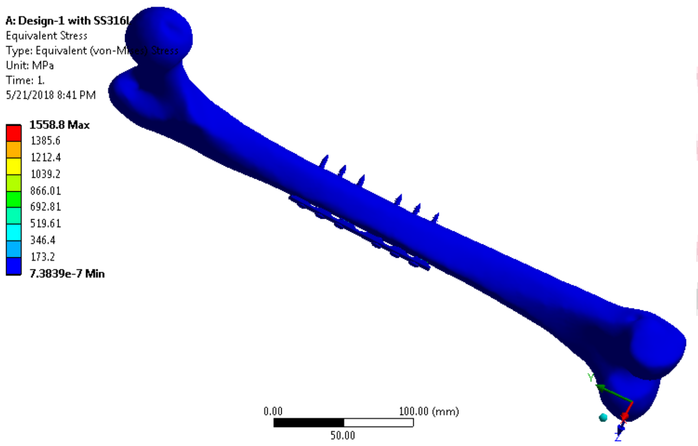

Figure 6: Equivalent Stress for SS316L

After performing the transient structural analysis the value of Equivalent Stress is observe as 1558.8 MPa and shown in figure 6 with different color contours.

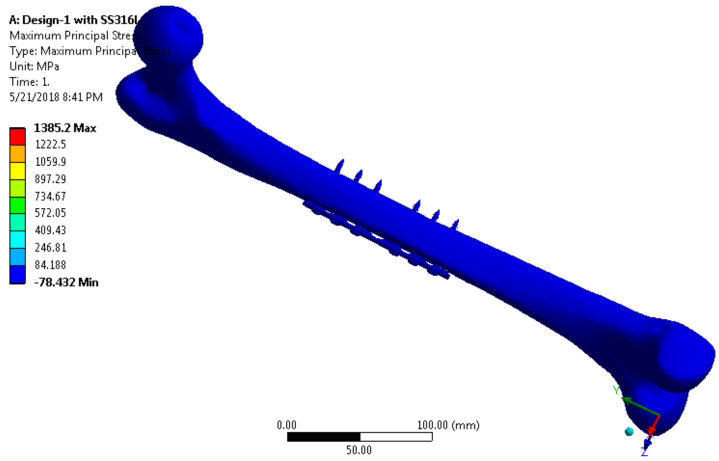

Figure 7: Principal Stress for SS316L

After performing the transient structural analysis the value of Principal Stress is observe as 1385.2 MPa and shown in figure 7 with different color contours.

Contact Analysis:

The position and motion of a contact element relative to its associated target surface determines the contact element status. ANSYS monitors each contact element and assigns a status:

- STAT=0 Open far-field contact
- STAT=1 Open near-field contact
- STAT=2 Sliding contact
- STAT=3 Sticking contact

IV. RESULT AND DISCUSSION

In the present work transient structural, contact and modal analysis on femur bone with plate assembly for different design and different materials have been performed and all results obtained from the above analysis have been compared in this chapter.

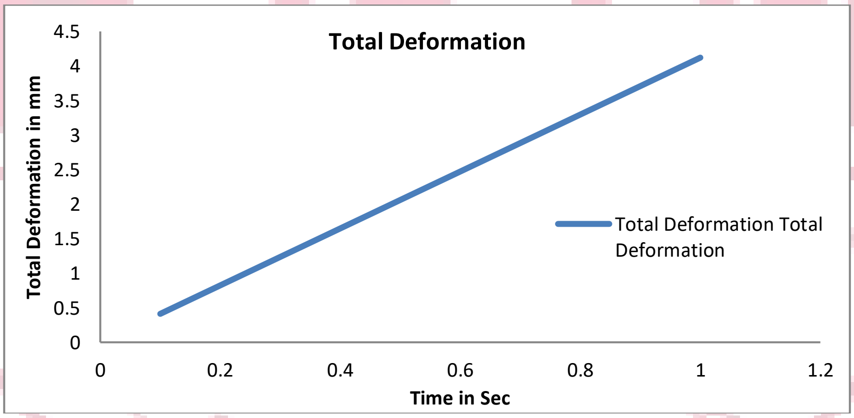

Figure 8: Total Deformation

Design with Ti6A14V:

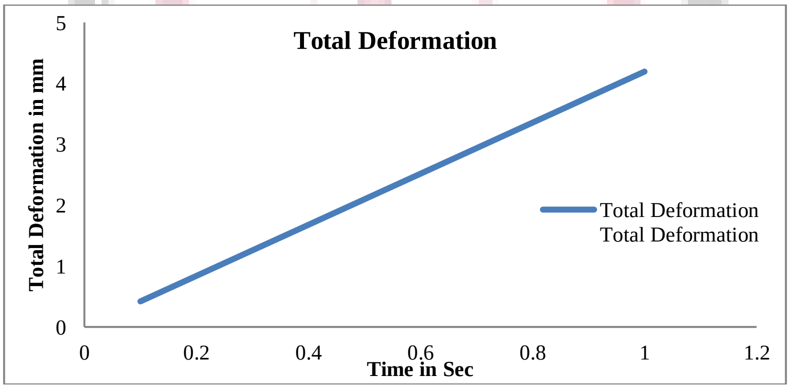

Figure 9: Total Deformation

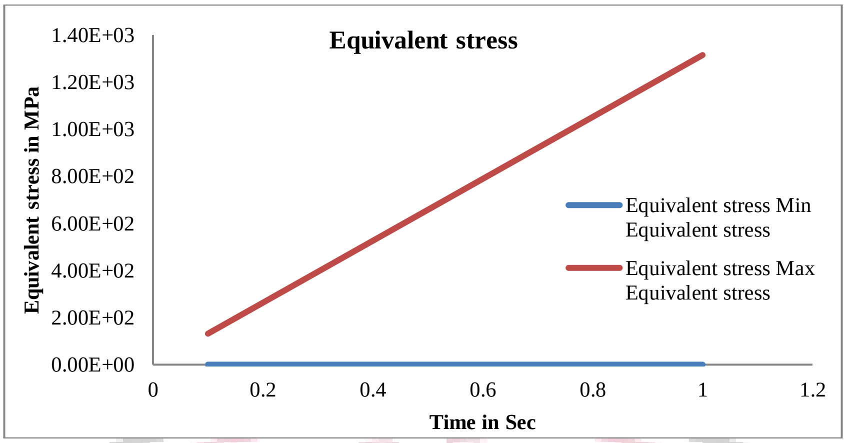

Figure 10: Equivalent stress

V. CONCLUSION

The structural contact and modal analysis of the femur bone is done with a single crack in middle of the shaft; this fractured bone is coupled with the prosthetic plate and screw of same materials. Assembled structured is analyzed by finite element method using transient structural analysis. s. With this approach, it is also possible to find out the very complicated fractured regions easily, and helps in smooth running of surgeries. Hence, this method paves way for better modeling and analyzing the anatomical structures like bones and other parts of body organs. In the present work a three dimensional CAD model of Femur bone with plate and locking screw is created with the help of CATIA software then imported in ANSYS workbench for further analysis. For contact analysis the contact status is sticking, frictional stress 1798.8 Mpa, contact pressure 948.27 Mpa Penetration 0.00035 mm and the minimum gap due to deformation is 0.013 mm. for the modal analysis the natural frequency range from 5.32Hz to 3973.7Hz.It has been observe from the above conclusion that Ti6Al4V gives the best suitable results and also suggested by doctors and also recommended best suitable biomaterial for femur bone implant.

REFERENCES

[1] H. Ahirwar, V. K. Gupta, and H. S. Nanda, "Finite element analysis of fixed bone plates over fractured femur model.," Comput. Methods Biomech. Biomed. Engin., vol. 24, no. 15, pp. 1742–1751, Nov. 2021, doi: 10.1080/10255842.2021.1918123.

[2] A. Arjunan, M. Demetriou, A. Baroutaji, and C. Wang, "Mechanical performance of highly permeable laser melted Ti6Al4V bone scaffolds," J. Mech. Behav. Biomed. Mater., vol. 102, p. 103517, 2020, doi: https://doi.org/10.1016/j.jmbbm.2019.103517.

[3] M. Fu, F. Wang, and G. Lin, "Design and research of bone repair scaffold based on two-way fluid-structure interaction," Comput. Methods Programs Biomed., vol. 204, p. 106055, 2021, doi: https://doi.org/10.1016/j.cmpb.2021.106055.

[4] A. Kumar and S. Jain, "Enhancement of Power Quality with Increased Levels of Multi-level Inverters in Smart Grid Applications," vol. 14, no. 4, pp. 1–5, 2022, doi: 10.18090/samriddhi.v14i04.07.

[5] X. Li et al., "The design and evaluation of bionic porous bone scaffolds in fluid flow characteristics and mechanical properties," Comput. Methods Programs Biomed., vol. 225, p. 107059, 2022, doi: https://doi.org/10.1016/j.cmpb.2022.107059.

[6] S. Wang et al., "Honeycomb structure is promising for the repair of human bone defects," Mater. Des., vol. 207, p. 109832, 2021, doi: https://doi.org/10.1016/j.matdes.2021.109832.

[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 Conf. Series: Materials Science and Engineering 330 (2018) 012027 doi:10.1088/1757-899X/330/1/012027.

[8] A. Kumar and S. Jain, "Critical Analysis on Multilevel Inverter Designs for," vol. 14, no. 3, 2022, doi: 10.18090/samriddhi.v14i03.22.

[9] A. Kumar and S. Jain, "Predictive Switching Control for Multilevel Inverter using CNN-LSTM for Voltage Regulation," vol. 11, pp. 1–9, 2022.

[10] C. Gupta, V. K. Aharwal, and M. Pradesh, "Design of Multi Input Converter Topology for Distinct Energy Sources," vol. 14, no. 4, pp. 1–5, 2022, doi: 10.18090/samriddhi.v14i04.09.

[11] V. Meena and C. Gupta, "A Review of Design , Development , Control and Applications of DC – DC Converters," no. 2581, pp. 28–33, 2018.

[12] Koris J., Blunn G., Coathup M., "DOES BONE REMODELLING PROTECT A FEMORAL IMPLANT: FINITE ELEMENT ANALYSIS", Published Online: 21 Feb 2018.

[13] S. Karuppudaiyan, J. Daniel Glad Stephen, V. Magesh, "FINITE ELEMENT ANALYSIS OF TIBIA BONE BY REVERSE ENGINEERING MODELLING APPROACH", International Journal of Pure and Applied Mathematics, Volume 118 No. 20 2018, 839-846, ISSN: 1311-8080 (printed version); ISSN: 1314-3395 (on-line version).

[14] C. Gupta and V. K. Aharwal, "Optimizing the performance of Triple Input DC-DC converter in an Integrated System," *J. Integr. Sci. Technol.*, vol. 10, no. 3, pp. 215–220, 2022.

[15] C. G. Aditya Hridaya, "International Journal of Current Trends in Engineering & Technology ISSN : 2395-3152 AN OPTIMIZATION TECHNIQUE USED FOR ANALYSIS OF A HYBRID International Journal of Current Trends in Engineering & Technology ISSN : 2395-3152," *Int. J. Curr. Trends Eng. Technol.*, vol. 06, no. October, pp. 136–143, 2015

[16] A. Kumar and S. Jain, "Multilevel Inverter with Predictive Control for Renewable Energy Smart Grid Applications," *Int. J. Electr. Electron. Res.*, vol. 10, no. 3, pp. 501–507, 2022, doi: 10.37391/IJEER.100317.

[17] P. Mahapatra and C. Gupta, "Study of Optimization in Economical Parameters for Hybrid Renewable Energy System," *Res. J. Eng. Technol. …*, no. 2581, pp. 39–46, 2020, [Online]. Available: http://www.rjetm.in/RJETM/Vol03_Issue02/Study of Optimization in Economical Parameters for Hybrid Renewable Energy System.pdf.

[18] A. Kumar and S. Jain, "A Review on Cascaded Multilevel Inverter with Different Levels," pp. 134–137, 2019.

[19] Feifei Jiang, Shengzhi Liu, Andy Chen, Bai-Yan Li, Alexander G. Robling, Jie Chen, Hiroki Yokota, "Finite Element Analysis of the Mouse Distal Femur with Tumor Burden in Response to Knee Loading", Int. J. of Orth. 2018 February 28; 5(1): 863-871, ISSN 2311-5106 (Print), ISSN 2313-1462 (Online).

[20] Evandro Pereira Palacio, Gilberto José Cac¸ão Pereira, Paulo Roberto de Almeida Silvares, Gabriel Guimarães Di Stasi, Caio de Andrade Staut and Trajano Sardenberg, "The effects of ethyl-2-cyanoacrylate and butyl-2-cyanoacrylate in the process of bone healing in rats. A controlled experimental study", r e v b r a s o r t o p . 2 0 1 8;5 3(1):53–59.

[21] Alberto de Castro Pochini, Marcus de Souza Barbosa Rodrigues, Larissa Yamashita, Paulo Santoro Belangero, Carlos Vicente Andreoli and Benno Ejnisman, "Surgical treatment of pectoralis major muscle rupture with adjustable cortical button", r e v b r a s o r t o p . 2 0 1 8;5 3(1):60–66.