Modelling the Tibial Bone Using Reverse Engineering Technique and FGM Modelling of Simple Bar Using ANSYS

Niket Ashok Shinde
Assistant Professor, Sinhgad Institute of Technology
doi.org/10.64643/IJIRTV8I11-187411-459

Abstract — **This work explores the area of biomedical imaging processing. Tibia is the largest bone after femur in the human body. Due to its complex geometry, developing an accurate and reliable model of Tibia bone is a challenging task for orthopedics. Therefore, to develop a 3D bone model Reverse Engineering Technique is used. This method minimizes the error in importing the models into the Finite element software to get better results. 3D slicer software is used to reconstruct the bone from Ct scan using DICOM files. A 3D model of human tibia bone has been modelled from CT scan images which is further segmented and modified to form an accurate 3D model by varying the segmentations parameters and methods. An open-source code 3D slicer is used to perform segmentation. Autodesk Fusion 360 software is used to convert it into a solid body and to reduce the facets and finally the model is imported to ANSYS software for meshing and analysis. This segmentation 3D CAD model can be used for analysis such as Model analysis, structural analysis and artificial bone manufacturing. In-plane stress and displacement solutions are found using the elasticity approach for functionally graded rod having rectangular, hollow, tapered and tapered hollow cross-section and are validated with Finite Element Solution in Ansys. The analytical results show a good agreement with FE results.**

Index Terms – **ANSYS, Biomechanics, Finite element analysis, Tibia bone, 3D slicer.**

I. INTRODUCTION

Bone in biological terms is defined as connective tissue because it binds together and provides protection to various parts of the body. The lower limb consists of thigh, leg and foot. The leg bone consists of Patella, Tibia and fibula. The fibula bone is not involved in knee joint structure; the upper end is located further down and is located at the lateral side of the tibia bone. Tibia bone is the second largest and the strongest bone after femur bone, it is also called as shinbone, the proximal(upper) end of tibia bone articulates with the femur bone forming the knee joint and distal(lower) end of the tibia bone articulates with the talus forming the ankle joint. Tibia consist of 2 epiphyses and one diaphysis, epiphysis are the two rounder extremities of the bone, upper extremities (closet to femur) and lower extremities (closet to knee). Diaphysis is also known as the shaft is the mid-section of the tibia. It is very complicated to develop the computational model of tibia since it contains three layers of architecture. i) Cortical also known as compact bone.it is the dense outer surface of bone that protects the internal cavity by forming a layer. It is found in the shaft of tibia and femur bone and forms the outer shell around the trabecular bone at the ends of the joint and vertebrae. ii) Trabecular (also known as spongy bone or cancellous bone) from the articular surface it transfers the mechanical load to the cortical bone. It is an anisotropic, heterogeneous and highly porous material that is found at the epiphyses of the tibia and femur bone and in vertebral bodies is a highly iii) Bone marrow is a soft tissue stored in the central cavity of bone shafts known as medullary cavity. Objective of this study is to model the femur bone by using reverse engineering technique and to compare the analytical and FE simulation results of functionally graded plane rod.

II. LITERATURE REVIEW

Javad Malekani et.al provides the guidelines for the user to reduce the computational time and increase the accuracy of analysis in ANSYS. Various parameters like solution and geometric model, material model, mesh generation, number of analysis step, program control parameters, large deformations, operating system, etc. These parameters affect the accuracy and increase the computational time therefore it should be chosen carefully [1].

Martin Kub´ı˘cek et.al used CT images of femur and tibia bone to create the geometric model and other model like cartilage and meniscus obtained with aid of literature. Material properties of models were created as isotropic and homogeneous.[2].

WanchalermTarapooma constructed the 3D model of Tibia bone from CT scan of by using MIMICS, GEOMAGICS and CATIA software. HyperMesh software used to generate the FE models than it is imported to ANSYS to study the stress distribution during stance phase running [5]

S. Karuppudaiyan et.al used CT scan images to develop the FE Bone model but for accurate model they required a better algorithm. In this paper A reverse engineering method is used to develop a finite element model of tibia bone to study the fracture risk and structural behavior of tibia bone under physiological loading conditions. This method can minimize errors in importing the models into the FEM software to obtain better results. This study helps doctors to have knowledge of implants during fracture. FE analysis was carried out and the results obtainedwere compared with literature. The maximum stress was obtained in the mid International Journal of Scientific & Engineering Research, Volume 8, Issue 1, January-2017 ISSN 2229-5518 diaphysis of the tibia bone. This stress is 8% is higher than the previous literature. Therefore, reverse engineering approach is one of the best methods to develop complex geometric model [6].

Naveen Tippanagoudar et.al carried out the analysis of Tibia bone during static loading condition and during impact loading condition. From the analysis it is found that, for static load the stress induced is within the Yield point, and the stress induced for impact load is higher than the yield strength of the bone therefore the bone would fracture under impact loading during crash [7].

Özkan et.althe congruent geometry model obtained from CT images and non-bone structure (cartilage, tendon, etc.) is develop in GEOMAGIC the model is assembled in MIMICS and then it is transferred to ANSYS workbench for FE simulation. Boundary conditions are applied by fixing the lower side of Talus, and a rotational moment is applied on the upper surface of the tibia, which is a connection interface between a tibial plateau and meniscus; analysis was performed with both fixed and broken fibula. With fixed fibula the stress and displacement values decreased and have less value than broken fibula.it can be concluded from the paper that the fixation of fibula fracture affects positively more than no fixation operations [3].

Ehsan Taheri et.alobtained the bone geometry from CT images of a 32-years old healthy individual. MIMICS software was used to separate the tibia bone from femur, fibula and other muscles and then export point clouds text file, to have a perfect model of compact and spongy parts and of tibia it was imported to solid works where the volume construction & extension of nodes were done. For analysis it was imported to Abaqus software. Material properties selected were anisotropic material. Torsion stiffness and bending of the CAD model was calculated and the results were compared with previous literature studies [12].

Kirthana et.al collected MRI or CT data to make a CAD model of bone. The bone is reconstructed using a 3D slicer and to obtain a smoother surface blender is used. Analysis is done to obtain the maximum stress and deformation. Weak zone areas are taken as regions of interest after an impact load is applied, prosthetic plates are modelled and assembled in the femur bone at the ROI. Different plates are used and assembled and analysis done and comparison made to find out the best material i.e., Titanium plate [14].

III. EXPERIMENTATION AND BONE MODELLING

2.1 Medical Imaging Technique X-ray, CT scan and MRI are the techniques used mostly to capture images of skeletal of humans. These techniques mainly differ in their capacity to penetrate tissues and bones and degree of resolution that can be achieved. CT (computerized Tomography) can process a series of X-ray images that have been taken around the body at different angles to create cross-sectional images of the bones(slices), soft tissues & blood vessels inside your body. computerized Tomography scan provide more detailed information than X rays images. In this study the Tibia bone is reconstructed from CT scan. A CT scan of individuals suffering from degeneration is taken and reconstructed from DICOM images represented as dcm. Major advantage of dcm format is that it does not provide the information about the source (anonymized data) and it cannot be misinterpreted

2.2 3D slicer The CT scan DICOM files of lower limb is imported into 3D slicer software.3D slicer software is used to construct a CAD model of tibia bone. It can read CT and MRI data.

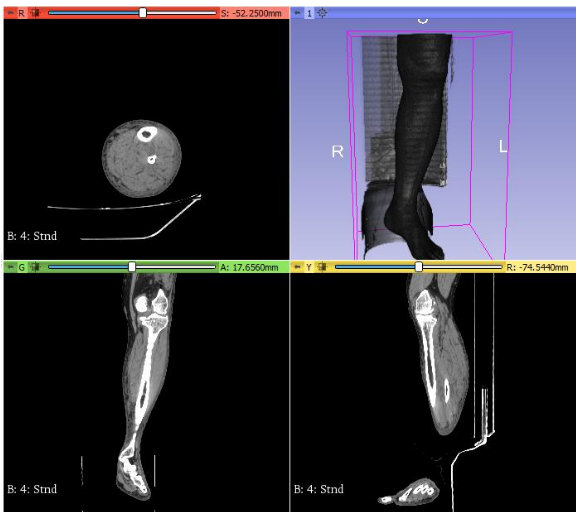

Fig 1: CT scan of lower limb of 73 years old in 3D slicer

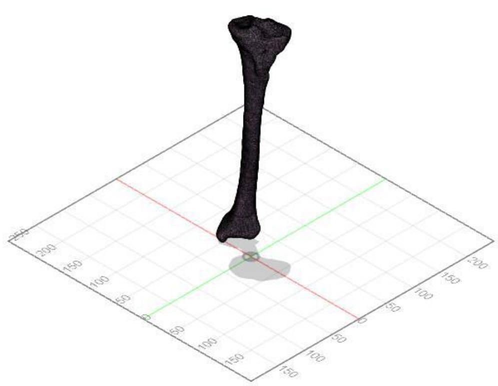

Fig 2: Volumetric meshing in Autodesk Fusion

IV. CALCULATIONS AND FE SIMULATION

The geometry of the tibia bone is almost like a cylindrical tapered hollow bar and the young's modulus of bone varies in nonlinear manner. To achieve the accurate FE analysis results we compared the analytical solution with the FE simulation results by assuming linear variation of young's modulus w.r.t length. Starting with a simple rectangular bar to a Cylindrical tapered hollow bar, analysis is done and the results are compared with an analytical solution. If both the results matched it can be concluded that the FE simulation that was performed on ANSYS software was correct. An analytical solution equation is required for Young's modulus varies linearly from X=0 to X=L. i.e., it varies linearly from Erootto Etip along the length of the bar.

E at a distance x,

$Ex=Er+(Et-Er)xL=Er+Kx$

Change in length with young's modulus varying linearly, $\Delta l=PL(Et-Er)[lnln(EtEr)]$

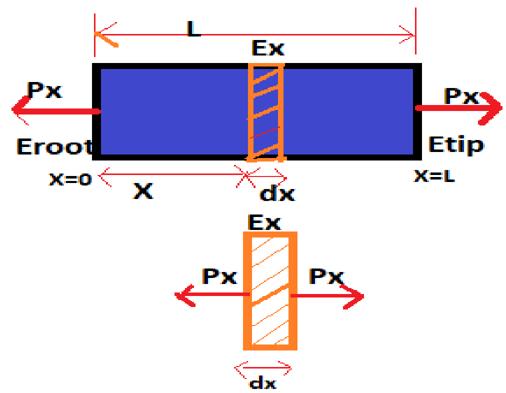

Fig 3: Rectangular bar with varying young's modulus

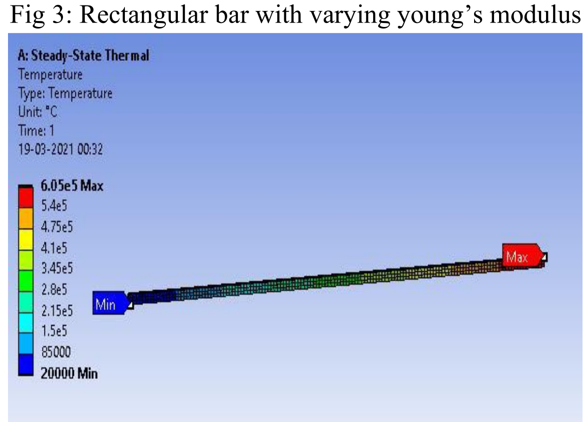

Fig 4: Linear spacial variation of temperature

Step 1 – Define material properties as a function of temperature and do simple conductive thermal analysis to get the distribution of temperature through the longitudinal direction of the bar

Assign Coefficient of thermal expansion(α)=0 to neglect the effect of temperature in the solution

Step 2- Carry over structural analysis after mapping the thermal analysis results

Eroot=20000 Troot=20000, Etip=605000 Ttip=605000(Assume)
Where E = Young's Modulus and T=Temperature.
Any relation can be maintained between them like power law or exponential law i.e. (non-linear variation)
From fig-10 it can be seen that the temperature is varying linearly from L=0 (20000 Celsius) to L=550mm (605000 Celsius) and since the temperature value is equal to young's modulus value, we get the linear variation of young's modulus w.r.t length.

Boundary Conditions:
For Finite element Analysis an important step required is the boundary condition for the model. Boundary conditions have a great impact on the analysis results. A fixed boundary condition is applied at L=0 surface and a load of 500N is applied at the free end (L=550) in X-direction for all the model (rectangular, hollow cylinder, tapered and hollow cylindrical tapered bar).

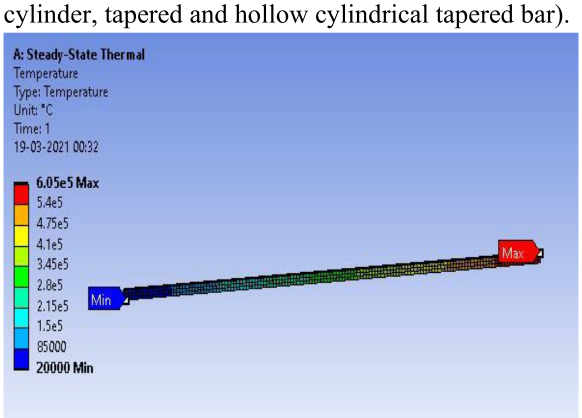

Fig 4: Linear special variation of temperature

V. CONCLUSION

Reverse Engineering is one of the best methods to develop 3D bone models. The work done in this study is an initial step and could lead to the development of an expert system. The CAD modelling of the tibia bone took place without any experimental work. This study may help orthopedic surgeons to have a detailed knowledge about the tibia bone to choose appropriate implants for fractures which help in providing an efficient and fast fixture to a patient with a fracture. The contributions towards orthopedic biomechanics of both doctors and engineers could hence prove to be important and extremely beneficial to doctors and patients. Tibia bone material properties vary in nonlinear manner and in material properties of ANSYS
software it does not provide a direct way to model variation therefore a thermal-structural analysis is performed on simple bar by varying the young's modulus linearly from x=0 to x=L and same value of temperature and young's modulus is taken and to neglect the expansion of bar due to thermal effect the coefficient of expansion value is taken as 0. FGM modelling of simple bar is with linearly varying young's modulus is calculated analytically and also solved in FE simulation software and their results are compared. As seen in the fig 12, fig 13, fig 14 and fig15 there is very small error and sometimes the error is zero. This method delivers high accuracy and it does International Journal of Scientific & Engineering Research, Volume 8, Issue 1, January-2017 ISSN 2229-5518 not require deep knowledge of FGM mathematical
models.

REFERENCES

[1] Javad Malekania, Prasad KDV Yarlagaddab,, Beat Schmutzc, YuanTongGud,Michael Schuetz 'How to increase the accuracy of analysis and reduce the computationaltime in ANSYS in the case of deformation study of orthopedic bone Plates' Advanced Materials Research Vols,2013.
[2] MartinKub´ıˇcek,Zdenˇek, 'STRESS STRAIN ANALYSIS OF KNEE JOINT', Engineering MECHANICS, Vol. 16, 2009, No. 5, p. 315–322
[3] A.Özkan,Y.K.işioğlu, 'The rotational loading capacity and effects of fibula on lower exterimity', Journal of Engineering Research and Applied Science, June 2013, pp 138-143.
[4] Mihai-Constantin Balaşa, Simona Mihai, Viviana Filip, Alexis-Daniel Negrea, GheorghiţaTomescu, 'MODELLING THE TIBIAL BONE USING CAD TECHNIQUES, STARTING FROM THE 3D SCAN MODEL' International Journal of Mechatronics and Applied Mechanics, 2018, Issue 3.
[5] WanchalermTarapooma and TumrongPuttapitukpornb 'Stress Distribution in Human Tibia Bones using Finite Element

Analysis', ENGINEERING JOURNAL Volume 20 Issue 3,2016.

[6] 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.

[7] Naveen Tippanagoudar, Anantha Krishna G L,' Finite Element Analysis of Tibia Bone' International Journal of Engineering Science and Computing, December 2018.

[8] G Narayanaswamy, Bindu A Thomas, 'Finite Element Analysis of Tibia Bone Model', International Journal of Engineering and Advanced Technology (IJEAT) ISSN: 2249 – 8958, Volume-9 Issue-1, October 2019.

[9] N. Suryarao, K. Sreedevi, MD. Touseef Ahamad, 'Analytical Study of the Tibia Bone under Static Load using Finite Element Method' International Journal for Research in Applied Science & Engineering Technology (IJRASET), Volume 5 Issue VIII, August 2017.

[10] Mr. Amit Patil, Mr. V. R. Gambhire, Mr. P. J. Patil, 'EXPERIMENTAL AND ANSYS ANALYSIS OF STRESSES IN REAL PRESERVED INTACT HUMAN TIBIA BONE UNDER STATIC BODY LOAD TO ENHANCE THE DESIGN OF IMPLANT FOR ORTHOPEDIC SURGERY.', International Journal of Advance Engineering and Research Development Volume 5, Issue 07, July -2018

[11] M. Ghosh, B. U. Chowdhury, M. S. Parvej, and A. M. Afsar, 'Modeling and analysis of elastic fields in tibia and fibula', AIP Conference Proceedings 1919, 28 December 2017

[12] Ehsan Taheri1, Behrooz Sepehri2, Reza Ganji3, 'Mechanical Validation of Perfect Tibia 3D Model Using Computed Tomography Scan', Engineering, December-2012,

[13] Fritz J. and Dolores H.Russ, ', 3-D Modeling and Finite Element Analysis of the Tibia'.

[14] S. Kirthana, M. BindhuSupraja, A.S.N. Vishwa, N. Mahalakshmi, StaticStructural Analysis on Femur Bone Using Different Plate Materials Today: Proceedings, Volume 22, Part 4,2020, Pages 2324-2333, ISSN 2214-7853.