Finite Element Analysis of Tibia Bone during the Rehabilitation Period

Pratik S. Thakre[1] Dr. M. Faizan[2] Dr. K. S. Zakiuddin[3]
[1]PG Student [2]Associate Professor [3]Professor & Head
[1,2]Department of Mechanical Engineering [3]Department of Orthopaedic
[1,2]Priyadarshini College of Engineering, Nagpur, MH, India [3]NKPSIMS India

Abstract— Biomechanics involves modelling and simulation of biological objects using mechanical laws. In humans, biomechanics often refers to the study of how the skeletal and musculature systems work under different conditions. In biomechanics more generally, scientists often try to apply physics and other mathematically based forms of analysis to discover the limits and capabilities of biological systems. The tibia is the major weight-bearing bone of the lower leg. The proximal portion of the bone, the tibial plateau, forms the lower surface of the knee joint. Today, rather than a field that scientists and philosophers dabble in, biomechanics is its own branch of human and biological science, with entire departments in hospitals and universities devoted to the subject's study. The tibial shaft bridges the distance to the distal tibia which grant the superb articular surface of the ankle joint at the tibiotalar articulation as well as the medial malleolus. Another key bony landmark is the tibial tuberosity which sits many centimeters below the joint line and the poorer patellar pole and serves as the attachment site for the patellar tendon. A strong fibrous structure, the interosseous membrane, or syndesmosis, connects the tibia and fibula along the length of the two bones. Proximally, this structure, reinforced by strong anterior and posterior ligaments, makes a synovial joint, the proximal tibiofibular articulation. Distally, the interosseous membrane and three ligaments, the anterior, posterior, and transverse tibiofibular ligaments stabilize the superior ankle joint. The objective of this research work is to perform the Finite element analysis of Tibia bone during the post-surgery period. The analysis will be performed to compare the healthy bone, bone after one month of surgery, bone after two months of surgery. This will be helpful for doctors and surgeons to understand the various aspects of bone strength during the rehabilitation period.

Key words: Tibia, Modelling, Simulation, Finite Element Analysis, Post-Surgery Period

I. INTRODUCTION

An artificial or natural material like metals, polymers, ceramics, composites etc. used to replace diseased or fractured biological structures to restore its form and function is called as biomaterials. Before using these biomaterials it must be necessary to see whether these satisfies required function and mechanical properties i.e. Bio functionality. Also most important these must be compatible with human body i.e. Biocompatibility. Since biomechanics is the study of human movement and interaction with the environment, the field has myriad applications in daily life and touches on many different sciences. Let's look at a few examples of the areas in which biomechanics is most commonly used.

The finite element method was first used in bone biomechanics for analysis of mechanical behaviour of skeletal parts in 1972 (Huiskes and Chao, 1983). Steadily this method has become very popular in biomechanics field. Finite element modelling (FEM) has three major stages to analyse the human bones i.e. pre-processor, solution, and the post process stage. In the pre-process stage a CAD model is required to be generated. Thegeometry and material properties (CT HounsfieldUnits) of bonecan beacquired from computed tomography (CT). The geometry of fracture fixation (implant) is usually developed on CAD software like CATIA, Solid works, Pro/E etc. Once the bone model is developed the mesh generation is carried out. The material properties to each model is assigned and finally the boundary conditions are applied (Fig. 1). It is essential to apply the correct boundary conditions in FEM to get accurate results.

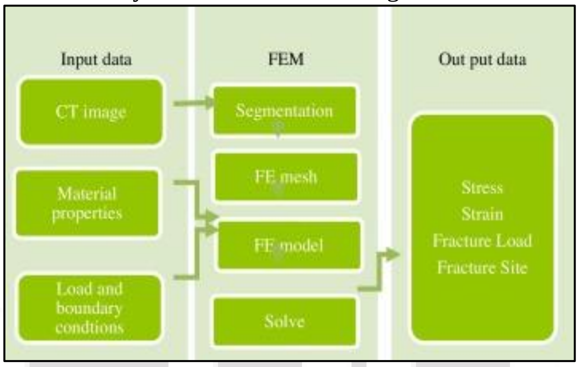

Fig. 1: Finite element modelling of bone based on CT images.

II. LITERATURE REVIEW

Pradeep Bokariya, [1], the objective of this research paper is to provide scientific method and technique for taking various measurements in different geographic regions and races. To find out the measurement of Cross-Section Index in middle, Cnemicus Index and Length-Thickness Index for both right and left Tibia, 60 (26 right and 34 left) intact adult tibia were got from the bone bank of Anatomy Department of MGIMS, Sevagram. For this reason a digital vernier caliper, osteometric board and measuring tape were used. The average of Cross Sectional index for Right tibias was 102.90± 22.78. Similarly mean of Cross Sectional index for left tibias was 124.31± 25.06. The mean of Cnemicus Index for Right tibias was 66.16 ± 10.67 and for left side these values came out to be 67.31 ± 7.35.The mean of Length-Thickness Index were 24.21 ± 0.96 and 24.45 ± 1.78 for right and left Tibias respectively. Ultimately a comparative statistical study was performed. In their study they found that the mean value of the length of the right and the left femora was statistically similar (there was difference which was insignificant).

They concluded that as the number of bones taken was small so significant differences were shown in the result.

Namrata Phate,[2],author used the three-dimensional finite element analysis to evaluate the stresses and displacements of human tibia bone under physiological loading. CT data of 17 years and 27 years elder male, and 37 years old female patients are used to model three-dimensional finite element models of the left proximal tibia bone, and half of an average body weight 65 kg (318.825N) was applied to each model of tibia bone. Finite Element Analysis conducted to calculate the Equivalent Von-Mises Stress, Maximum Principal Stress, Total Deformation and Fatigue Tool from the total proximal tibia bone and comparing the results. These analyzed results may provide a great foundation for further studies of bone injury prevention, bone transplant and subject-specific fracture mechanism.

Stevanovic.D,[3],In this investigation author describes a preliminary analysis of a created FE model, which was performed in order to check model integrity and validity. A procedure for creation of subject specific finite element (FE) models of human tibia is described in the paper. The very important phases of the procedure are: extraction of polygonal model from medical images, creation of surface development based on polygonal model and creation of FE model. To acquire this goal, material properties were allotted to FE model by means of automatic mapping of CT numbers onto FE models, based on correlation between bone density and elastic properties of bone parts. Optimal mesh density was done through convergence checks, performed on a series of FE tibia models of growing mesh density.

Radu C,[4], purpose of this paper is to reconstruct a human tibia by using 2D tomographic slice sand to perform a static finite element analysis. The 3D human tibia was processed using biomedical imaging and editing software packages. The work resulted in three-dimensional solid model can be used in finite element analysis. A comparison with engineering may help us to understand the principles that presumably guide the evolution of bone strength. Experiments provide insights and data, which can then be interpreted within the context of analytical frameworks. These investigations are greatly impacted by recent technological advances in imaging, computational mechanics, genetics and molecular biology. The integration of these techniques will provide important insight into skeletal development and disease.

Radu C,[5],in this investigation the human limb model is generated, characterized by solid tetrahedral element mesh was able to analyze the stress in the purpose of validating mechanical and material properties of bone.It will help surgeon and biomechanical researches to develop improved implants and treatment method for patient suffering from bone loss and diseases.

Nareliya and Kumar,[6],authorreviewed some papers related to the finite element analysis and elaborated the anatomy of the femur bone. In all 47 technical papers were reviewed and explanation of FEA was well written and it was concluded that, for modeling the human femur bone computational techniques were used and for observing the response FEM was used. In most of the research articles CT data of dry, frozen or moist bone had been used to get the 3D well dimensioned model. In case of the THR (total hip replacement) and effect of nailing research was carried by using FEM. It was also observed that the angle of inclination varies between 7° to 28° for different individuals under loading conditions for analysis of stress and deflections. In various articles it was found that the in-homogenous models of femur predicts good accuracy for the measured stress field and homogenousmaterial femur gave less accurate results.

Yousif and Aziz ,[7], had done the biomechanical analysis of human femur bone during normal walking and standing conditions. They modeled a human femur bone of a 40 year old healthy individual whose weight is 75 kg and which was reconstructed from DICOM (CT) images. They had fixed the distal end of the femur and on the head of the femur the hip contact forces had been applied for the calculation of the normal stresses in normal walking and standing up conditions. After that the boundary conditions were interchanged and the result average was considered. It was observed maximum normal stress for both normal walking and standing up conditions was observed at the neck of the femur.

Wanchalerm Tarapooma,[8],presents the accuracy of finite element models of human tibia bones generated from CT-images used for analyzing stress distribution under loading. The effects of bone structures and material properties (isotropic and orthotropic materials) on stress distribution during stance phrase running were studied. Three-dimensional tibia models were constructed by using Mimics V.10.01, Geomagic V.10 and Catia V.5 software. Then these models were imported into the Hypermesh V.12 software to generate the FE models. Finally the FE models are imported to ANSYS (APDL) V.14.5 software to analysis the stress distribution in the tibia bones.

III. Conclusions

Till date, the research is being done on the solutions on the tibia bone fracture. There is many remedies on the tibial bone fractures. Material properties of the bone are not homogenous and it varies from individual to individual and it directly affects the stress state in the bone. In this review eight research papers related to femur modeling, tibia stress analysis, femur stress analysis and their anthropometry were thoroughly studied. From all the above papers it may be concluded that for the analysis of either tibia, model is needed. Then these models were imported into the Hypermesh V.12 software to generate the FE models. Finally the FE models are imported to ANSYS (APDL) V.14.5 software to analysis the stress distribution in the tibia bones. The objective of this research work is to perform the Finite element analysis of Tibia bone during the post-surgery period. The analysis will be performed to compare the healthy bone, bone after one month of surgery, bone after two months of surgery. The problems and issues faced by the orthopedic surgeons during tibia fractured surgery have been solved by these investigations. This research work will helpful for doctors and surgeons to understand the various aspects of bone strength.

References

[1] The anthropometric measurements of tibia, Pradeep Bokariya, Bharat Sontakke, JE Waghmare, Aaditya Tarnekar, BH Tirpude, MR Shende, J Indian Acad Forensic Med. October-December 2012, Vol. 34, No. 4
[2] Three-dimensional finite element analysis of human tibia bone, Namrata Phate1, Raji Nareliya2, Veerendra Kumar3, (IJSRET), ISSN 2278 – 0882 Volume 3 Issue 1, April 2014
[3] Finite element model of human tibia and preliminary analysis, Stevanov ić, D., Korunović, N., Trajanović, M., Trifunović, 11th International Scientific Conference Novi Sad, Serbia, September 20-21, 2012

[4] 3D modeling and static finite element analysis of human tibia. Radu C. International Journal of Scientific Research Engineering & Technology (IJSRET), ISSN 2278 – 0882 Volume 3 Issue 1, April 2014.
[5] 3D modeling and static finite element analysis of human tibia. Radu, C.advanced engineering 2(2008)1, ISSN 1846-5900
[6] Nareliya R and Kumar V (2012),“Finite Element Application to Femur Bone: A Review”, Journal of Biomedical and Bioengineering, Vol. 3, No. 1, pp. 57-62.
[7] Yousif A E and Aziz M Y (2012), “Biomechanical Analysis of the Human Femur Bone During Normal Walking and Standing Up”, IOSR Journal of Engineering (IOSRJEN), Vol. 2, No. 8, pp. 13-19.
[8] Article, Stress Distribution in Human Tibia Bones using Finite Element Analysis Wanchalerm Tarapooma and Tumrong Puttapitukpornb,*
[9] Department of Mechanical Engineering, Faculty of Engineering, Kasetsart University, Bangkok 10900, Thailand.