Australian Journal of Basic and Applied Sciences, 9(35) November 2015, Pages: 331-335

ISSN:1991-8178

Australian Journal of Basic and Applied Sciences

Journal home page: www.ajbasweb.com

Unconventional Modeling and Stress Analysis of Femur Bone under Different Boundary Condition

M. Ragulkumar

Assistant Professor, Dept. of ECE, Shree Sathyam College of Engineering and Technology, Sankari

ARTICLE INFO

Article history:
Received 1 November 2015
Accepted 28 November 2015
Available online 18 January 2016

Keywords:
ANSYS; Finite Element Analysis; Femur bone; Biomechanics;

ABSTRACT

Femur is leg bone of the human body undergoing the more deformations. Being longest and the heaviest size, failure of a femur neck is the most common among them bone failures in human especially of woman. Orthopedic implantation has done by case of failure. Before implantation method is necessary to be analyzing their perfectness in the case of its material property, load resistance, size, shape, surface treatment chances of failure. Since each femur carries 1/2 the body weight, analysis is done for as 25kg, 50kg, 100kg, load includes the case of patients carrying the certain weight. From these analysis methods is found their load has been no effects on failure of implantation, since these values of stresses and deformations is very low. Thus failure of implantations is maximum by wearing this corrosion, all due to improper material selection of implant. Finally the success of implantations depends on implant material and sizes, implantation method and its handling by the patient.

To Cite This Article: M. Ragulkumar., Unconventional Modeling and Stress Analysis of Femur Bone under Different Boundary Condition. ***Aust. J. Basic & Appl. Sci.****, 9(35): 331-335, 2015*

INTRODUCTION

The Femur the longest and strongest bone in the skeleton is almost perfectly cylindrical in the greater part of its extension. In the erecting posture is not vertical, being separated from above its following by as Considerable interval, which its corresponds to the breadth of their pelvis, but inclining gradually downward and medial ward, so as to approach its fellow toward it's a lower part, for the purpose of bringing the knee-joint near The degree of this inclination varies in different persons the line of gravity of the body, and is greater in the female than in the male, on account of the greater breadth of the pelvis. The femur like on this other long bone is divisible into a body and two extremities.

The Upper Extremity (proximal extremity) the upper extremity presents for examinations of a head, a neck, a greater and a lesser trochanter. The Head (caput femoris)- The head which is globular and forms rather than a hemisphere, is directly upward, medialward, and little forward, the greater part of its convex ion being above and in front. In the surface is smooth and coated with then cartilage in this fresh state, except over an avoid depression, the capitis femoris, which is situated as below and their behind the center of their head, and gives attachment to the ligament teres (Ashwain Kumar, Himan Shu jaiswal, 2014).

Then, P-FE auto mesh is generated by within the each region (cortical or trabecular) the heterogeneous density is mainly described by as continuous spatial functions that are generated by using Least Mean Square (LMS) approximations based on the CT scan data. Finally the young modulus is evaluated by continuously according to the functional representation of the density. Numerical investigation shows good results of the suggested method compared to voxel based method. Mechanical inventor experiments in fresh frozen femur measuring head deflection and strains at several points are currently used for FE model calibration and validation (Amalraju, D., A.K. Shaik Dawood, 2012).

Material used in SMP is stainless steel 316 L for stem and same material i.e. stainless steel 316 L for head. This prosthesis is implanted in this patient surgical technique. In present method study steel ball is replaced by Ultra High Molecular Weight Poly Ethylene (UHMWPE) ball (Chawla, A. and S.S. Mukherjee).

A perspective three-dimensional geometry model for the femoral bone using subject-specific geometry from X-ray computed tomography image data (CTI) is used to create 3D (Computer Aided Design) CAD model. Force acting on femur is different gait regimes were reviewed and compared while loaded with standard and then with Proximal

Corresponding Author: M. Ragulkumar, Assistant Professor, Dept. of ECE, Shree Sathyam College of Engineering and Technology, Sankari1,
E-mail: ragulragul91@gmail.com1,

stem. A final method is developed by Finite Element Analysis model based on the created CAD model with forces acting on both femurs loaded with standard and proximal stem is analyzed. (David, P., 2003).

The articulating surface was kept intact to avoid altering the gait and to preserve the correct patellar grove so as to prevent resurfacing and patellar dislocation. A set of spine curves were created along the interface surface in a radial pattern and a single spine is created to by connect all the curves in a central plane. A swept blend command has used to be created by the smooth articulating surfaces. The bone implant interface was created by using their new set of curves in the same plane as the original curve. The inner curves are offset from their original curve and then manually edit to avoid any undercuts (Dave W., 2013).

The biocompatibility of implant quality stainless steel has been proven by successful human implantation for decades. Compositions of microstructures and tensile properties are cobalt chrome, zirconium and stainless steel, titanium is used for internal fixation is standardized in IS and ASTM material specifications. The metallurgical requirements are strings to an ensured sufficient corrosion resistance, nonmagnetic response and satisfactory mechanical properties. Torsional properties of stainless steel screws are different for as titanium screws. Stainless steel bone screws are easier to handle because the surgeon it can feel the onset of plastic deformation and this provides adequate pre-warning to avoid over torquing the screw (Eran Peleg, 2010).

Femur bone:

The femur, as stated is the longest limb bone with several distinctive bony elements. It is fairly distinct bone with a high level of robusticity and dense of compact bones due to it’s being the mainly supporting a limb doing ambulation. The head of their femur fits into the acetabulum of the hip bone (ileum, is chum, and pubis bones). Unlike they humorous and the globoid cavity joint, it has been direct ligament attachment between the femoral bone and acetabulum, the ligamentunteres, which fits it’s snully from the forever capitis depression on the femoral heads into their hip joints, helping to stabilize the joint. It is also a heavily walled with muscles, from the trunk of these torso down to the knee, with the gluteal (lateral posterior), Quadriceps (anterior), adductors (medial), and hamstring (posterior) muscle groups acting on the bone at various points.

The femur can easily side as they trochanters is medial and posterior positioned, with the lineal aspire running directly posterior and the adductor tubercle located medially on the medial epicondyle.

Fig. 2.1: Femur bone.

Fracture Issues:

The stiffness and strength of the bone is reduced as until the failure of the first types occurred an under a much lower load. Predicting and preventing bones fracture is an very important topics in orthopedics due to their very high frequency, surgical complications and socio economic impact.

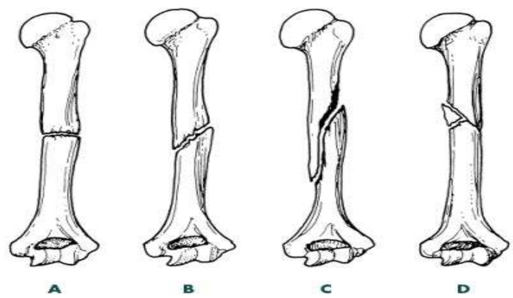


Fig. 2.2: Modified hybrid latch flip-flop.

A fracture is mainly defined as disruption in bone continuity; the degrees of disruptions can vary from the crack in this cortex to it multifragmentation. Fractures are often associated with local tissue trauma. Again, the severity can vary from mild edema to vascular compromise. Fractures are most common, and more people will sustains one across a lifetime.

First, a weak pull up PMOS transistors MP1 with gate connected to the ground is used in the first stages of the TSPC latch. This gives rises to a pseudo nMOS logic style design, and the charge keeper circuits for the internal node *X* can be saved. In an addition to their circuit simplicity, this mainly approach also reduces the load capacitance of node. Second, a pass transistor MNx controlled by their pulse clock is includes so that input data can drives node Q of the latch directly (the signal feed through scheme). Along with the pull up transistor MP2 at the second stage inverter of the TSPC latch, this extra passage facilitates auxiliary signal driving from the input source to node Q.

The node level be has been quickly pulled up to shorten the data transition delay. Third, the pull-down network of the second stage inverter is completely removed. Instead, of the newly employed pass transistor MNx provides a discharging path. The role played by MNx is thus two folded, i.e., providing the extra driving to node Q during 0 to 1 data transitions, and discharging node Q during "1" to "0" data transitions. Compared with these latch structures used in SCDFF design, the circuit savings of these proposed design includes a charge keeper (two inverters), a pull down network (two nMOS transistors), and a control inverter. The only extra components will introduced by an nMOS pass transistor to support signal feed through.

Finite Element Analysis:

Finite Element Method is widely accepted as a power tool for biomechanics modelling. Finite Element Analysis is a useful tool for the analysis of implants in fractures of femur. Such a study not only help one to understand the relation between the form and function , but also be an important input for the surgeon during this work.

The ultimate purposes of Finite Element Analysis is to mathamatically recreate the behaviour of an actual engineering system. In other words, an analysis must be a mathamatical model of physical prototype. In the broadest sensing model comprises, boundary conditions, real constant, all nodes, elements, meterial properties, and other features that are used to represent the physical system.

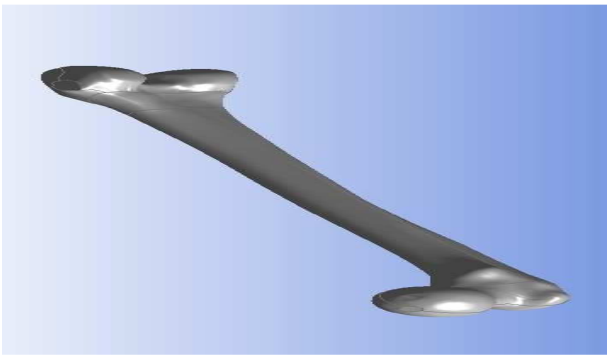


Fig. 3.1: The Whole Femur Bone 3D Image.

The objective of the study method is mainly evolves a comprehensive FE models of the bone using a 3D geometry of skeletal components of the bony structure for bone image and it's to evaluate the deformations, stress and strains acting on the 3D models of the human foot is using ANSYS software.

Modal Analysis of Bone Under Age Of 5:

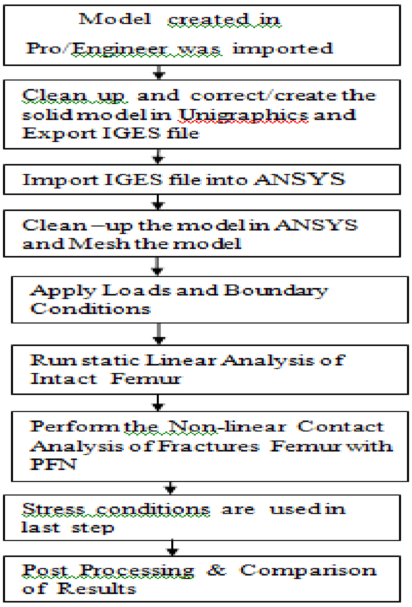
The bar chart is mainly plotted by taking the various numbers of age groups in X-axis and both the Von Mises stress conditions and strains in y axis. Here age groups are set as 30, 31-50, and 50-70 and loading condition put 200kg, 100kg, and 70kg.

Conclusion:

Bone, is a biological tissue, possesses a very complex hierarchical structure. From a mechanical point of view, it is transversely isotropic with a higher modulus in the longitudinal direction. It is asymmetric with higher strength in compression than in tension and shear. The properties of bone tissue are even more complex. Various theories have been proposed including stress or strain-base criteria, von Mises.

What we implemented in this study is a very simplified version of bone material properties. Our goal was to demonstrate the ANSYS 4.0 technology and how it may be used to study bone fracture properties. In ANSYS 4.0 software we implemented the material properties is deformation , Von-Mises stress and strain Equivalent condition, and linear blucking, model analysis is calculated. In future studies we will demonstrate how composite failure criteria may be implemented using this user subroutine to study bone fracture.

Flow Chart:



Simulation Result:

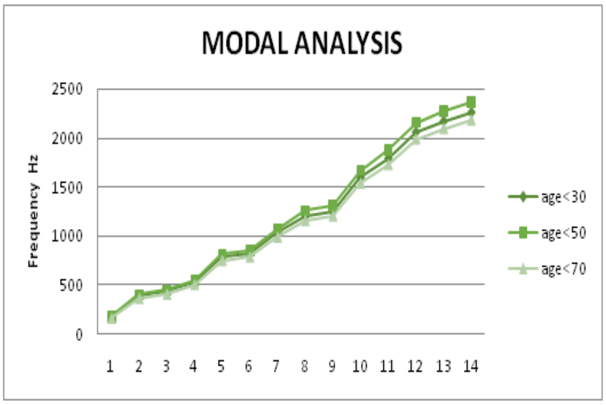
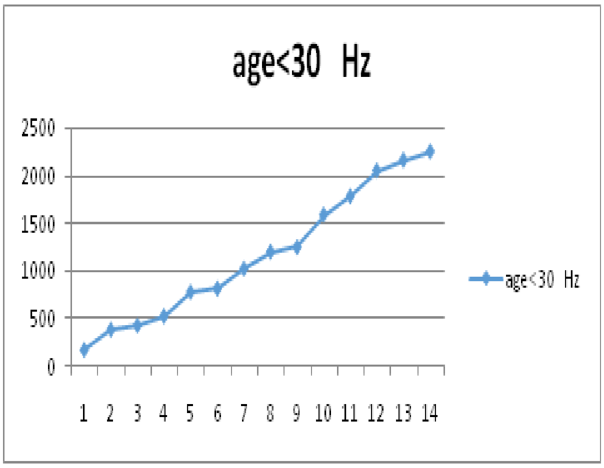
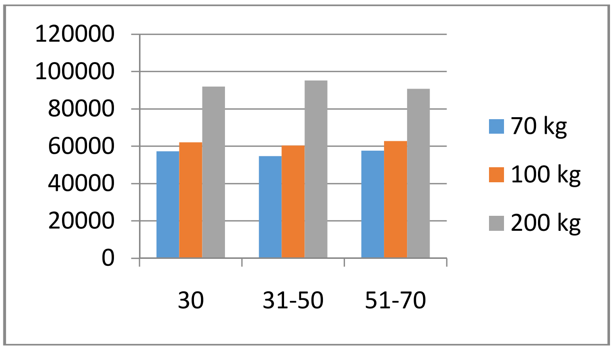


Fig. 4.1: Simulation Result.





REFERENCES

Ashwain Kumar, Himan Shu jaiswal, 2014. " Free Vibration Modes Analysis of Femur Bone Fractures Using Varing Boundary Conditions based on FEA," Procedia Materials science, 6: 1593-1594.

Amalraju, D., A.K. Shaik Dawood, 2012. " Mechanical Strength Evaluation Analysis of Stainless steel and Titanium Locking Plate for Femur bone fracture" An International Journal (ESTIJ), 2-3.

Chawla, A. and S.S. Mukherjee, "Finite element meshing of human bones from MRI/CT Raw Data".

David, P., Beason, J. Greg, Dakin, R. Robert, E. Lopez, Jorge, Alonso, 2003. "Bone mineral density correlates with fractures load in experimental side impacts of the pelvis" Journal of Biomechanics, 36: 219-227.

Dave W., Chen, Chun-Li Lin, 2013. " Biomechanical consideration of total hip arthroplasty following failed fixation of femoral intertrochanteric fractures- A finite element analysis" Elsevier Peleg *et al.*/Medical Engineering Physics 35: 569-575.

Enrico Schileo, Fulvia Taddei, 2008. " Subject-specific finite element models implementing a maximum principal strain criterion are able to estimate failure risk and fracture location on human femurs tested in vitro " Journal of Biomechanics, 41: 356-366.

Eran Peleg, Maarten Beek, Leo Joskowicz, 2010. " Patient specific quantitative analysis of fracture fixation in the proximal femur implementing principle strain ratios. Method and experimental validation" Elsevier Peleg *et al.*/ Journal of Biomechanics, 43: 2684-2688.

Fulvia Taddei, Luca Cristofolini, 2006. " Subject – Specific finite element models of long bones: An in vitro evaluation of the Overall accuracy ," Journal of Biomechanics, 39: 2457 – 2467.

Kaman, M.O. and N. Celik, 2014. " Numerical stress analysis of the plates used to treat the tibia bone fracture" Journal of Applied Mathematics and Physics, 2: 304-309.

Muhammad Shahzad Masood, Atique Ahmad, Rizwan Alim Muffi, 2013. "Unconventional Modeling and Stress Analysis of Femur Bone under different boundary conditions" International Journal of Scientific & Engineering," 4-12. ISSN 2229-5518.

Oess Ninja, P., Bernhard. Weisse, J. Bradley, 2009. "Magneto elastic Strain Sensor for Optimized Assessment of Bone Fracture Fixation" IEEE SENSORS Journal, 9-8.

Raji Nareliya and Veerendra Kumar, 2011. " Biomechanical Analysis of Human Femur Bone" International Journal of Engineering Science and Technology (IJEST), 3-4.

Ramana, S.V. and P. Govinda Rao, 2013. " Modelling and static analysis of femur bone by using different implant materials" Journal of Mechanical and Civil Engineering, 7(4): 82-91.

Sandeep Das and Saroj Kumar, " Finite element analysis of femur fracture fixation plates" International Journal of Basics and Applied Biology .

Saqib Noor, Y.T., Feng, 2013. "Finite Element Analysis Modelling of Proximal femoral Fractures ,including post – fixation Periprosthetic ", Injury, Int, J, Care Injured, 44: 791-795.

Stulpner, M.A., B.D. Reddy, G.R. Starke, A. Spirakis, 1997. "A Three Dimensional Finite Analysis of Adaptive Remodelling in The Proximal Femur", J. Biomechanics, 30(10): 1063-1066.

Todd, L., Bredbenner, Ryan S. Potter, 2010. " Statistical shape modeling describes variation in tibia and femur surface geometry between control and Incidence groups from the Osteoarthritis Initiative database," Journal of Biomechanics, 43: 1780-1786.

Tomasz, Topolinski, Adam Mazurkiewicz, 2012. "Microarchitecture Parameters Describe Bone Structure and Its Strength Better than BMD" The Scientific World Journal, Article ID 502781, 7. doi:10.1100/2012/502781.

Turner, A.W.L., R.M. Gillies, R. Sekel, P. Morris, 2005. "Computational bone remodeling simulations and comparisons with DEXA results" Journal of Orthopaedic Research, 23: 705-712.

ZHAO Jun-hai, MA Shu – fang,2009. " Finite Element Analysis of femur stress under bending moment and compression load" Biomedical Engineering and Informatics. BMEI '09. 2nd International Conference on 10.1109/BMEI.2009.5305316.