International Journal of Progressive Research in Science and Engineering
Volume-1, Issue-2, May-2020
www.ijprse.com

Finite Element Analysis of Stress in Femur Bone

Thasneem Fathima

Biomedical Signal Processing and Instrumentation, M.Tech, RV College of Engineering, Karnataka, India

Corresponding Mail:*thasneemfathima.95@gmail.com*

Abstract Finite element method (FEM) is a technique of solution of the boundary value problems. It can be explained as a numerical method for solving differential and integral equations. FEA is used to analyse the human femur bone. FEA analysis is performed on model of femur bone by varying the loads. The maximum stresses generated in this analysis. The failure stress is taken as 100 MPa. The Result indicate the failure of femur bone under the loading of certain weight under axial loading and bending load. That will clearly indicate the strength of femur bone in axial direction & during bending while average is taken. The study assumed different weights for loading similar to that of base paper 25Kg, 35Kg, 50Kg, 69Kg, 150Kg, 250Kg, 414Kg, & 500Kg. I found as weight is increasing there is more of equivalent stress generated on the bone its average is compared with the results of literature survey. Axial strength of femur is almost six times than bending. Human femur can withstand ten times the load of its body weight. In Osteoporosis when BMD decreases to the fracture threshold, an increase in the strains up to four times is the main reason of permanent failure in the joint. As per literature we can also use this study of Finite element analysis method to check the fragility of bone. So that detection of Osteoporosis might be possible easily for osteoporotic bone in future.

Key Words—Finite Element Method, Osteoporosis, Bone, ANSYS, Fragility.

I. INTRODUCTION

FINITE ELEMENT METHOD: Finite element method (FEM) is a technique of solution of the boundary value problems. It can be explained as a numerical method for solving differential and integral equations. Finite element analysis (FEA) is the practical application of FEM. FEA is a computational tool for carrying out engineering analysis. It can be used for analysis of new product designs as well as for the existing designs using the equations of mechanics of materials. In this Seminar, FEA is used to analyze the human femur bone in ANSYS Software.

FEMUR BONE: Femur bone/thigh bone. It's the longest, heaviest and strongest bone in the human body. The length of this bone is almost 26% of the height of person. Femur bone is divided into three parts: upper extremity, body and lower extremity. Upper part consists of head, neck and the tow trochanters. Body is the long and almost cylindrical in shape. It is slightly arched. Lower extremity is bigger than upper extremity. It is slightly cuboid in form but its diagonal diameter is bigger than its anteroposterior.

FEA ON FEMUR BONE: Finite element analysis (FEA) is very powerful technique to the stress analysis of non-homogeneous and nonlinear biological systems. FEA is one of the common techniques to examine the structural stresses developed in engineering mechanics. It has been used in many engineering applications including the orthopedic biomechanics, to calculate the stresses in human bones & also Finite Element method used for mechanical analysis of osteoporosis hip joint [3].

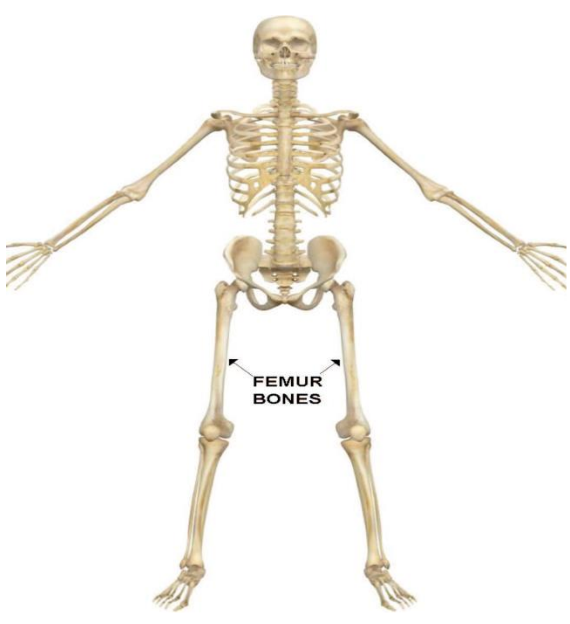

Fig.1. Human skeleton where femur bones are marked. Courtesy [2]

FEA helps in identifying the zones of high stresses and assist in implants design. The CAD model can be used to build FEM model (mesh of nodes and elements for analysis).

International Journal of Progressive Research in Science and Engineering
Volume-1, Issue-2, May-2020
www.ijprse.com

FEA analysis can be repeatedly performed on this model with different set of loading conditions and material properties. It is essential to use the correct material properties and geometric size to simulate the mechanical behavior of wide range of bone quality and size. Two different kinds of loading conditions are applied on the femur bone model; in axial direction (parallel to bone) and in bending direction (normal to the bone).Aim is to create a simulation model that can demonstrate the stresses and strains which may happen on real bone. The accuracy of model is verified and compared with results available in literature of [2].

II. Literature Survey

Topic	Methods
Finite Element analysis of human fractured femur bone implantation with PMMA thermoplastic prosthetic plate [1]	Finite Element analysis to identify max & min stresses on femur bone having no crack & loading conditions & Fixing PMMA plate using ANSYS
Finite element analysis of human femur bone [2]	The FEM model was built using solid tetrahedral element (20-noded 186 structural solid, ANSYS®). The model was analyzed for its sensitivity using loads
Stress Analysis for Osteoporosis Head of Femur [3]	Finite Element method for mechanical analysis of osteoporosis hip joint using ANSYS 12.1 simulation environment.
A review on application of finite element modelling in bone biomechanics [4]	The geometry of metacarpal bone was developed on mimics& the solid model was analyzed on ANSYS 10.0.
Configuration on hybrid planting to improve internal fixation on femur bone model [5]	The bone screw mounting configurations have successfully modeled with ANSYS Multiphysics/LS-Dyna v.18.1

In Finite Element analysis of human fractured femur bone implantation with PMMA thermoplastic prosthetic plate [1] femur bone is the longest & shortest bone in the body, this bone is contained a linear elastic, isotropic & homogenous material of calcium phosphate. It needs to support maximum weight of the body in between hip & knee joint during static loading condition. Bone fracture is one the common trauma. One method of rectifying study is of the finite element analysis of femur bone fixation with polymethyl methacrylate thermoplastic prosthetic plate at mid-shaft position in static loading condition. Result analysis based on the stents. To prove that PMMA is best suitable, compared the minimum principal stress with respect of other biomaterials.

In Finite element analysis of human femur bone [2] an effort is made to analyses the stresses experienced by the human femur. In order to achieve these results a CAD model was developed by using the 3-D scanning of generic human femur for an individual of 70 kg weight (approx. averaged adult weight). The marrow cavity has been approximated as a hollow cylinder. The FEM model was built using solid tetrahedral element (20-noded 186 structural solid, ANSYS®). The model was analyzed for its sensitivity. The results were computed for the range of loads. In this analysis, the maximum stress and its location were noted. In addition, the critical value of load was estimated for ultimate failure (i.e. fracture). The evaluated results give an understanding of the natural safety factor. The presented results are of significant importance in replication of the natural design parameters in creating the synthetic bone substitutes.

In Stress Analysis for Osteoporosis Head of Femur [3] Human hip joint diseases, such as osteoporosis, osteonecrosis, osteolysis, and osteoarthritis, induce pain and loss of mobility to millions of people around the world. Stress analysis is useful for identifying the effects of the abnormality on joint functions. Finite element analysis (FEA) is very powerful technique to the stress analysis of non-homogeneous and nonlinear biological systems. In this study osteoporosis was simulated using finite element analysis in ANSYS 12.1 simulation environment. Osteoporosis cause loss of bone mineral density (BMD). Thus its effects are on bone physical and mechanical properties must be known since it can cause joint dysfunctions. In this study stresses, strains, and equivalent von-mises stresses were estimated in the head of femur during static stance for range of different bone mineral densities (1.32 – 0.4 g/cm3) to simulate the cases of normal healthy adult, aging, osteopenia and osteoporosis, from the results a curve for BMD vs. Strain has been sketched showing an increase up to four times in strains for the case of osteoporosis. The study assumed orthotropic behavior of bone, hence Young and Shear moduli decreased with decreasing BMD this case very large increase in strains and slight increase in tensile and compressive stress of the femur neck.

In A review on application of finite element modelling in bone biomechanics [4] the finite element modelling has been developed as an effective tool for modelling and simulation of the biomedical engineering system. Finite element modelling (FEM) is a computational technique which can be used to solve the biomedical engineering problems based on the theories of continuum mechanics. This paper presents the state of art review on finite element modelling application in the four areas of bone biomechanics, i.e., analysis of stress and strain, determination of mechanical properties, fracture fixation design (implants), and fracture load prediction. The aim of this review is to provide a comprehensive detail about the development in the area of application of FEM in bone biomechanics during the last decades. It will help the researchers and the clinicians alike for the better treatment of patient and future development of new fixation designs.

In Configuration on hybrid planting to improve internal fixation on femur bone model [5] study presents a stress analysis of hybrid plating constructions on femur fracture. The bone screw mounting configurations have successfully modeled with ANSYS Multiphysics/LS-Dyna v.18.1. The bone model was achieved from the CT-scanning of the human femur bone. The interactions between femur bone and hybrid plating were observed. Locking compression plates with 8 holes for bone screws were used. Applied axial compression load has developed stress distribution at all segments. Locking screws endured the bending forces and generated bending moment. Stress concentrations were noticeable at the screws neck. No locking screws have produced a lower stress but bearing to be loosed since has small angular rigidity due to unthreaded screw heads. The most stable bone screw configuration was model A with N-L-N-L L-N-L-N pattern. The alternating sequence of screw configuration resulted in lower stress distribution at all segments, has small screw displacements and enduring lowest stress at each segment, especially femur bone.

III. METHODOLOGY

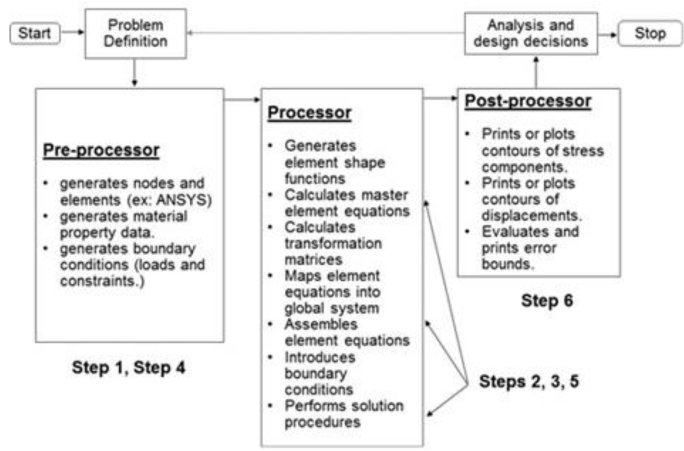

Fig.2. Methodology Diagram of typical FEA steps. Courtesy [3]

By using ANSYS 19.2 finite element analysis of the bone is done for stress detection during bending and axial and average stress is tabulated for checking future fracture risk because of loading weight on bone it varies for different bone depending on age, obesity etc.

IV. MATERIAL PROPERTIES

Segmentation and Geometry:

CT scan Dicom image a 3D model of Femur bone is considered & studied using ANSYS s/w

1. *Bone Orthotropic Properties:*

Material properties of human femur vary between subjects therefore and it is difficult to assign any particular material properties. By nature the behavior of the bones are anisotropic, still for simplifying purpose some researchers assume that the behavior is isotropic for stimulation Linear static analysis was performed and physiological conditions (role of muscles in sharing the load) were ignored.

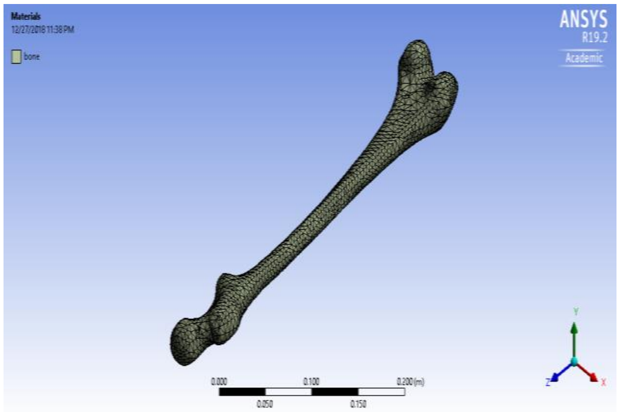

Fig.3. 3D model of Femur bone

2. *Meshing and boundary Conditions*:

The number of elements used is 8696 (Tetrahedral) while the number of nodes is 17797. The hip reaction force components are -405, -246, and +1719 N in X, Y and Z respectively acting on the femoral component head center. The model assumed to be fixed from the bottom.

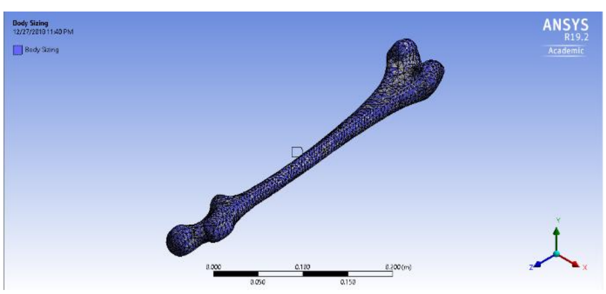

Fig.4. FEA mesh of femur bone

3. *Loading:*

Two different kind of loading conditions were applied to simulate real case scenarios. In first case axial loading (compression) is applied in direction of the bone. This case simulates the weight handled by femur in upright standing position. In second case bending load (perpendicular) is applied to femur bone. In both cases boundary constraint was applied on the other end of the femur bone.

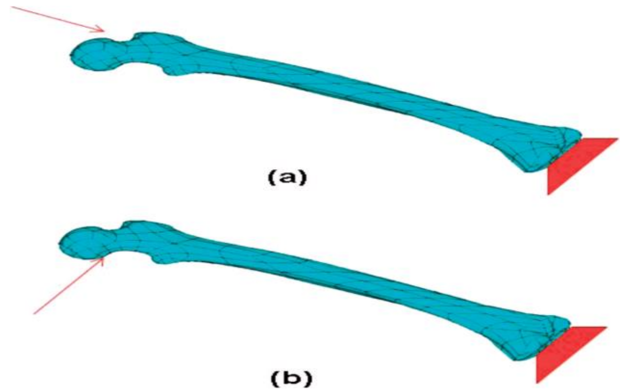

Fig.5. (a) Axial loading condition with boundary constraint on other end.
(b) Bending loading condition with boundary constraint on the other end courtesy [2]

V. Results and Discussion

FEA analysis is performed on model of femur bone by varying the loads. The maximum stresses generated in this analysis. The failure stress is taken as 100 MPa based on experimental data [2]. The results indicate the failure of femur bone under the loading of certain weight under axial loading and certain at load under the bending load. That will clearly indicate the strength of femur bone in axial direction & during bending the study assumed orthotropic behavior of bone, hence Young and Shear moduli decreased with decreasing BMD that leads to very large increase in strains and slight increase in tensile and compressive stress of the femur [3]

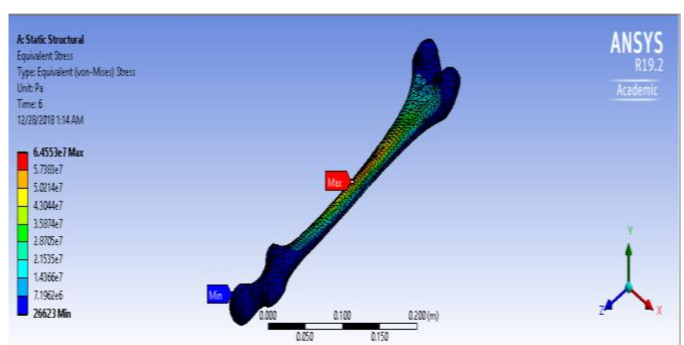

Fig.6. FEA of 25Kg Equivalent Stress

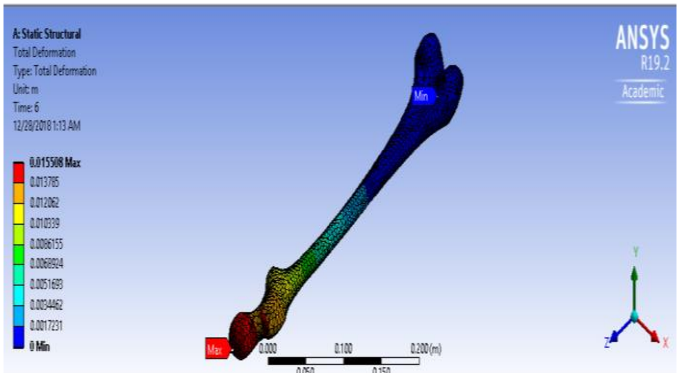

Fig.7. FEA of 25 Kg Deformation

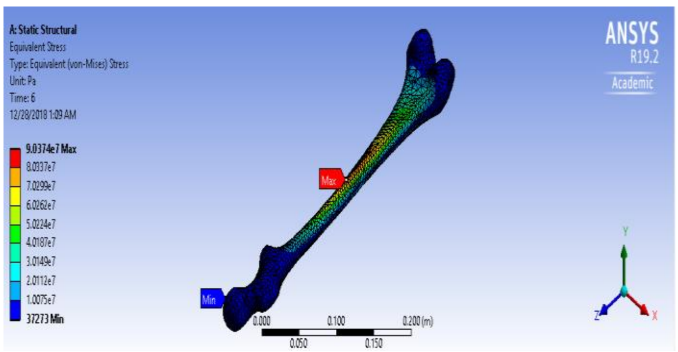

Fig.8. FEA of 35 Kg Equivalent stress

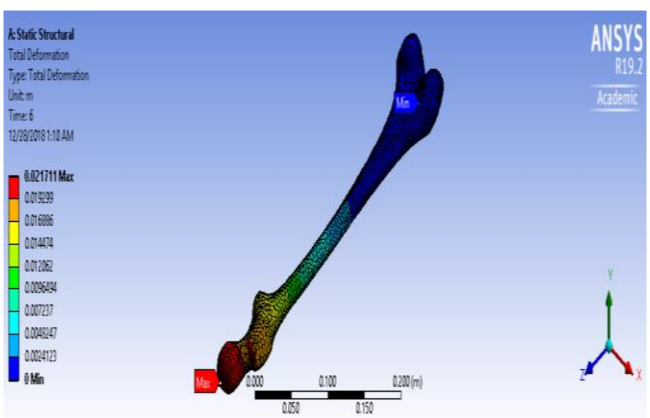

Fig.9. FEA of 35 Kg Deformation

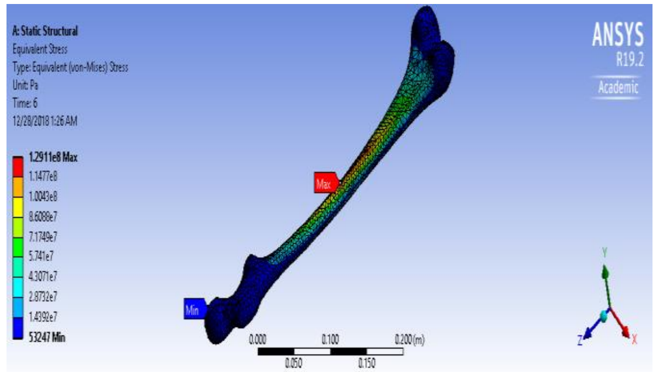

Fig.10. FEA of 50 Kg Equivalent stress

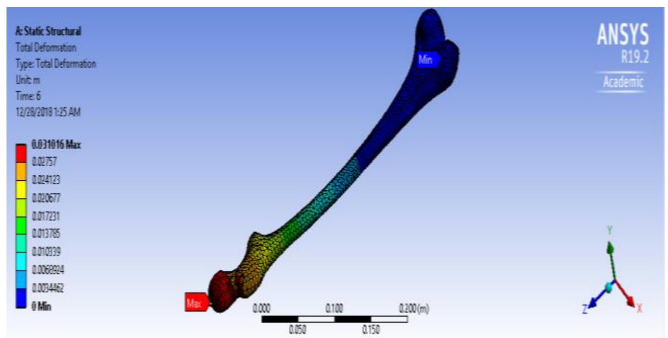

Fig.11. FEA of 50 Kg Deformation

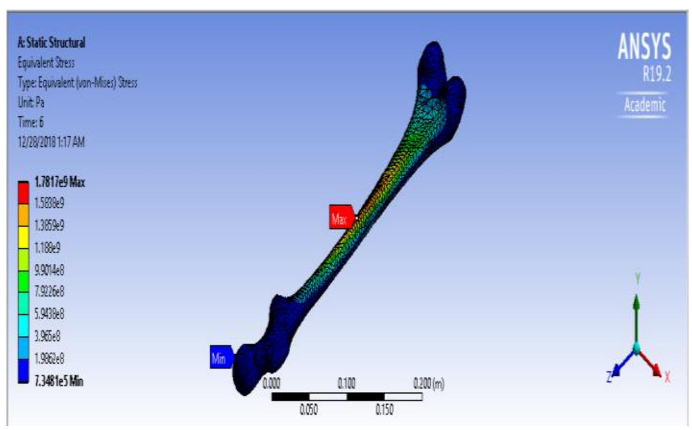

Fig.12. FEA of 69 Kg Equivalent stress

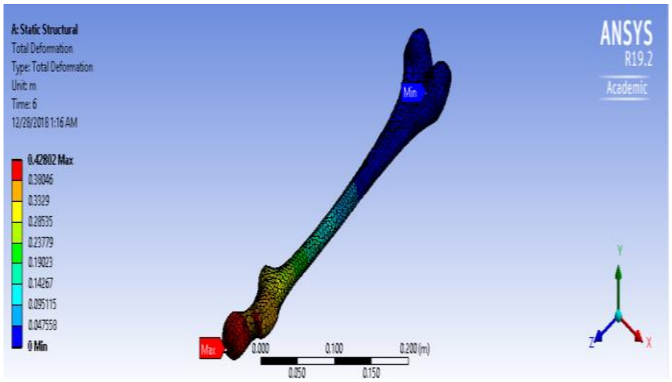

Fig.13. FEA of 69 Kg Deformation

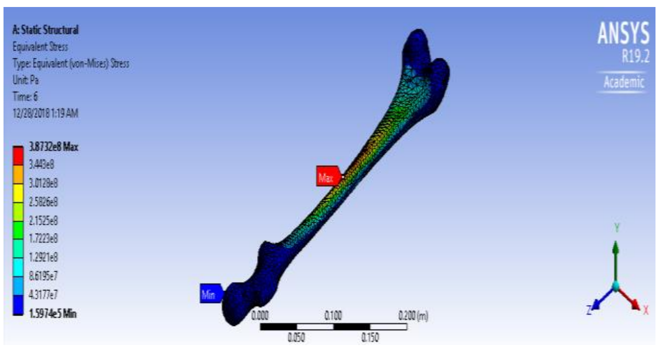

Fig.14. FEA of 150 Kg Equivalent stress

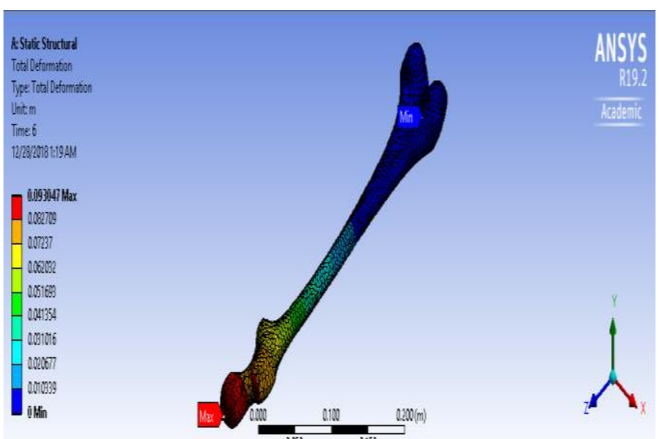

Fig.15. FEA of 150 Kg Deformation

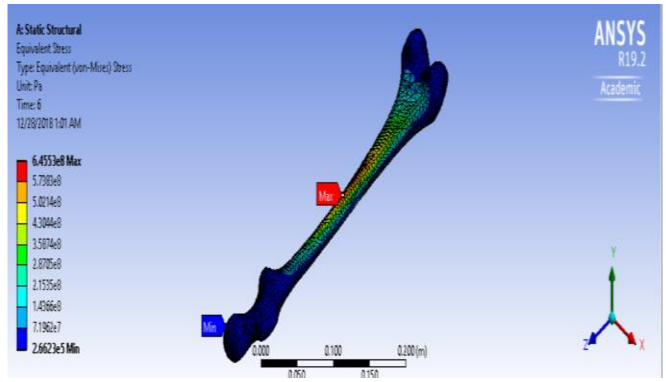

Fig.16. FEA of 250 Kg Equivalent stress

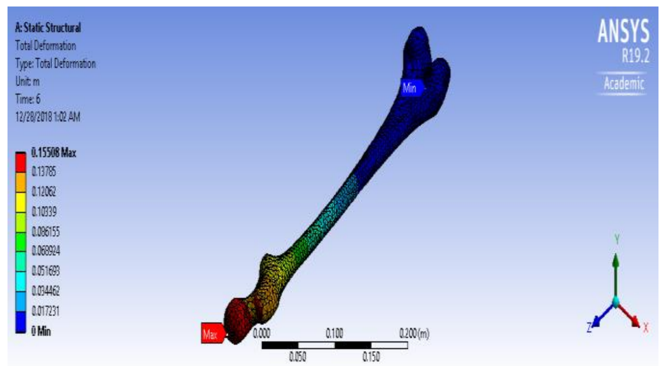

Fig.17. FEA of 250 Kg Deformation

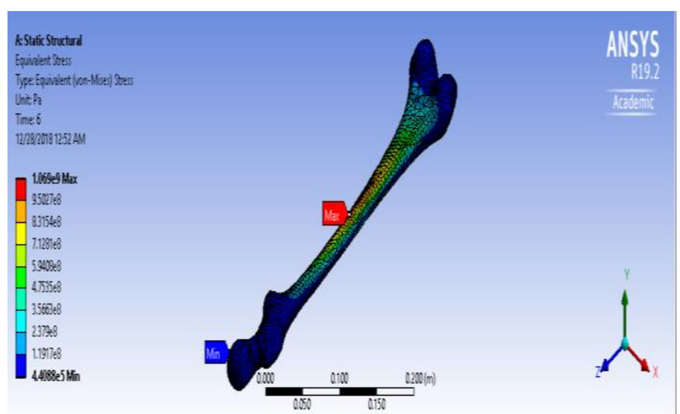

Fig.18. FEA of 414 Kg Equivalent stress

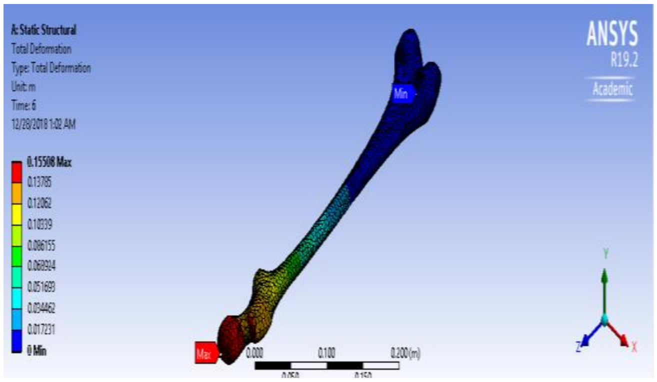

Fig.19. FEA of 414 Kg Deformation

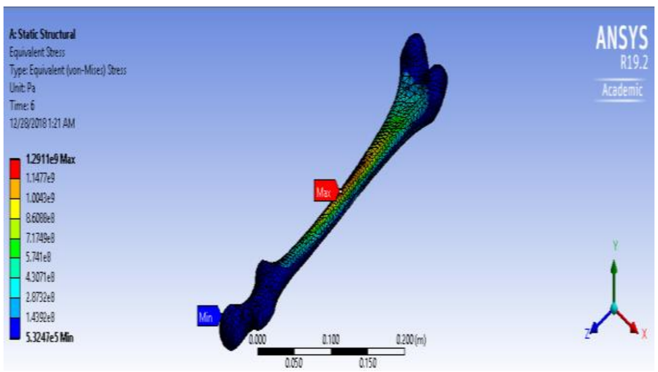

Fig.20. FEA of 500 Kg Equivalent stress

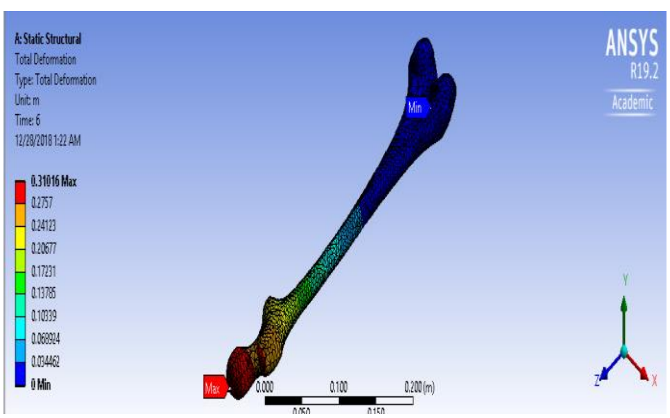

Fig.21. FEA of 500 Kg Deformation

VI. Conclusion

Following conclusions can be drawn from FEM analysis of human femur bone: Axial strength of femur is almost six times than bending. Human femur can withstand ten times the load of its body weight. In Osteoporosis when BMD decreases to the fracture threshold, an increase in the strains up to four times is the main reason of permanent failure in the joint.

Loading Weight (Kg)	Expected Stress (MPa)- [2]	Equivalent Stress Average (MPa)
25	6.04	10.70
35	9.31	14.75
50	24.23	21.70
69	30.21	29.00
150	36.41	43.20
250	60.01	63..3
414	99.83*	93.73*
500	121.12*	210.0*

*indicates that as we have high stress nearing to 100 based on literature [2] as we know if the stress is more it's the sign for bone tending to fracture risk hence this study is to study FEA of femur bone which is at a risk when its loading weight is increasing based on person's body weight.

We have considered few weights above for study depending on the study we found as in weight increasing we can there is more of equivalent stress generated on the bone its average is compared with the literature survey [2] we found approximately few readings same as shown in table 1,

As per literature we can also use this study of Finite element analysis method to check the fragility of bone. So that detection of Osteoporosis might be possible easily

REFERENCES

[1]. Ajay, Manish, “Finite Element analysis of human fractured femur bone implantation with PMMA thermoplastic prosthetic plate”, 11th International Symposium on plasticity & impact machines, implant 2016, Elsevier 2017.

[2]. Uzair N. Mughal, Hassan A. Khawaja and M. Moatamedi, “Finite element analysis of human femur bone”, the international journal of multiphasic 2015, volume 9, number 2, 2015.

[3]. Prof. Dr.Albert Elia Yousif, Eng.Aqeel Abdulkhaliq Abdulhadi, “Stress Analysis for Osteoporosis Head of Femur Finite Element method for mechanical analysis of osteoporosis hip joint”, The First National Conference for Engineering Sciences FNCES'12 / November 7-8, 2012.

[4]. Sandeep Kumar Parashar, Jai Kumar Sharma, “A review on application of finite element modelling in bone biomechanics”, Department of Mechanical Engineering, Rajasthan Technical University, Kota 324010, India Received 8 February 2016.

[5]. Nafisah arina hidayati, et. al, “Configuration on hybrid planting to improve internal fixation on femur bone model”, Brawijaya University, Mechanical Engineering Department, Malang, Indonesia.