

Finite Element Analysis of Femur Fracture Fixation Plates

Sandeep Das¹ and Saroj Kumar Sarangi²

¹Dept. of Mechanical Engineering, Biju Pattnaik University of Technology, Bhubaneswar, Odisha

²Dept. of Mechanical Engineering, Siksha 'O' Anusandhan University, Bhubaneswar, Odisha

¹sandeep_625@yahoo.co.in, ²sarojksarangi@gmail.com

Abstract: Biomechanics is the application of mechanical principles on the living organisms and utilizing the principles of physics, simulation and study of biomechanical structures are carried out. Finite Element Method is one of the widely accepted tools for modelling the biomechanical structures. The femur bone is the most proximal bone of the leg in vertebrates capable of walking and jumping. This paper presents the analysis of Femur bone fracture fixation plates using Finite element method. The Femur bone is modelled using Mimics software and analysis is carried out in an ANSYS environment. The fracture fixation plate is modelled using the commercially available Solidworks CAD software. The stress distribution at the fractured site of the femur is obtained when the system is subjected to torsional as well as compressive loadings along with various healing stages. The effects of the use of different biomaterials for the plates and screws on the stress distribution characteristics are also investigated.

Keywords: Femur, Fracture, Biomaterials, ANSYS.

1. INTRODUCTION

The femur bone is the most proximal bone of the leg in vertebrates capable of walking and jumping. In human anatomy the femur is the longest and largest bone but strongest in compression only. The femur is responsible to bear the largest percentage of body weight during normal weight bearing activities. Its length is 26% of the person's total height, a ratio that is useful in anthropology, because it offers a basis for a reasonable estimate of a subject's height from an incomplete skeleton. At the top end the femur meshes with the acetabulum to form the hip joint and at its bottom it meshes with tibia bone to form the knee joint. The upper (proximal) end of the femur contains head, neck, two trochanters and adjacent structures. The body of the femur is long, slender and almost cylindrical in structure. The lower (distal) end of the femur is larger than upper end. It consists of two oblong eminences known as condyles.

Trauma is the measure cause of death and disability in both developed and developing countries. Femur bone fracture is one of the common traumas. Femur fractures are an important

subject of investigation in orthopaedic trauma because they are the strongest, longest and heaviest bones in the human body. Injury to normal healthy femur is often caused by high energy phenomena such as sporting accident, motor vehicle accident or fall.

In general when a diaphyseal fracture of a femur bone occurs, prosthetic device such as bone plates are placed across the fracture line on the lateral femur surface with bicortical screws in situation where casts cannot be applied to the affected area. The goal of this study was to examine the biomechanical feasibility of simple bone plate of different biomaterial for providing an effective environment for callus generation. A simply shaped bone plates with different material properties are introduced to enhance the biomechanical function of the plates. Femur bone with horizontal and oblique crack at the cortical zone is modelled and bone plate is placed across the fracture line with the screws. Then the model was evaluated using finite element analysis to find out the best suitable material of the implant.

2. FINITE ELEMENT MODELLING

2.1 Cad Models

Computer tomography (CT) scans were performed in every 0.5 mm along the length of the femur. Then CT scan were stocked in the DICOM format and imported into Mimics software in order to generate the 3D model of the femur. Then the 3D model of femur was exported to Solidworks CAD software. The 3D model had no intramedullary canal. The canal was created by cutting away bone using the cut function in Solidworks. Then horizontal and oblique crack were created by using the same cut function in Solidworks as shown in Fig 1.

The CAD software was also used to create the solid models of the implant like bone plate and screws based on Vernier calliper measurements of the geometry as shown in Fig 2. Then the femur-bone plate system was created by assembling the individual components of the femur, bone plate and

3.2 Boundary conditions.

Bonded contact with no slip was modelled for bone to bone interfaces. This simulated full bony growth around the femur fracture interface. Bone plates are fastened to bone surfaces by screw. So contact stresses are generated in both of the contacting bodies. To calculate the contact stresses, frictional contact elements at the interface between the implant and the bone surface were used in finite element model. The frictional coefficient of ($\mu=0.2$) was taken.

To validate the femur model with [2] a compressive load of 3000 N, representing 4 times body weight, for a 75 kg person was taken. i.e. 3000N by the vertical displacement of force application point on the superior aspect of the femoral head. Because of the FE model linearity assumption, this approach was equivalent to using the slope of the force-vs.-displacement graph.

In the horizontal and oblique crack model a compressive load of 700 N and torsional load of 50 N was taken at the femoral head and a fixed support was provided at lateral condyle, medial condyle and patellar surface i.e. lower surface.

3.3 Validation of femur bone.

It is required to validate the femur bone before analysing the fractured bone. This task proceeded as per [2] which contained stress values at medial and lateral side of human femur. A femur bone of height 415 mm was taken and the stress values in 15 points were analysed and compared with the earlier model as shown in Fig 4. The stress results found slight differences between the earlier femur and our femur model. These differences were caused by different geometry structure of the femur and the finite element quality of the femur model depends on the hardware parameters of used computers.

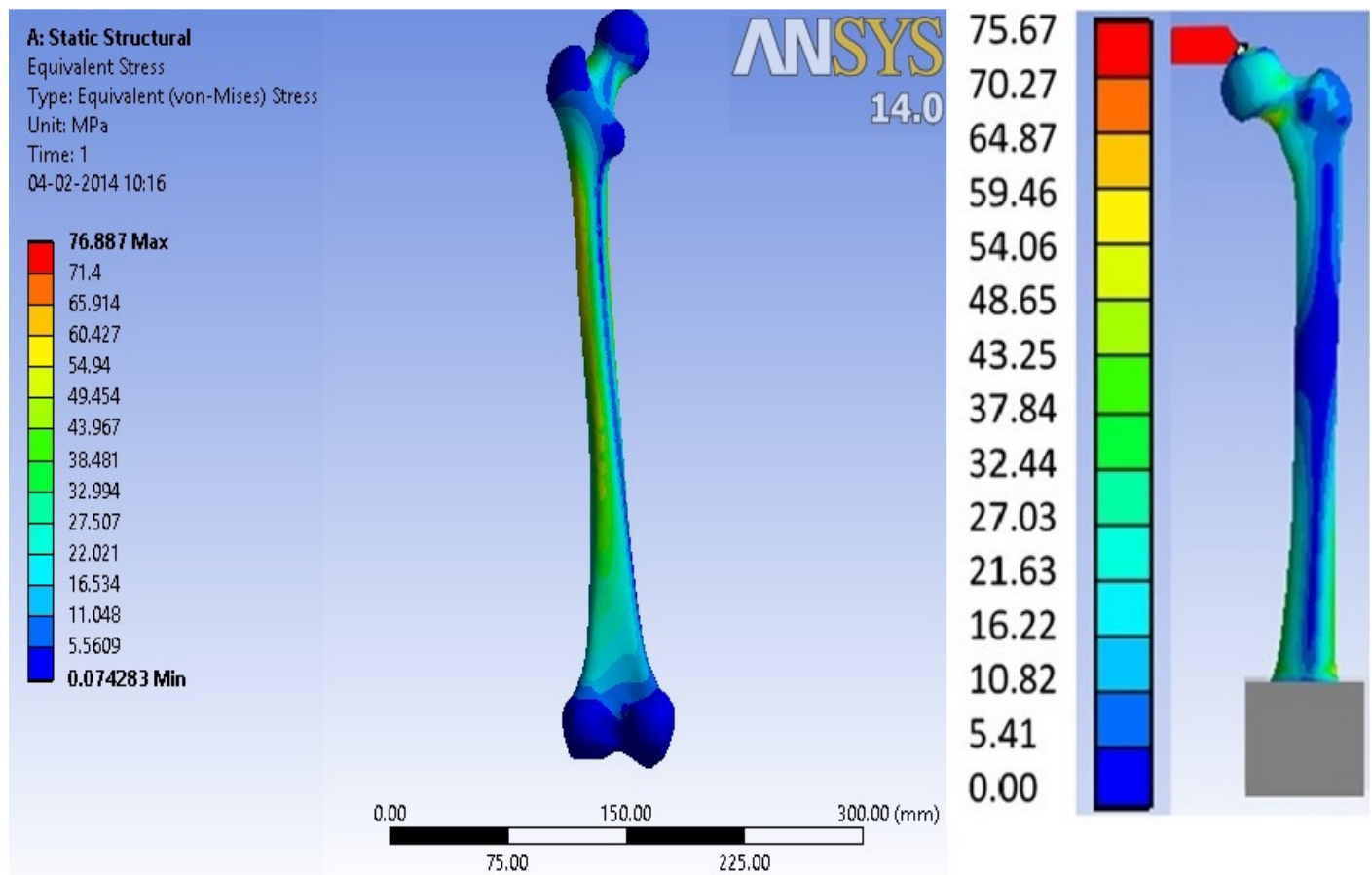


Fig. 4

3.4 Results

This study investigated the stress distribution, deformation and fatigue failure of femur with implant of different material properties for a weight of 70 kg male during normal position. The entire process required a couple of hour. Only static load was applied on the models. The deformation of bone-plate system in compressive and torsional load is shown in Fig 5.

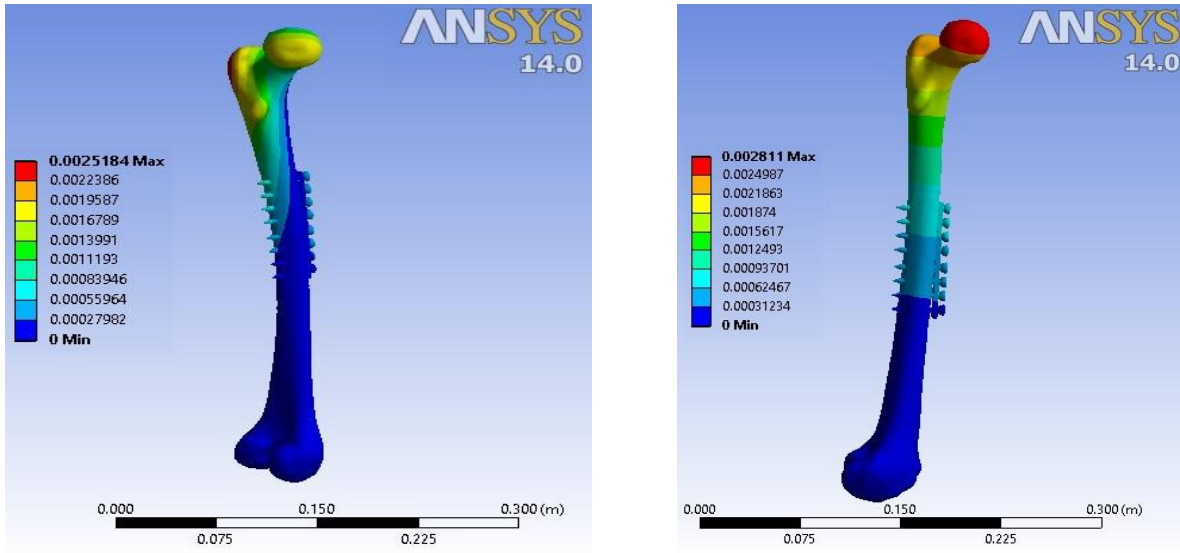


Fig. 5

The effect of bone – plate material on resultant von mises stresses at different loading conditions is indicated in Fig. 6 and Fig. 7. From this figure it is concluded that stresses at the bone plate decreases significantly when using the titanium material instead of stainless steel and cobalt chrome. This is because of flexibility of titanium plate compared to other plates. Additionally, the result indicates that the stresses at the fracture site increases when taking torsional load into consideration.

high even in the titanium plate. When the healing time increases the fractured zone became stronger compared to the initial healing stage and the load can be transferred to the bone. At 50% and 100% healing an additional callus is generated at the fracture interface and the bone became able to take more loads. In this case the bone and the plate behave like composite plate, causing the stress at the fracture interface increases, decreasing the stress on the bone plate.

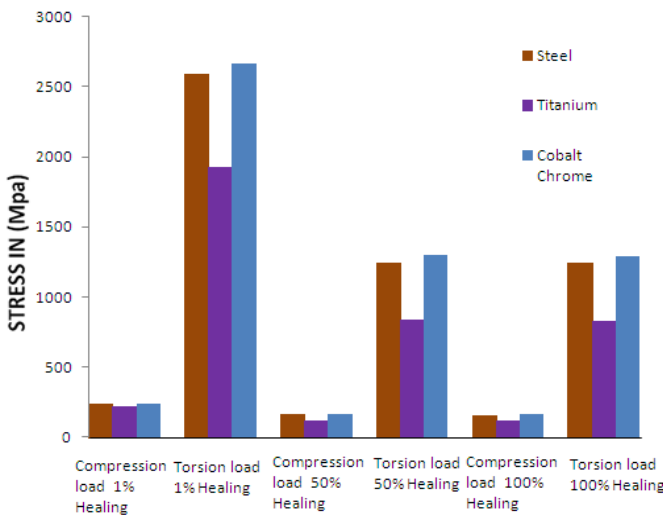


Fig. 6. Von mises stresses in Horizontal fracture

As at the initial stage of healing bone is unable to take load because of immaturity of fractured zone compared to normal bone tissue, so the load bearing section consists only the bone plate. The callus bone cannot carry load due to its low Young's modulus. So the stress at the bone plate is relatively

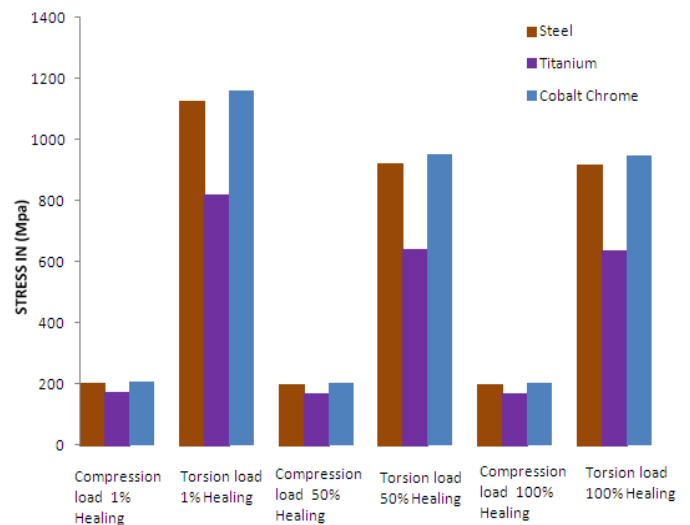


Fig. 7. Von mises stresses in Oblique fracture

Fig. 8 illustrates that how the von mises stress at fracture site varies with different types of loading condition for different types of bone-plates. From the analysis it is found that titanium material is having less von mises stresses compared to other two materials.

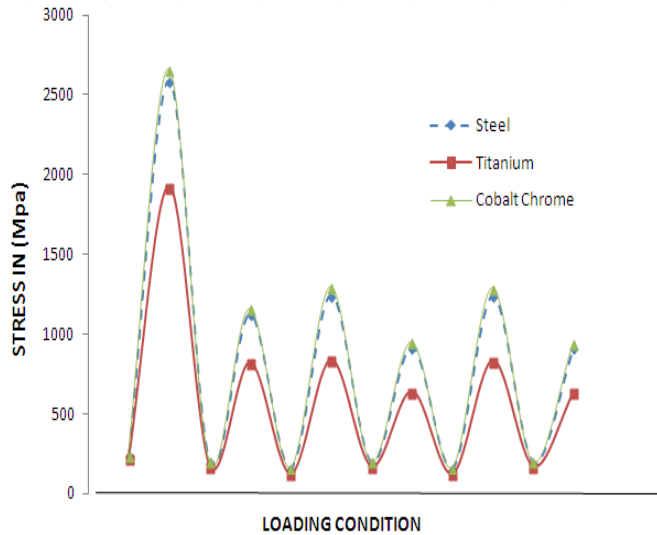


Fig. 8

4. CONCLUSION

In this paper femur bone with simple bone plate was modelled and analysed. Three different materials for fabricating bone plate and screws were selected. Analysis was done in the assembled models. More focus was given on the stresses on the bone plate and screws. In the FE analysis it is found that torsional load has significant effect on the resultant stress in the fracture fixation bone plate system, which should be taken into consideration while design and analysis of bone plate. From the result it can be concluded that titanium material bone plate generates relatively higher stress in the fractured zone compared to other two bone plates. These higher stresses at the callus zone provide better bone healing potential and provide better stability to the fracture fixation. Further work to be done to reduce the contact stress at the fracture site for the additional improvement of the bone plate design.

REFERENCES

- [1] H. Fouad. (2013) Assessment of function-graded materials as fracture fixation bone-plates under combined loading conditions using finite element modelling. *Medical Engineering & Physics*. 33: 456–463
- [2] Hamid Ebrahimi, Mikhail Rabinovich, Viktor Vuleta, Daniel Zalcman, Suraj Shah, Anton Dubov, Kavya Roy, Faisal Sharaf Siddiqui, Emil H. Schemitsch, Habiba Bougherara, Rad Zdero. (2012) Biomechanical properties of an intact, injured, repaired, and healed femur: An experimental and computational study. *Journal of the mechanical behavior of biomedical materials*, 16: 121–135
- [3] Suk-Hun Kim, Seung-Hwan Chang, Dae-Sung Son. (2011) Finite element analysis of the effect of bending stiffness and contact condition of composite bone plates with simple rectangular cross-section on the bio-mechanical behaviour of fractured long bones. *Composites: Part B*, 42: 1731–1738
- [4] Raji Nareliya, Veerendra Kumar. (2011) Biomechanical analysis of human femur bone. *International Journal of Engineering Science and Technology*. 3: 3090-3094.
- [5] Purvi S.D. Patel, Duncan E.T. Shepherd, David W.L. Hukins, (2010) The effect of screw insertion angle and thread type on the pullout strength of bone screws in normal and osteoporotic cancellous bone models. *Medical Engineering & Physics*. 32: 822–828
- [6] Panagiotis E. Chatzistergos, Evangelos A. Magnissalis, Stavros K. Kourkoulis. (2010) A parametric study of cylindrical pedicle screw design implications on the pullout performance using an experimentally validated finite-element model. *Medical Engineering & Physics*. 32: 145–154
- [7] Kuo-Hua Chao, Yu-Shu Lai, Wen-Chuan Chen, Chia-Ming Chang, Colin J. McClean, Chang-Yuan Fan, Chia-Hao Chang, Leou-Chyr Lin, Cheng-Kung Cheng. (2013) Biomechanical analysis of different types of pedicle screw augmentation: A cadaveric and synthetic bone sample study of instrumented vertebral specimens. *Medical Engineering & Physics*. 35: 1506–1512
- [8] P.S.R. Senthil Maharaj, R. Maheswaran, A. Vasanthanathan. (2013) Numerical Analysis of Fractured Femur Bone with Prosthetic Bone Plates. *Procedia Engineering*, 64: 1242 – 1251.